Project 4

Microwave filters

This project is focused on the design of planar microwave filters which are widely used in the hardware of communication systems. The significant difference between ideal LC filters and their planar realization is emphasized.

Design and compare realized microwave filters:

1) using ideal L and C elements,
2) using lumped element and distributed element components.

For the design of the LC filter prototype use, *e.g.*, AWR Wizard iFilter Filter Synthesis.
Suppose available etching technology with an achievable minimal trace width and gap width of 0.1 mm. Also suppose thin film technology with an achievable minimal trace width and gap width of 0.01 mm. Thin film technology can be utilized in exceptional cases only.

Use commercially available substrates. Consider SMA connectors connected to the printed circuit board (PCB). You can choose connectors 32K145-400L5 (central pin diameter 0.92 mm), or 32K243-40ML5 (central pin diameter 0.25 mm), or any other suitable version for PCB mounting.

Analyze $|S_{11}|$ and $|S_{21}|$ of the filters connected to the Z_v = 50 Ω microstrip lines in the frequency band $0.1f_0$ to $3f_0$. Try to achieve good agreement of the properties of both possible versions 1) and 2). Draw the corresponding graphs. Show the final AWR schematic and final layout of all filters, except the high-pass filter. Comment as to why the transmission coefficient of the band-pass and the band-stop filter are periodic and how to improve their performance at high frequencies.

Low-pass filter

Design a low-pass filter connected to Z_v=50 Ω microstrip lines. The following parameters are desired:

$\lvert S_{11}\rvert < -20\ \mathrm{dB}$	for	$f < f_0$
$\lvert S_{21}\rvert > -0.4\ \mathrm{dB}$	for	$f < f_0$
$\lvert S_{21}\rvert < -20\ \mathrm{dB}$	for	$f > 2f_0$

1) Suppose ideal components and the structure:

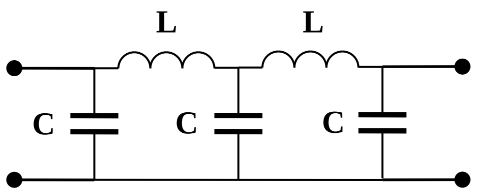

Fig. 1. Low-pass filter with ideal L and C components.

2) Create an analogue circuit using planar lumped element components. Apply the structure:

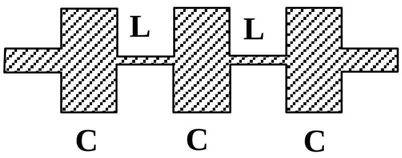

Fig. 2. Low-pass filter with planar lumped element components.

High-pass filter

Design a high-pass filter connected to $Z_v = 50\ \Omega$ microstrip lines. The following parameters are desired:

$|S_{11}| < -20$ dB for $f > f_0$

$|S_{21}| > -0.4$ dB for $f > f_0$

$|S_{21}| < -20$ dB for $f < 0.5 f_0$

1) Suppose ideal components and the structure:

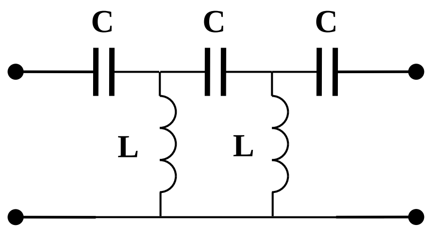

Fig. 3. High-pass filter with ideal L and C components.

2) Create an analogue circuit using planar lumped elements and distributed element microwave components. Apply the structure:

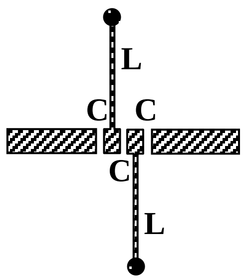

Fig. 4. High-pass filter. Distributed elements L, lumped elements C.

C capacitors in the slots of the microstrip are realized by means of MIM capacitors (ideal).

Band-pass filter

Design a band-pass filter connected to Z_v = 50 Ω microstrip lines. The following parameters are desired:

$\lvert S_{11}\rvert < -10$ dB	for	$0.8 f_0 < f < 1.2 f_0$
$\lvert S_{21}\rvert > -f_0(\mathrm{GHz})/10$ dB	for	$0.8 f_0 < f < 1.2 f_0$
$\lvert S_{21}\rvert < -20$ dB	for	$f < 0.5 f_0$ and $1.5 f_0 < f < 2.5 f_0$

1) Suppose ideal components and the structure:

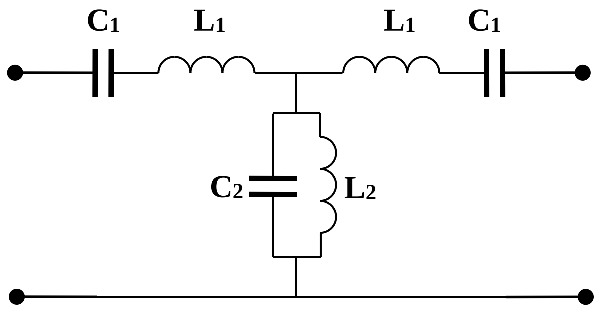

Fig. 5. Band-pass filter with ideal L and C components.

2) Create an analogue circuit using microstrip coupled lines. Apply the structure:

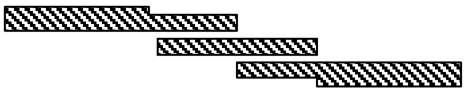

Fig. 6. Band-pass filter with microstrip coupled lines.

Band-stop filter

Design a band-stop filter connected to Z_v = 50 Ω microstrip lines. The following parameters are desired:

$|S_{11}| < -10\ \mathrm{dB}$ for $f < 0.5 f_0$ and $f > 2 f_0$

$|S_{21}| > -0.5\ \mathrm{dB}$ for $f < 0.5 f_0$ and $f > 2 f_0$

$|S_{21}| < -20\ \mathrm{dB}$ for $0.8 f_0 < f < 1.2 f_0$

1) Suppose ideal components and the structure:

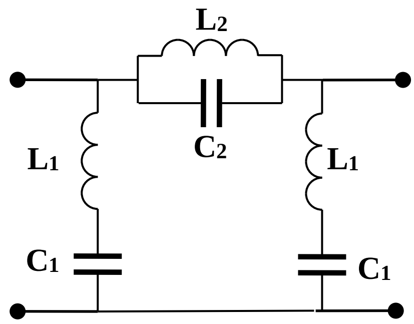

Fig. 7. Band-stop filter with ideal L and C components.

