TECHNICAL
OVERVIEW

Chapter 6

Microwave Discrete and Microstrip Filter Design

PathWave Advanced Design System (ADS)

Background

Microwave filters play an important role in any RF front end for the suppression of out of band signals. In the lumped and distributed form, they are extensively used for both commercial and military applications. A filter is a reactive network that passes a desired band of frequencies while almost stopping all other bands of frequencies. The frequency that separates the transmission band from the attenuation band is called the cutoff frequency and denoted as fc. The attenuation of the filter is denoted in decibels or nepers. A filter in general can have any number of pass bands separated by stop bands. They are mainly classified into four common types – namely low-pass, high-pass, bandpass, and band stop filters.

An ideal filter should have zero insertion loss in the pass band, infinite attenuation in the stop band, and a linear phase response in the pass band. An ideal filter cannot be realizable as the response of an ideal lowpass or band pass filter is a rectangular pulse in the frequency domain. The art of filter design necessitates compromises with respect to cutoff and roll off. There are basically three methods for filter synthesis. They are the image parameter method, insertion loss method, and numerical synthesis. The image parameter method is an old and crude method, whereas the numerical method of synthesis is newer but cumbersome. The insertion loss method of filter design on the other hand is the optimum and more popular method for higher frequency applications. The filter design flow for insertion loss method is shown in Figure 1.

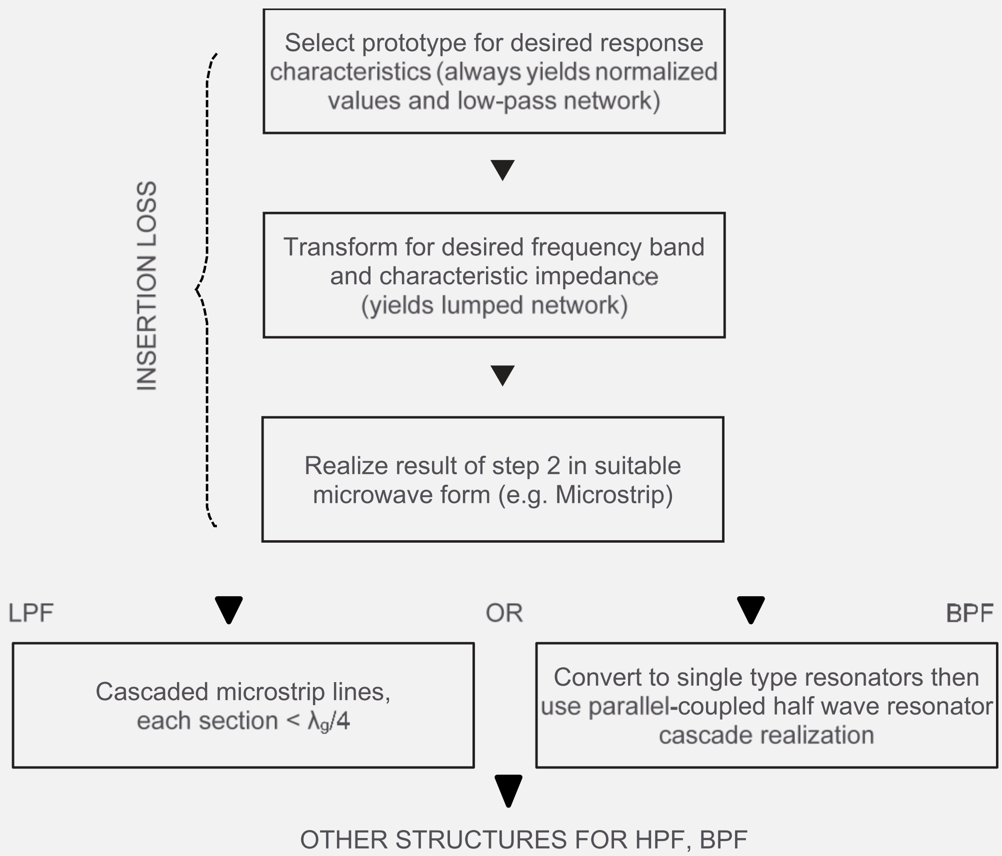

Figure 1. Filter design flow for insertion loss method

Since the characteristics of an ideal filter cannot be obtained, the goal of filter design is to approximate the ideal requirements within an acceptable tolerance. There are four types of approximations – namely Butterworth or maximally flat, Chebyshev, Bessel, and Elliptic approximations. For the prototype filters, maximally flat or Butterworth provides the flattest pass band response for a given filter order. In the Chebyshev method, sharper cutoff is achieved and the pass band response will have ripples of amplitude 1+k2. Bessel approximations are based on the Bessel function, which provides sharper cutoff, and Elliptic approximations results in pass band and stop band ripples. Depending on the application and the cost, the approximations can be chosen. The optimum filter is the Chebyshev filter with respect to response and the bill of materials. Filters can be designed both in the lumped and distributed form using the above approximations.