2) Create an analogue circuit using microstrip coupled lines. Apply the structure:

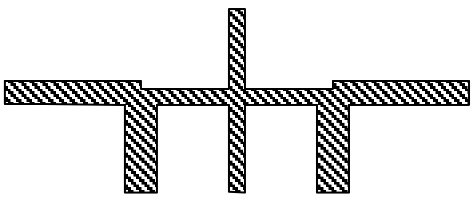

Fig. 8. Band-stop filter with microstrip lines.

Project solution procedure

1) Ideal LC filter prototypes.

Open AWR and set the default project's units to GHz, pF and nH. Set the frequency range of the project to $0.1f_0 \div 3f_0$. In the AWR Project tab, go to the Wizards folder, double-click on iFilter Filter Synthesis and then select Design→Lowpass→Lumped→Lumped Element Filter→OK. The Wizard is shown in Figs. 9 and 10.

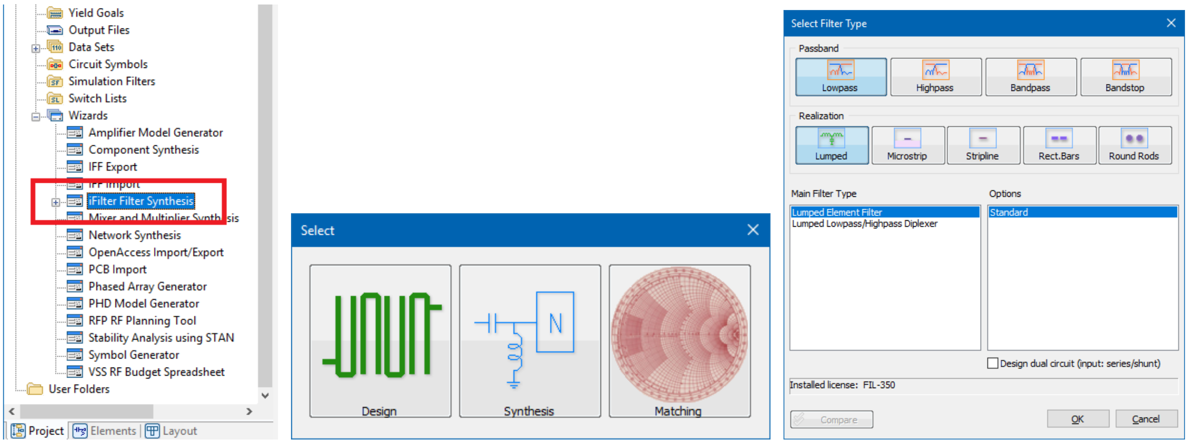

Fig. 9 iFilter Filter Synthesis wizard.

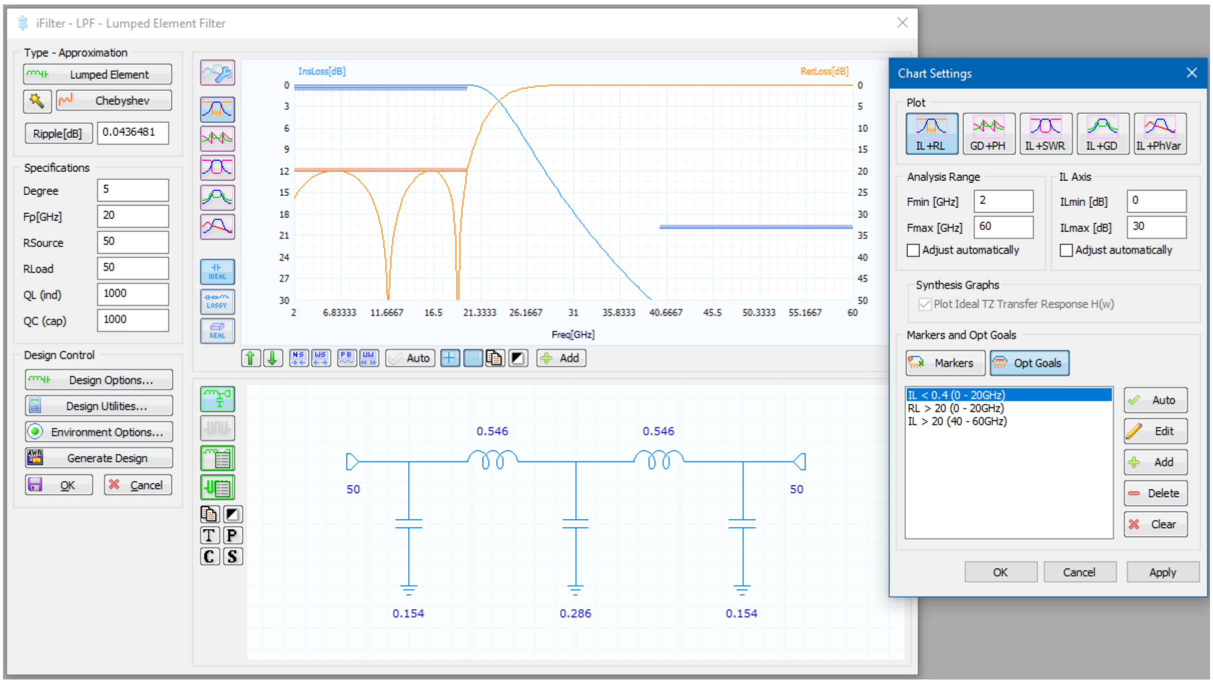

Fig. 10 Low-pass filter design using iFilter Filter Synthesis with L and C elements.

We will show the design of a Low-pass filter with f_0 = 20 GHz, but this design is highly similar to all other types of filters. Sometimes it is necessary to design a dual circuit, *i.e.*, switching between a shunt and series element on the input of the filter. It can be done by clicking on the Lumped Element button and selecting Design dual circuit→OK. Select an approximation function (Chebyshev works fine). The degree of the filter is the wanted number of lumped elements, or resonant circuits, and Fp is the corner frequency (f_0). Clicking on the Ripple[dB] button allows you to specify the Return Loss, which is 10 or 20 dB for different types of filters in this project. The settings of

the graph can be created by clicking on the button where frequency and insertion loss axis range can be set. It is also useful to define the markers or optimization goals to be checked if the required characteristics of the filter are satisfied. Utilize only ideal LC components () and keep in mind that insertion and return loss have different axes on the left and right sides of the graph.
You can redraw the final schematic of the filter to the AWR project using ideal lumped elements from Elements tab→Lumped Element→Capacitor→CAP and Lumped Element→Inductor→IND. The values of the LC components in the iFilter schematic are shown in the AWR default units. The iFilter design can also be automatically generated for the AWR project by clicking on the Generate Design button. Several export options can be selected there, but to export the schematic in a simple fashion, none of the options need to be selected. Set Base Name without spaces. After clicking on OK the design procedure is saved under the previously selected base name. If optimization goals have been utilized to check the frequency characteristic of the LC filter, these goals are also exported into the Optimizer Goals folder in the AWR project. For band-pass and band-stop filters, it is necessary to suitably set BW and Fo to fulfil the frequency characteristic requirements.

2 a) Low-pass filter
This filter design consists of low- and high-impedance transmission lines. An ideal serial lumped inductor can be sufficiently replaced by a short high-impedance transmission line. Its width should be as narrow as possible, *i.e.*, 0.1 mm in our case. Design equations are stated on Slide 119 of the lectures. By using desired inductance L, impedance Z_L of the 0.1 mm line on the specific substrate with effective permittivity ε_{ef}, eq. (5.1.44) lets you compute the proper length l of the line. A specific example of the realization of 0.55 nH inductance on the RO4350 substrate can be found in Fig. 11. Agreement between the ideal inductor and short high-impedance transmission line is sufficient. It is necessary to compare both the amplitude and phase of the reflection coefficient.

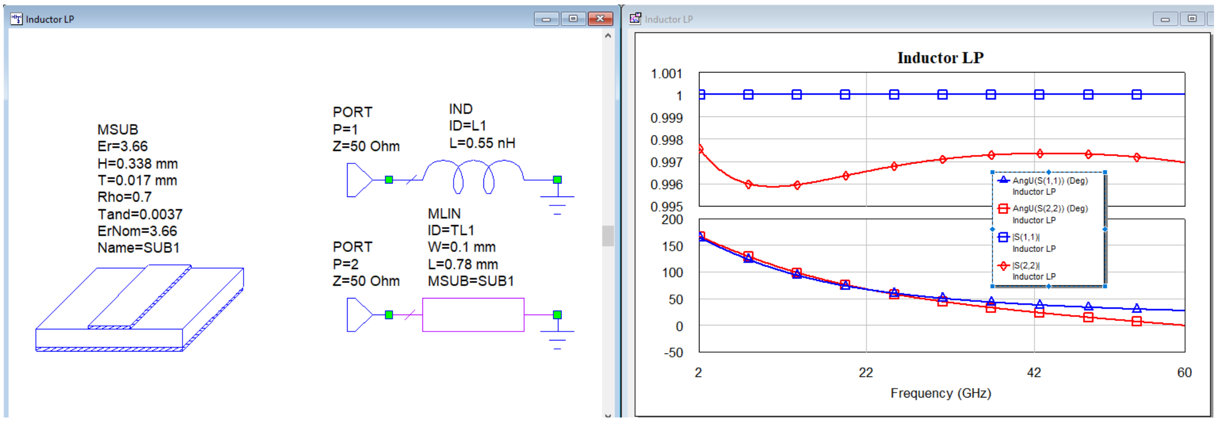

Fig. 11 Inductance 0.55 nH and its practical realization by the shorted high-impedance microstrip line and a comparison of reflection coefficients.

The lumped capacitor can be replaced by a short section of the low-impedance transmission line. For desired capacity C, length l of the line can be computed from the equation shown on Slide 127. The impedance of line Z_C should be as low as possible while ensuring single mode propagation. The lowest reachable impedance on the utilized substrate can be computed using eq. (2.7.8) from Slide 32 of the lectures. Suppose the first higher waveguide mode TE10 to have a frequency $f_m^{TE10} = f_0$. A comparison of the transmission through the ideal shunt capacitance 0.15 pF and the capacitor formed from the short section of the low-impedance line can be seen in Fig. 12. The resistor between ports 1 and 2 prevents the direct connection of ports which is forbidden in AWR.

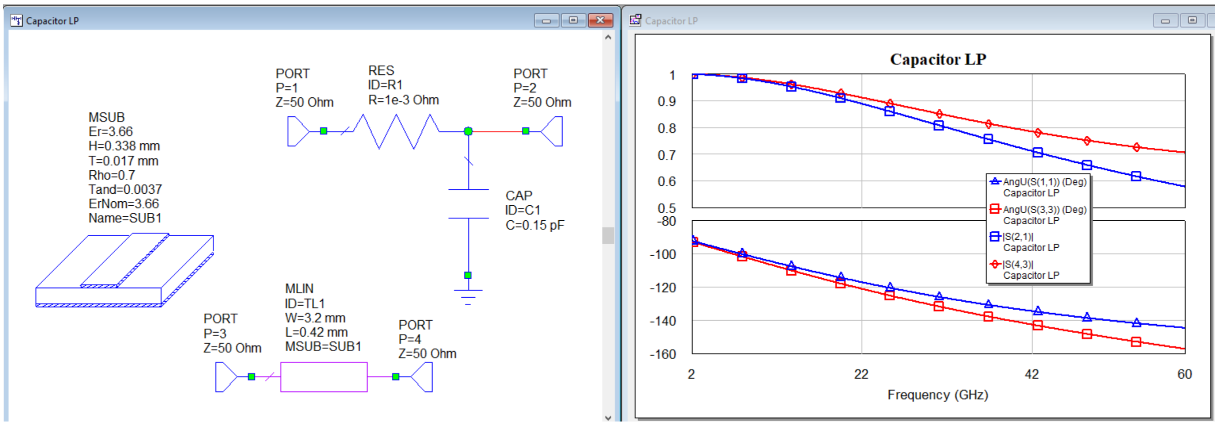

Fig. 12 Shunted capacitance 0.15 pF and its practical realization by the low-impedance microstrip line and a comparison of transmission coefficients

After computing the length of the second capacitor, all sections of the transmission lines can be connected. It is necessary to slightly tune the lengths of the individual components to fulfill the filter requirements. The final schematic of the low-pass filter and its parameters are shown in Fig. 13.