Design of Microwave Filters

The first step in the design of microwave filters is to select a suitable approximation of the prototype model based on the specifications.

Calculate the order of the filter from the necessary roll off as per the given specifications. The order can be calculated as follows:

Butterworth Approximation:

$$L_A(\omega') = 10\log_{10}\left\{1+\varepsilon\left(\frac{\omega'}{\omega_c}\right)^{2N}\right\}$$

Where $\varepsilon = \{\text{Antilog}_{10} L_A/10\}-1$ and $L_A = 3$ dB for Butterworth

Chebyshev Approximation:

$$L_A(\omega') = 10\log_{10}\left\{1+\varepsilon\cos^2\left[n\cos^{-1}\left(\frac{\omega'}{\omega_1'}\right)\right]\right\} \qquad \text{when } \omega' \leq \omega_1' \text{ and}$$

$$L_A(\omega') = 10\log_{10}\left\{1+\varepsilon\cosh^2\left[n\cosh^{-1}\left(\frac{\omega'}{\omega_1'}\right)\right]\right\} \qquad \text{when } \omega' \geq \omega_1'$$

Where ω_c is the angular cutoff frequency

ω' is the angular attenuation frequency

$L_A(\omega')$ is the attenuation at ω'

N is the order of the filter

Where $\varepsilon = \{\text{Antilog}_{10} L_{Ar} / 10\} - 1$ and L_{Ar} is ripple in passband

The next step in the filter design is to calculate the prototype values of the filter depending on the type of approximation. The prototype values for the Chebyshev and Butterworth approximations can be calculated using the following equations:

Butterworth Approximation:

$g_0 = 1$

$g_k = 2\sin\{(2k-1)\pi / 2n\}$ where k = 1, 2, …, n and

$g_{N+1} = 1$

Where n is the order of the filter

Chebyshev Approximation:

$$\beta = \ln\left(\coth\frac{L_{Ar}}{17.37}\right) \text{ where } L_{Ar} \text{ is ripple in the passband}$$

$$\gamma = \sinh\left(\frac{\beta}{2n}\right)$$

$$a_k = \sin\left[\frac{(2k-1)\pi}{2n}\right], k = 1, 2, 3, \ldots, n$$

$$b_k = \gamma^2 + \sin^2\left(\frac{k\pi}{n}\right), k = 1, 2, 3, \ldots, n$$

$$g_1 = \frac{2a_1}{\gamma}$$

$$g_k = \frac{4a_{k-1}a_k}{b_{k-1}g_{k-1}}, k = 2, 3, \ldots, n$$

$$g_{n+1} = 1 \text{ for } n \text{ odd}$$

$$= \coth^2\left(\frac{\beta}{4}\right) \text{ for } n \text{ even}$$

After computing the prototype values, the prototype filter must be transformed with respect to frequency and impedance to meet the specifications. The transformations can be done using the given equations below.

For Lowpass Filter:

After impedance and frequency scaling:

$$C_k^{'} = \frac{C_k}{R_0\omega_c}$$

$$L_k^{'} = \frac{R_0L_k}{\omega_c} \text{ where } R_0 = 50\ \Omega$$

For the distributed design, the electrical length is given by:

Length of capacitance section (βl_c) : C_kZ_l / R_0

Length of inductance section (βl_l) : L_kR_0 / Z_h

Where Z_l is the low impedance value and Z_h is the high impedance value

For bandpass filter:

After impedance and frequency scaling:

$$L_1^{'} = \frac{L_1Z_0}{\omega_0\Delta}$$

$$C_1^{'} = \frac{\Delta}{L_1Z_0\omega_0}$$

$$L_2^{'} = \frac{\Delta Z_0}{\omega_0C_2}$$

$C_2' = \frac{C_2}{Z_0 \Delta\, \omega_0}$

$L_3' = \frac{L_3 Z_0}{\omega_0 \Delta}$

$C_3' = \frac{\Delta}{L_3 Z_0 \omega_0}$

Where Δ is the fractional bandwidth $\Delta = (\omega_2 - \omega_1) / \omega_0$

Calculating the electrical length of the distributed design of the bandpass filter will be discussed later in this chapter.

Simulation of a Lumped and Distributed Lowpass Filter Using ADS

Typical Design

Cutoff Frequency (f_c)	: 2 GHz
Attenuation at f = 4 GHz	: 30 dB ($L_A(\omega)$)
Type of approximation	: Butterworth
Order of the filter	: $L_A(\omega') = 10\log_{10}\left\{1+\varepsilon\left(\frac{\omega}{\omega_c}\right)^{2N}\right\}$

Where $\varepsilon = \{\text{Antilog}_{10} L_A/10\} - 1$

Substituting the values of $L_A(\omega)$, ω, and ω_c, the value of N is calculated to be 4.

Protype Values of the Lowpass Filter

The prototype values of the filter are calculated using the formulas given earlier:

$g_0 = 1$

$g_k = 2\sin\{(2k - 1)\pi / 2n\}$ for k = 1, 2, …, n, and

$g_{N+1} = 1$

The prototype values for the given specifications of the filter are:

$g_1 = 0.7654 = C_1$

$g_2 = 1.8478 = L_2$

$g_3 = 1.8478 = C_3$

$g_4 = 0.7654 = L_4$

Lumped Model of the Filter

The lumped values of the lowpass filter after frequency and impedance scaling are given by the formulas given earlier:

$C_k^{'} = \frac{C_k}{R_0 \omega_c}$

$L_k^{'} = \frac{R_0 L_k}{\omega_c}$ where R_0=50 Ω

The resulting lumped values are:

$C_1^{'}$ = 1.218 pF

$L_2^{'}$ = 7.35 nH

$C_3^{'}$ = 2.94 pF

$L_4^{'}$ = 3.046 nH

Distributed Model of the Filter

For the distributed design, the electrical length is given by:

Length of capacitance section (βl_c)	: $C_k Z_l / R_0$
Length of inductance section (βl_l)	: $L_k R_0 / Z_h$

Where

Z_l is the low impedance value

Z_h is the high impedance value

R_0 is the source and load impedance

ω_c is the desired cutoff frequency

If we consider Z_l = 10 Ω and Z_h = 100 Ω, then

βl_{c1} = 0.1530

βl_{i2} = 0.9239

βl_{c3} = 0.3695

βl_{i4} = 0.3827

Since β = 2π / λ, the physical lengths are given by

l_{c1} = 1.680 mm

l_{i2} = 10.145 mm

l_{c3} = 4.057 mm

l_{i4} = 4.202 mm

Schematic Simulation Steps for Lumped Lowpass Filter

1. Open the Schematic window of ADS.
2. From the **Lumped-Components** library, select the appropriate components necessary for the lumped element lowpass filter. As you place components on the schematic, enter the component values that were calculated earlier.

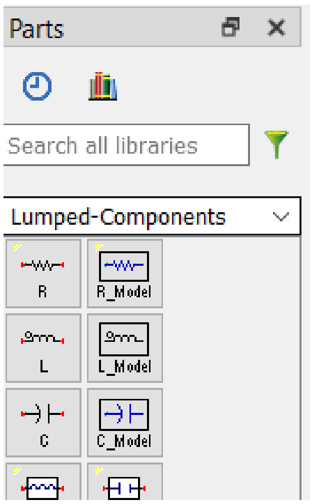

Figure 2. Parts library for lumped elements

3. Connect the components with wire and terminate both ports of the lowpass filter using terminations from the **Simulation_S-Param** library.
4. Place the S-Parameter simulation controller from the Simulation S-Param library and set its parameters as:

   Start = 0.1 GHz

   Stop = 5 GHz

   Number of Points = 101 (or Step Size = 49 MHz)

The design of the lumped element low pass filter is shown in Figure 3.