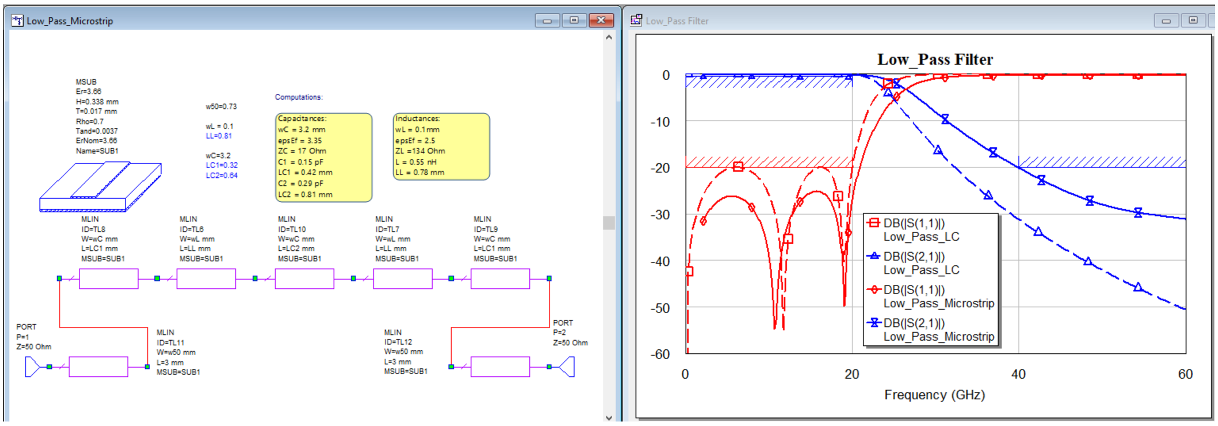

Fig. 13 Schematic of the low-pass filter and its parameters.

To finish the design, it would be necessary to add the discontinuity modeling steps between individual lines with different widths, but AWR does not contain a model which would describe such a huge impedance step. To fully validate the design, it would be necessary to model the filter in a 3D electromagnetic simulator. The final layout is shown in Fig. 14.

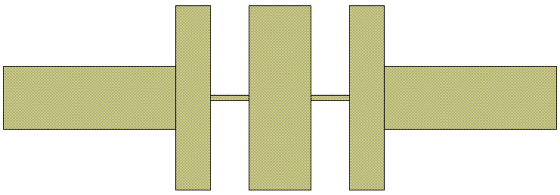

Fig. 14 Layout of the low-pass filter.

2 b) High-pass filter

All capacitors are considered as ideal, which is reasonable when MIM capacitors can be utilized. Both shunt inductances can be realized as a short section of the high-impedance transmission line with an end connected to ground through a via. The higher the impedance, the better the inductor representation, *i.e.*, it is suitable to design the transmission line with width $w = 0.1$ mm and on a substrate with low ε_r. Design equations are stated on Slide 120 of the lectures and it is necessary to express length l. Fig. 15 shows a comparison of ideal inductor $L = 0.29$ nH and its planar realization. Element modeling the via can be found in Microstrip→Other→MVIA1P. It is suitable to utilize a thin substrate because the electrical length of the via is not usually negligible and computed length *l* of the high-impedance line has to be shortened by the substrate's thickness. The diameter of the via should be comparable to the width of the line.

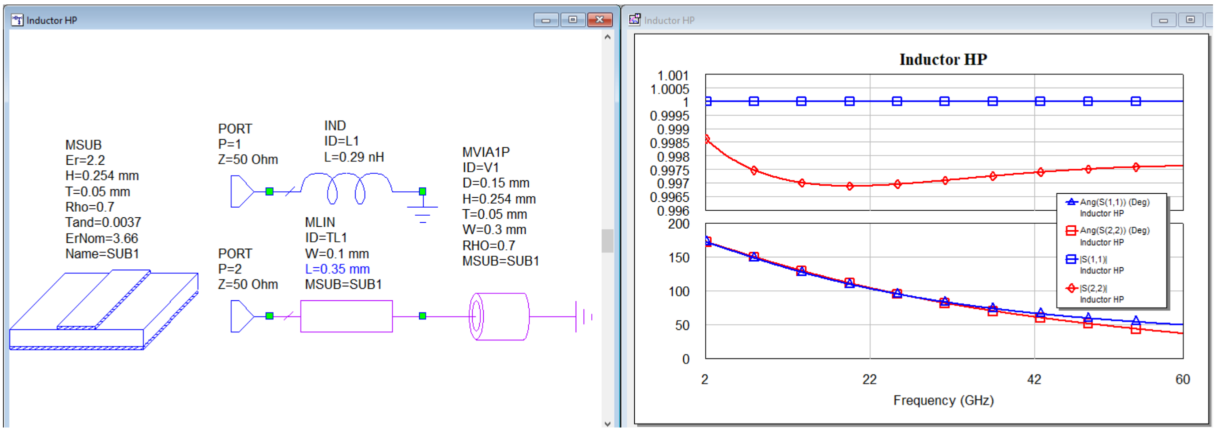

Fig. 15 Inductance 0.29 nH and its practical realization by the shorted high-impedance microstrip line with the via and a comparison of reflection coefficients.

The final schematic and parameters of the high-pass filter is shown in Fig. 16. It is necessary to tune the length of the inductor and capacitor values to fulfil design requirements.

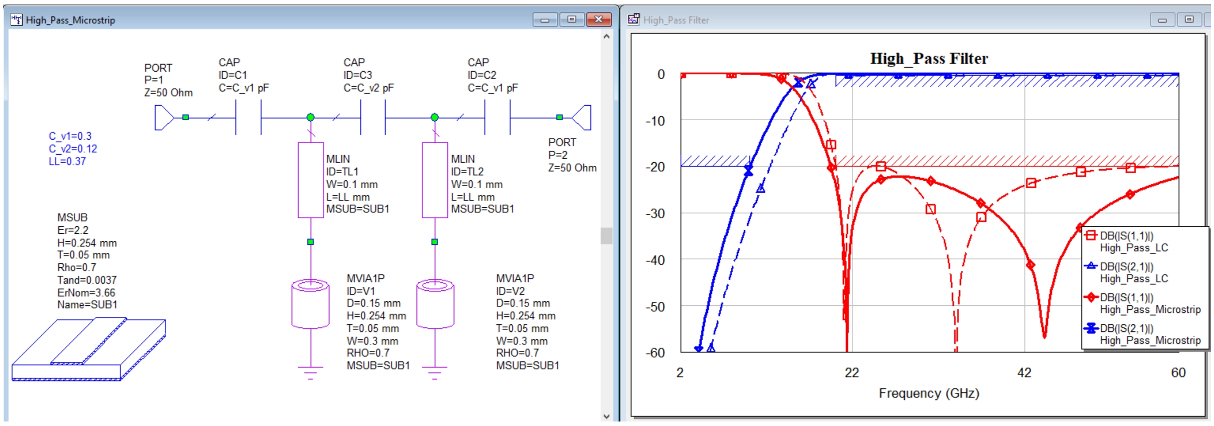

Fig. 16 Schematic of the high-pass filter and its parameters.

2 c) Band-pass filter

The band-pass filter consists of a cascade of $\lambda_g/4$ coupled lines. A single coupled-line section can be simulated using Microstrip→Coupled Lines→MCFIL which already works as a band-pass filter, but not with sufficient selectivity. Fig. 17 shows the schematic and the result of a simulation of the single coupled-line section with design frequency f_0 = 20 GHz. Tune its parameters to roughly filter the band around f_0 and utilize it as the starting point of the subsequent design.

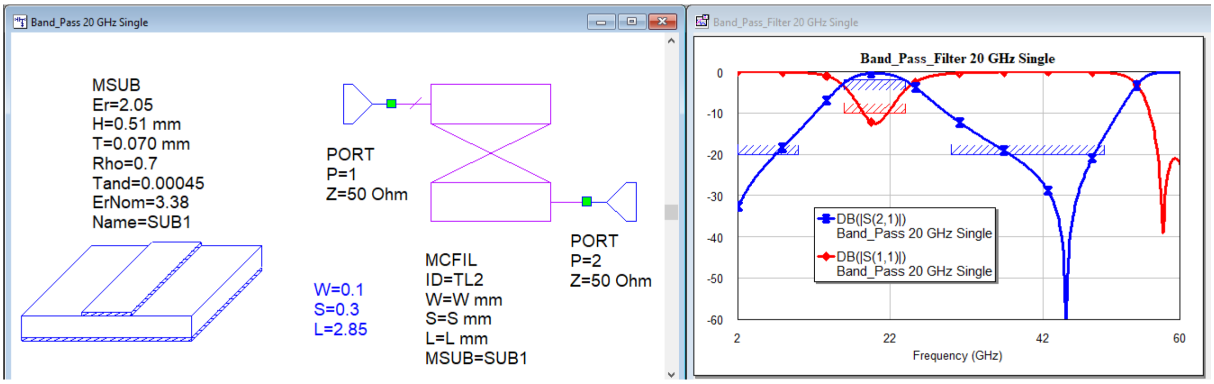

Fig. 17 Schematic and parameters of the single section of the coupled lines.

Connecting two sections in series increases the selectivity of the filter but also creates some peaks in transmission characteristic. Try to have the width of both microstrips the same in both sections, or properly add and set discontinuity model MSTEP between sections. It will probably be necessary to have different lengths and widths of the gaps of both sections to obtain a reasonable filter characteristics. In some cases, filter requirements can be fulfilled using just two sections. Fig. 18 shows the schematic and simulation results of two sections with the same width of both sections. There are five variables which set the filter characteristics. This is usually too much for manual tuning, hence we will utilize optimization for this purpose (described later).

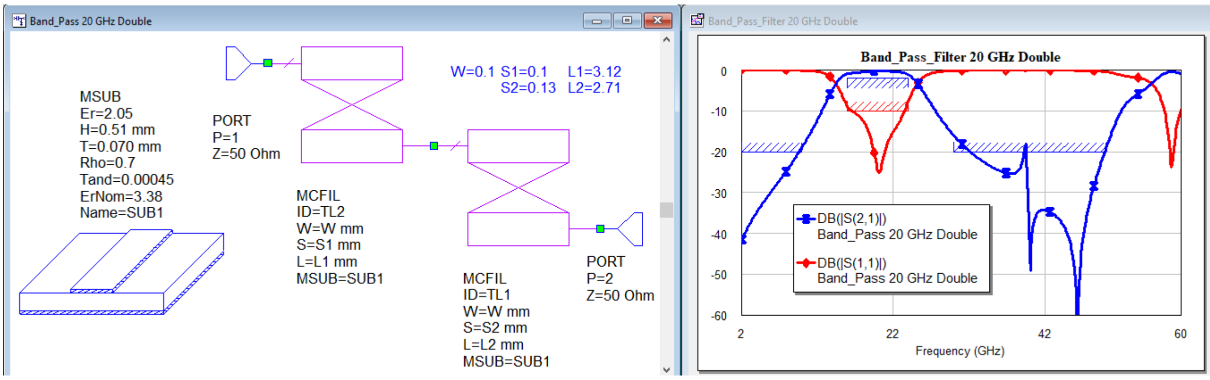

Fig. 18 Schematic and parameters of the two sections of the coupled lines tuned as a band-pass filter.