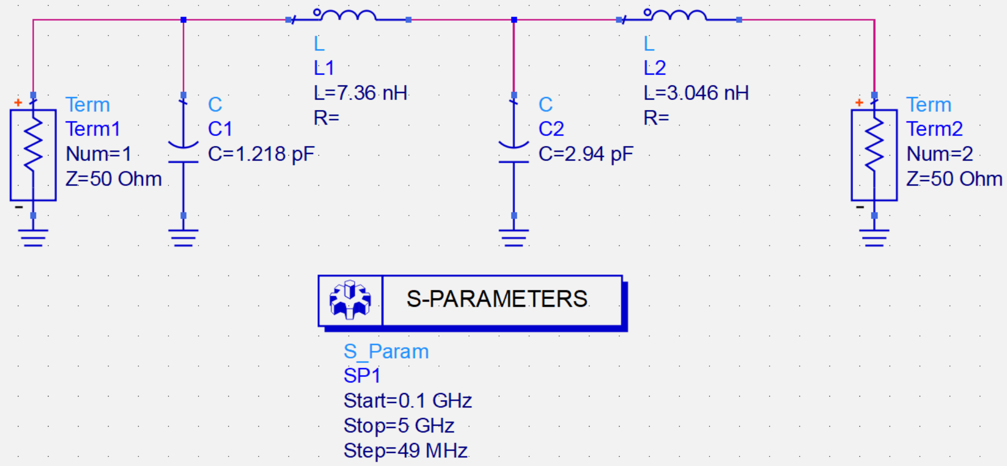

Figure 3. Schematic of the completed low pass filter

5. Simulate the circuit by clicking **F7** or the simulation gear icon.
6. After the simulation is complete, ADS automatically opens the Data Display window, which shows the results. If the Data Display window does not open, click **Window > New Data Display**. In the Data Display window, select a rectangular plot. In the popup window, plot S(1,1) and S(2,1) as dB.
7. Click and insert a marker on S(2,1) trace around 2GHz to see the data display graph as shown in Figure 4. The point at which S(2,1) is at -3 dB is the cutoff frequency. According to our design, this should be around 2 GHz.

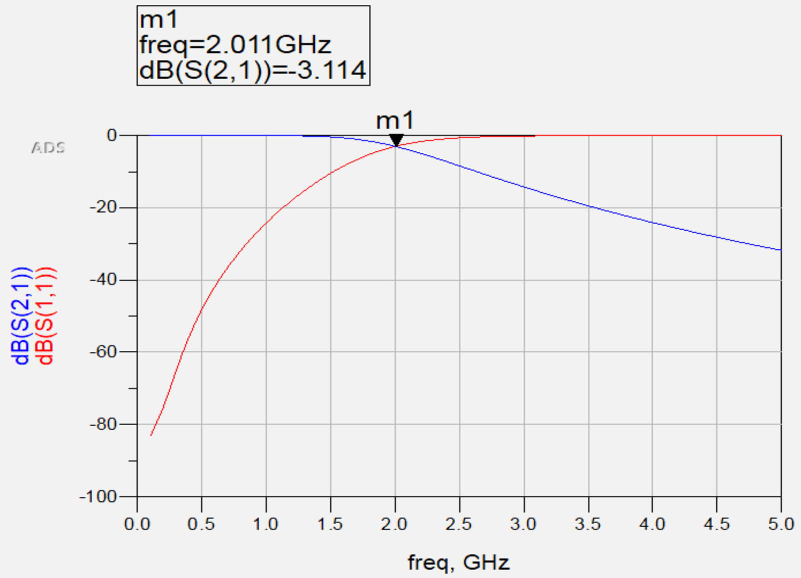

Figure 4. Graph showing the S(1,1) and S(2,1) of the lowpass filter

Results and Observations

The simulation of the lumped element model shows that the lowpass filter has a cutoff frequency of about 2 GHz and has a gentle roll off, which is expected for a Butterworth filter.

Layout Simulation Steps for Distributed Low Pass Filter

Calculate the physical parameters of the distributed lowpass filter using the design procedure given above. Since the length has already been calculated, the only parameter left to calculate is the width of the Z_l and Z_h transmission lines for the stepped impedance low pass filter. As a reminder, Z_l = 10 Ω and Z_h = 100 Ω, which means the low impedance line width is 24.7 mm and the high impedance line width is 0.66 mm. Both calculations were done with a dielectric constant of 4.6 and a thickness of 1.6 mm. For simplicity, the low impedance and high impedance line widths are shown below:

$w_l = w_{c1} = w_{c3} = 24.7$ mm

$w_h = w_{i2} = w_{i4} = 0.66$ mm

Calculate the length and width of the 50 Ω line using Line Calc (**Tools > Line Calc > Start Line Calc**) in the schematic window. Change the values for Substrate Parameters, Component Parameters, and Electrical Characteristics (Z0 and E_Eff) to match Figure 5. Click **Synthesize** to generate the Physical Characteristics.

The 50 Ω input and output transmission line characteristics should be:

Width = 2.93 mm

Length = 4.53 mm

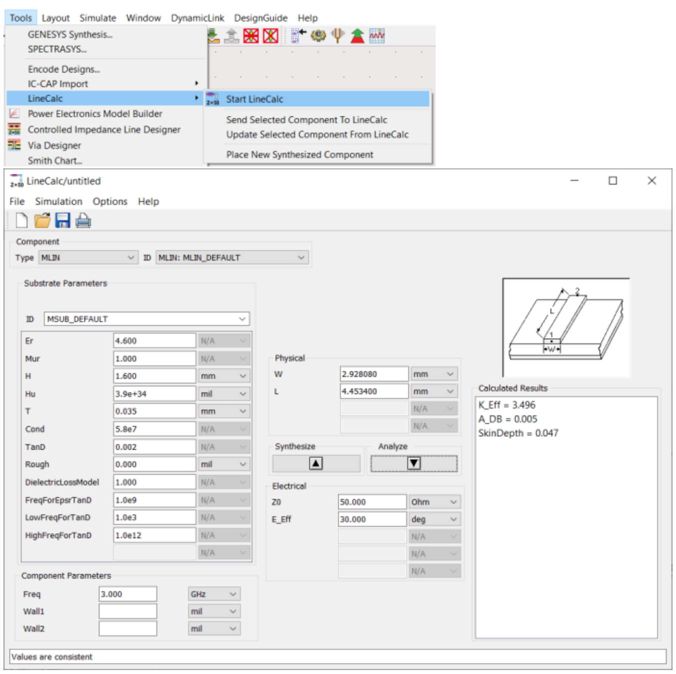

Figure 5. Using Line Calc to calculate the parameters for 50 Ω line

Now that the parameters have all been determined, create a model of the stepped impedance lowpass filter in the layout window. There are two ways to do this: using library components or by drawing rectangles.

To create the model using library components, select the **TLines-Microstrip** library, and use the **MLIN** components. The TLines-Microstrip library is shown in Figure 6. To create the model by drawing rectangles, go to **Insert > Rectangle**.

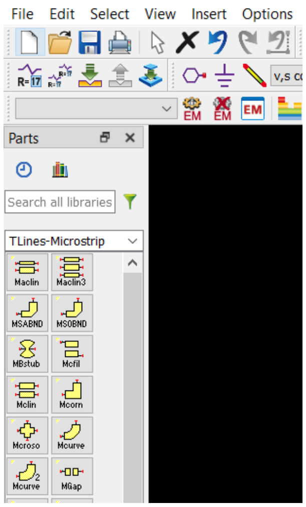

Figure 6. TLines-Microstrip library

Ensure that each of the different sections are connected. Connect Pin 1 to the input and Pin 2 to the output. The completed model of the stepped impedance (distributed) lowpass filter is shown in Figure 7.

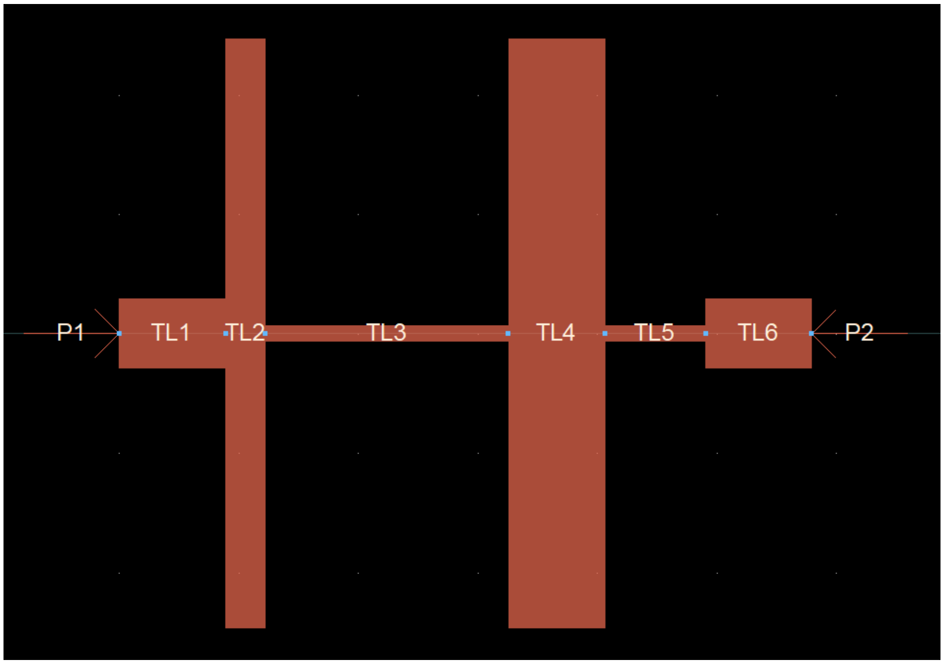

Figure 7. Layout of the stepped impedance lowpass filter

Setup the EM simulation as described earlier in the EM simulation chapter. The following properties will be used to define the substrate:

Material	: FR_4
Er	: 4.6
Height	: 1.6 mm
Loss Tangent	: 0.0023
Metal Thickness	: 0.035 mm
Metal Conductivity	: Cu (5.8E7 S/m)

The EM Setup window, showing the substrate definition, is shown in Figure 8. Make sure to enable Edge Mesh (**Options > Mesh > Edge Mesh**) and select ‘Auto-determine edge width’.

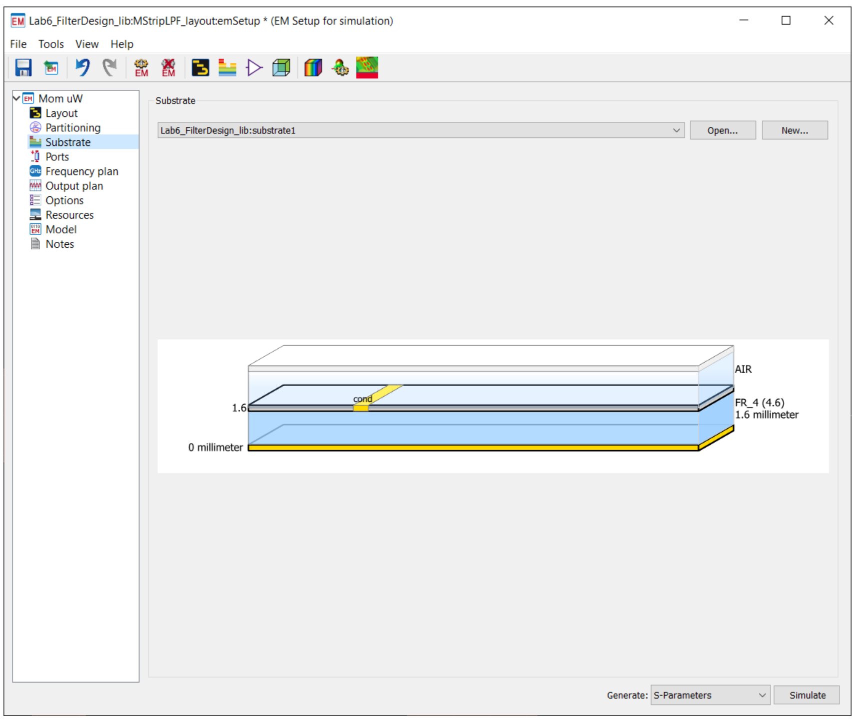

Figure 8. EM Setup window showing the substrate definition

After setting up the simulation, press **Simulate**. As with earlier, create a rectangular plot showing S(1,1) and S(2,1). This plot is shown in Figure 9.

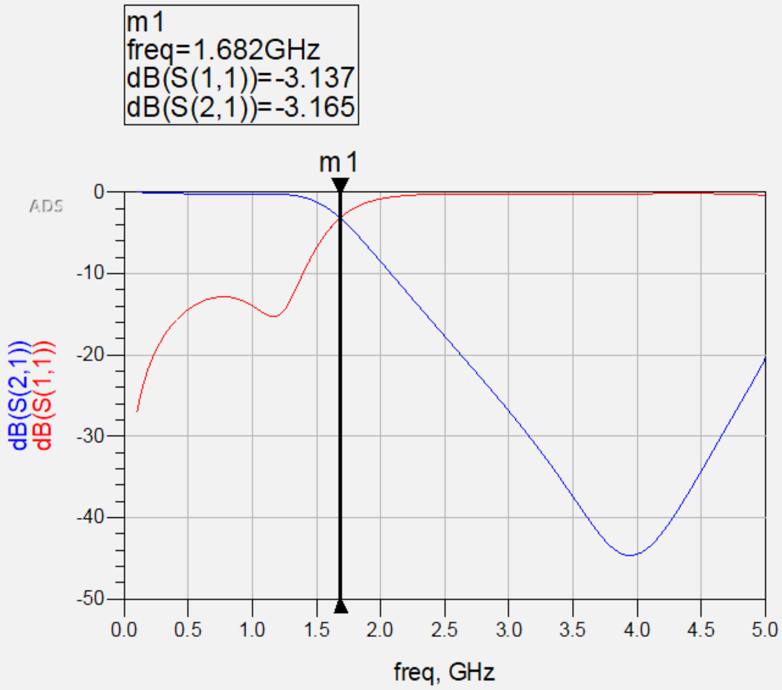

Figure 9. Graph showing the S(1,1) and S(2,1) of the distributed lowpass filter

Notice that the 3 dB cutoff has shifted from 2 GHz to 1.68 GHz, as the theoretical calculations don't allow for accurate analysis of open-end effects and a sudden change in the impedance of the transmission lines. In order to get closer to the 2 GHz cutoff frequency specification, the lines need to be optimized. This optimization can be carried out using the Momentum simulator or by performing a parametric sweep on the lengths of the capacitive and inductive lines.