Adding the following section could help you reach the desired filter requirements. It is usually convenient to make the filter structure symmetric because the LC structure is also symmetric. Therefore, the number of variables will remain the same as in the previous case as both terminal sections are the same. The final schematic, filter response and layout is shown in Fig. 19. It is convenient to select a substrate with low permittivity and higher thickness. Simulating connecting 50 Ω lines in AWR is meaningless in this case because there is no proper implementation of an impedance step of high ratio. An electromagnetic field simulator would be necessary to validate the filter connection and final parameters.

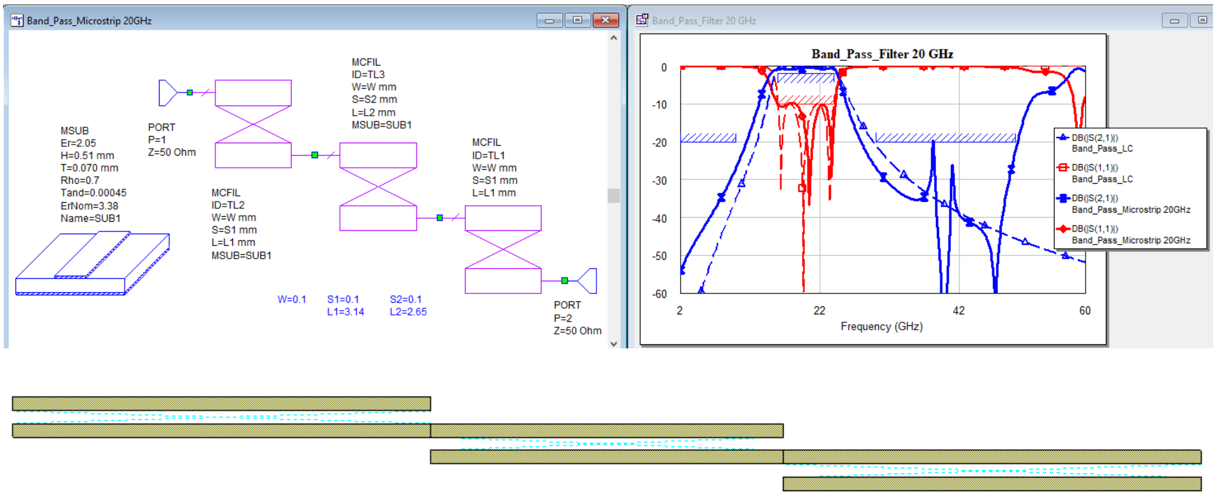

Fig. 19 Schematic, parameters and layout of the band-pass filter.

When a designed structure parametrized by more than three parameters and final circuit requirements are challenging, then manual tuning of the structure is quite difficult. Fortunately, AWR contains an optimization module which can be utilized for the optimization of any circuit parameter. First of all, optimized quantity has to be shown in a graph. Then, the optimization goal has to be defined. It can be done by right clicking on Optimizer Goals→Add Optimizer Goal... Then, the parameters of the goal can be defined. Just the measurements with an outcome as a real number can be optimized, *i.e.*, a complex reflection coefficient cannot but its absolute value or phase can. The goal will be graphically shown in a related graph. It can be seen in Fig. 20.

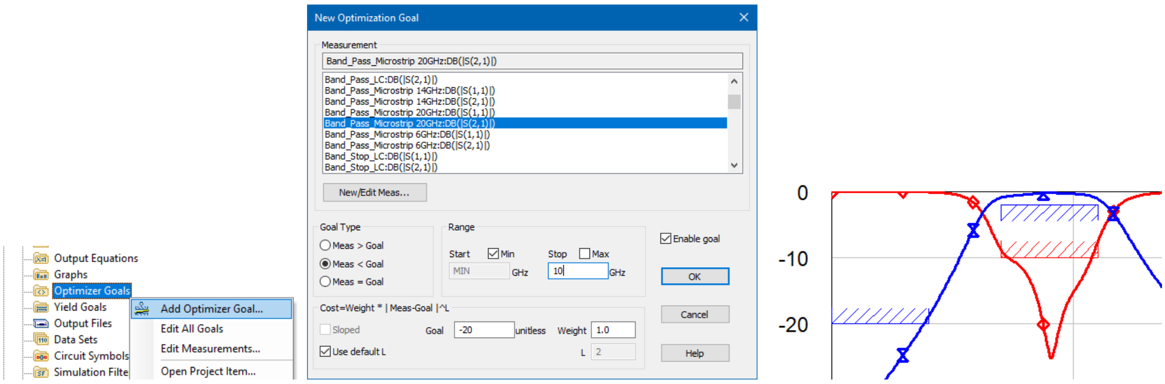

Fig. 20 Definition and appearance of the optimization goal in a graph.

The definition of variables, which will be changed by the optimizer, can be defined in a list of all variables. It can be opened by clicking on the button (Variable Browser). The browser contains a list of all variables and you can select which variable is to be optimized and what its constraints will be. We generally recommend to always constrain the range of individual optimized variables because you should have always at least a rough estimate of what the correct value of a certain circuit parameter is. Microwave circuits usually have periodically repeating properties regarding frequency and the optimizer could find just the local minimum of the optimization task. Optimization control can be opened by clicking on the Simulate→ Optimize button. There you can select from many optimization methods and optimization can be started by pressing the Start button. The optimization progress can be observed in its respective graph with the optimization goal, as shown in Fig. 21.