Parametric EM Simulations in ADS

To begin parametric simulation on the layout, we need to define the variable parameters that will be associated with the layout components. Click **EM > Component > Parameters**, as shown in Figure 10.

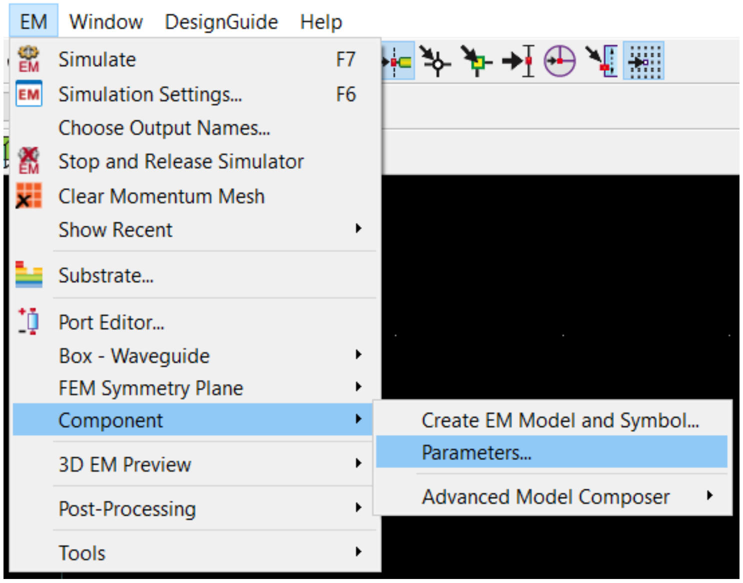

Figure 10. How to open parameter variable definitions

In the Design Parameters window, define variables for the length of capacitive and inductive lines. For the default value, enter the lengths that were calculated earlier, and make sure to select the appropriate unit. Set **Type** to **Subnetwork**, as these parameters will be associated with the microstrip library components, which have parameterized artwork. If you were trying to parameterize the polygon/rectangle-based components, you would select the **Nominal/Perturbed** method, which requires additional information.

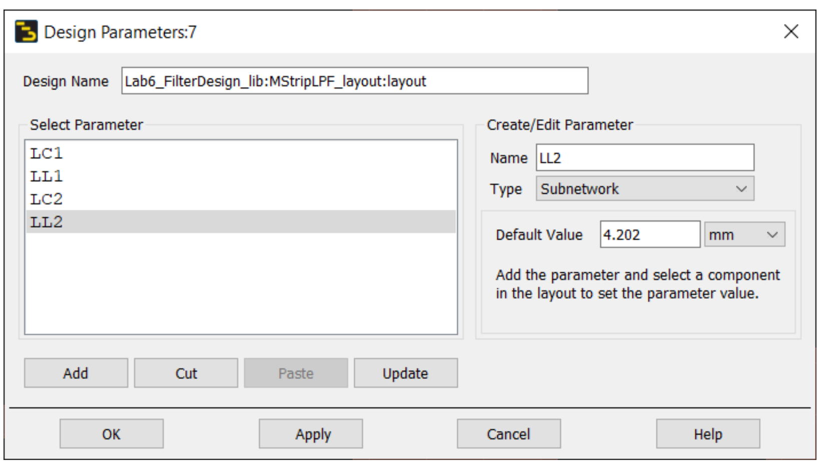

Figure 11. Design parameter definitions

After defining the parameters, double-click the respective components in the layout window and change the length values to the corresponding variable names. Note that no units need to be defined here, as the units are specified in the parameter definition. An example is in Figure 12.