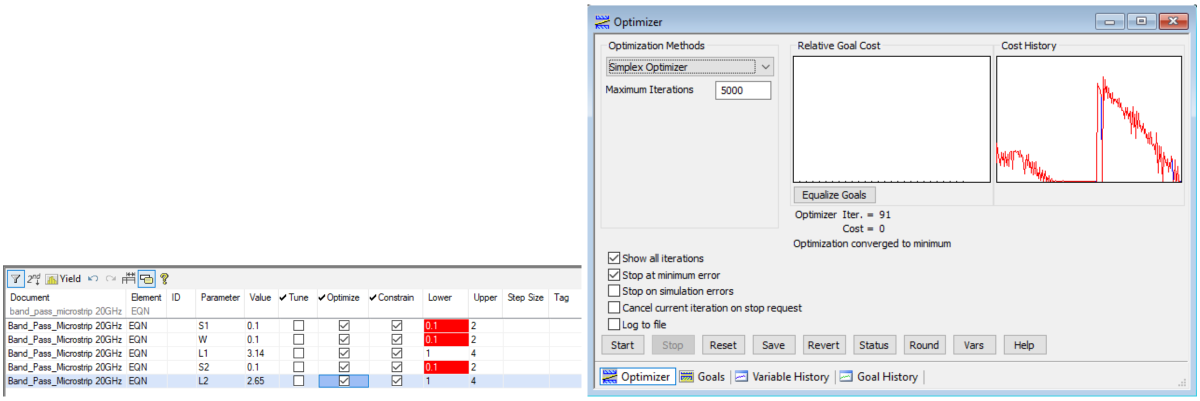

Fig. 21 Variable browser and optimizer.

2 d) Band-stop filter

The band-stop filter consists of an open microstrip stub in its central part which can be simulated using the Microstrip→MLEF element which simulates the fringing capacity at the end of the strip, even when it has zero length, as shown in Fig. 22. The capacity of the open end of the 50 Ω microstrip line on a Rogers RO4350 substrate with 0.168 mm thickness is approx. 8 fF.

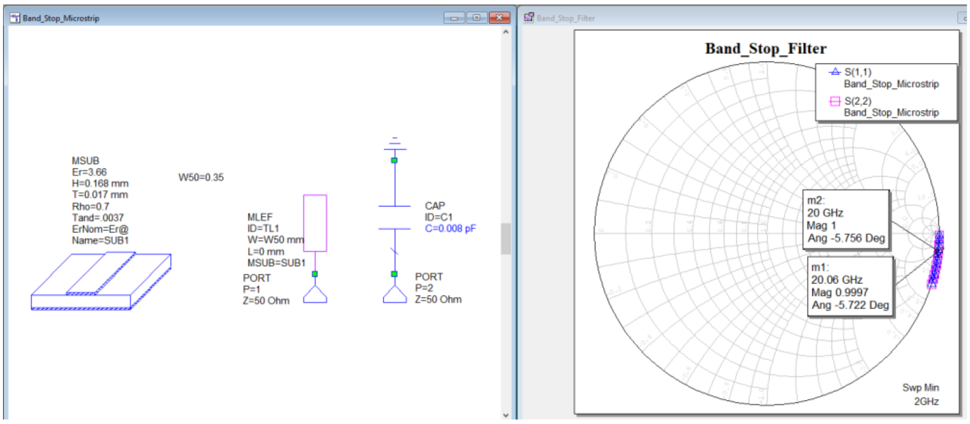

Fig. 22 Fringing capacity of the open end of a microstrip line.

Making the open stub up to $\lambda_g/4$ long, its open end (almost infinite impedance) is transformed along the stub length to zero impedance, hence it now acts as a serial RLC circuit with resonant frequency f_0. In this example $f_0 = 20$ GHz. A comparison of both circuits is shown in Fig. 23 and illustrates the similarity of both circuits in a wide range around f_0.

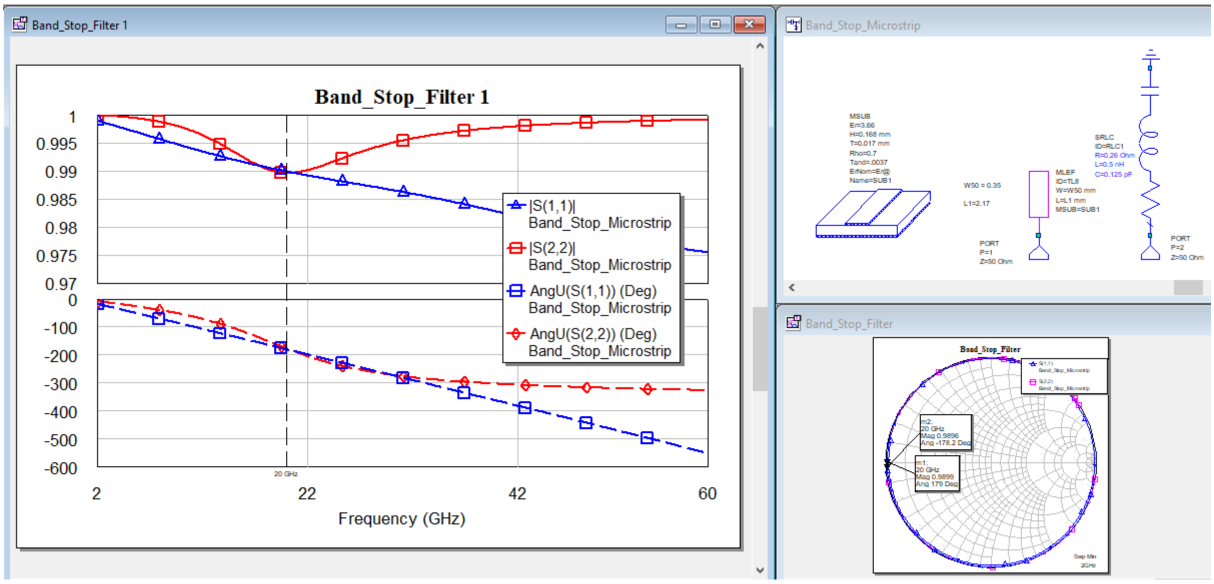

Fig. 23 Comparison of a $\lambda_g/4$ open stub and serial RLC circuit with f_0 = 20 GHz.

By adding a T junction and almost zero resistor it is possible to compare both circuits connected in parallel between two ports. The schematic and results are shown in Fig. 24. The similarity of both circuits is clear around frequency f_0.

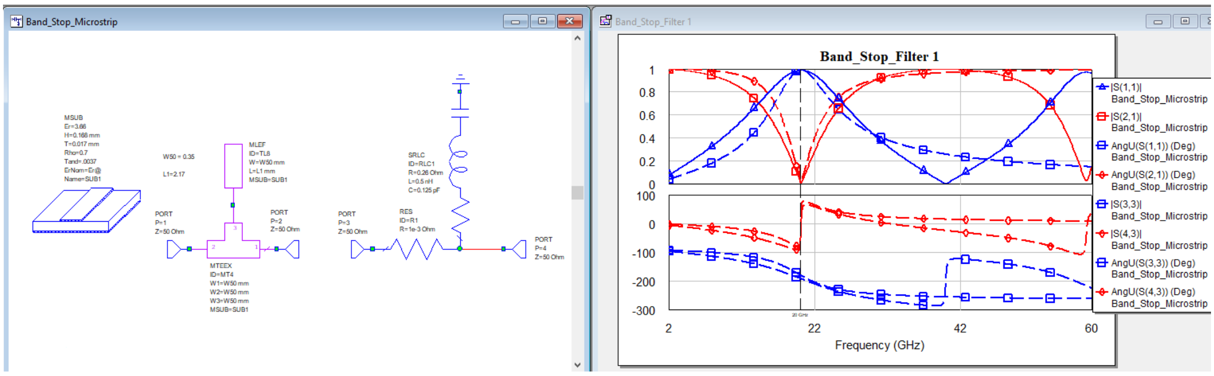

Fig. 24 $\lambda_g/4$ open stub and serial RLC circuit connected in parallel between two ports.

Connecting two $\lambda_g/4$ transmission lines to the T junction transforms a serial resonance circuit to a parallel resonant circuit. The length of both lines should be set to obtain high impedance seen from port 1 at design frequency f_0. A properly set length of lines is shown in Fig. 25 with the circuit now acting as a parallel resonant circuit connected in series between both ports.

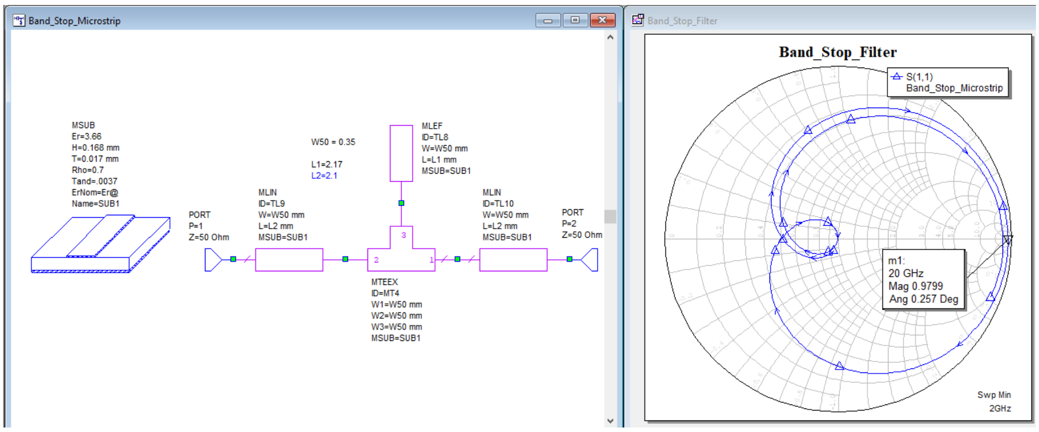

Fig. 25 Transformation of the serial resonant circuit to the parallel one using two $\lambda_g/4$ transmission lines.

Adding an additional two $\lambda_g/4$ open stubs at the end of the circuit adds serial resonance circuits connected to ground. Adding proper T junctions and connecting 50 Ω lines means the filter is ready for a final tuning. The final schematic is shown in Fig. 26.

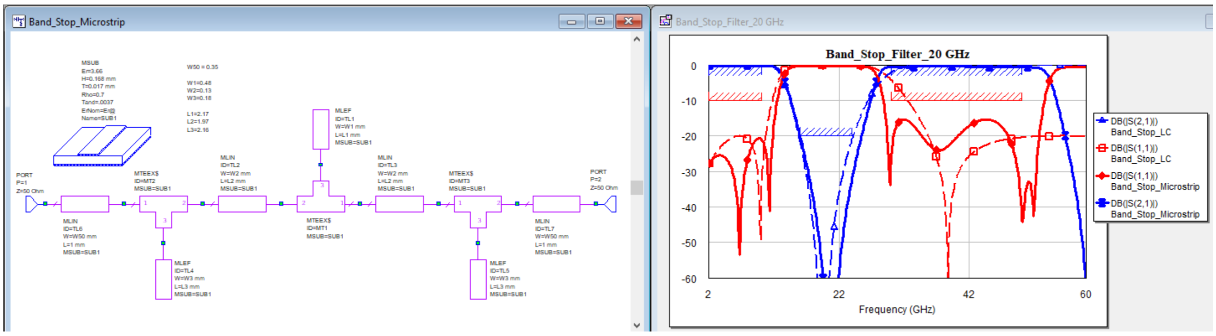

Fig. 26 Final schematic and parameters of the planar band-stop filter.

The structure of the filter should be symmetric, but the actual parametrization is up to you. In this example, all stubs and lines have their individual lengths and widths, but it is not usually necessary to have so many parameters. In the assignment, a sketch of a layout with two open stubs in the central part is shown. It is not always necessary to have two stubs in a design but it could enhance the filter final parameters when the second stub is properly tuned. As you know from your previous design of a band-pass filter, the characteristics of filters made from resonant-length microstrip lines are periodic and it will probably not be possible to fulfil the expected requirements. Try to match the parameters of LC and the microstrip filter up to approx. $2.5f_0$, or define your own reasonable filter requirements. The layout of the final filter is shown in Fig 27.

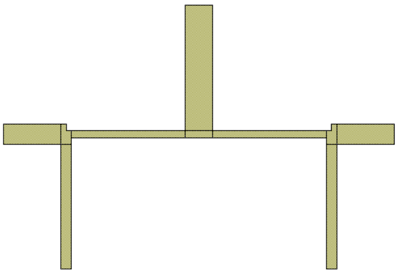

Fig. 27 Final layout of the planar band-stop filter.