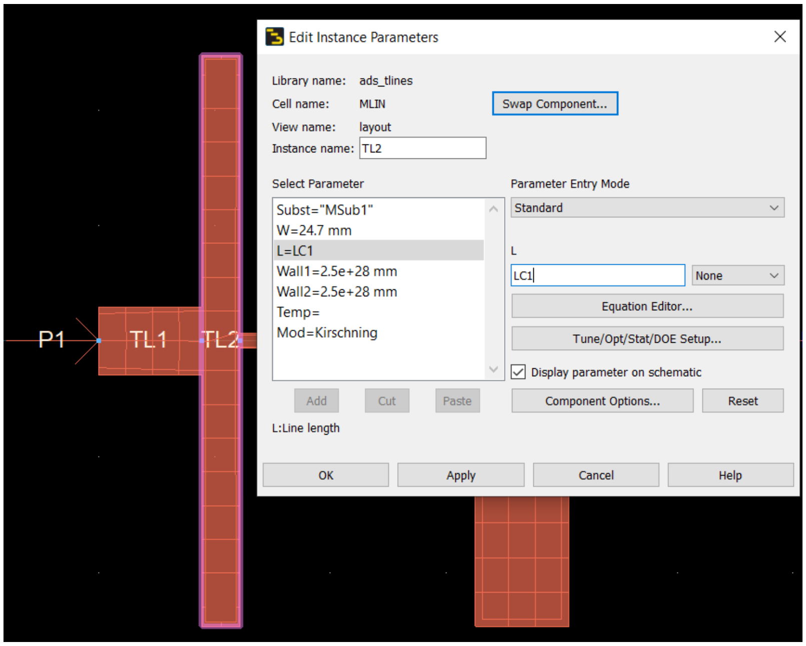

Figure 12. Redefining the length parameters

After defining the parameter values for each transmission line, the next step is to create an EM model and symbol that can then be used for parametric EM cosimulation. To create a parametric model and symbol for the layout, go to **EM > Component > Create EM Model and Symbol**. Select both options in the window that pops up.

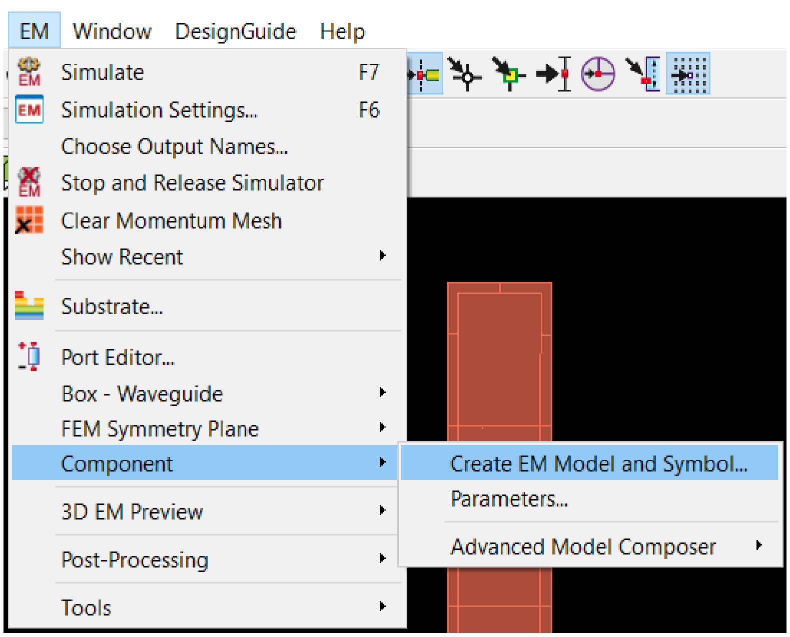

Figure 13. Creating an EM model and symbol

After the EM model and symbol have been created, they will appear below the layout cell, as shown in Figure 14.

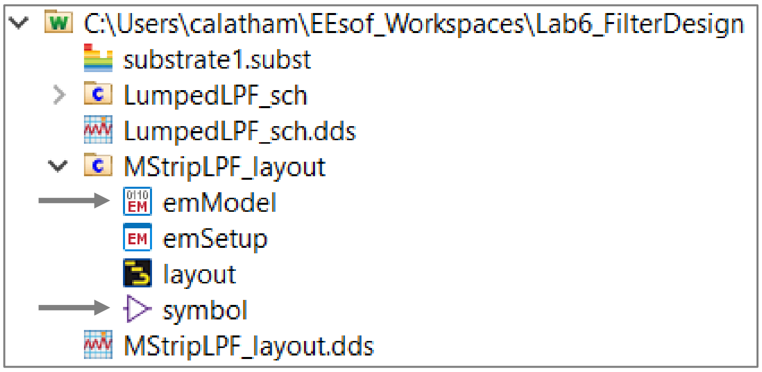

Figure 14. EM model and symbol shown in the layout

Open a new schematic cell and drag and drop the emModel component to place it as a subcircuit. The parameters defined in the layout view (from Figure 11) are the default values for the emModel component. These values will be swept using the Parameter Sweep component in the ADS schematic, but the Parameter Sweep is done on variables contained in a VAR block. Therefore, you will need to redefine the default values in the VAR block and change the parameters for the emModel to use the variables in the VAR block (i.e. set LC1=L1, which is defined the VAR block). In this case, the variables L1-L4 are defined in the VAR block and assigned to the emModel component. This is shown in Figure 15.

For our initial sweep, add a **Parameter Sweep** component (from the **Simulation-S_Param** library). Open the Parameter Sweep component and define **Parameter to Sweep** as **L2** (first inductive line). Set the sweep to go from **6.145** to **12.145** in steps of **1**. Under the Simulations tab, define **Simulation 1** as **SP1**.

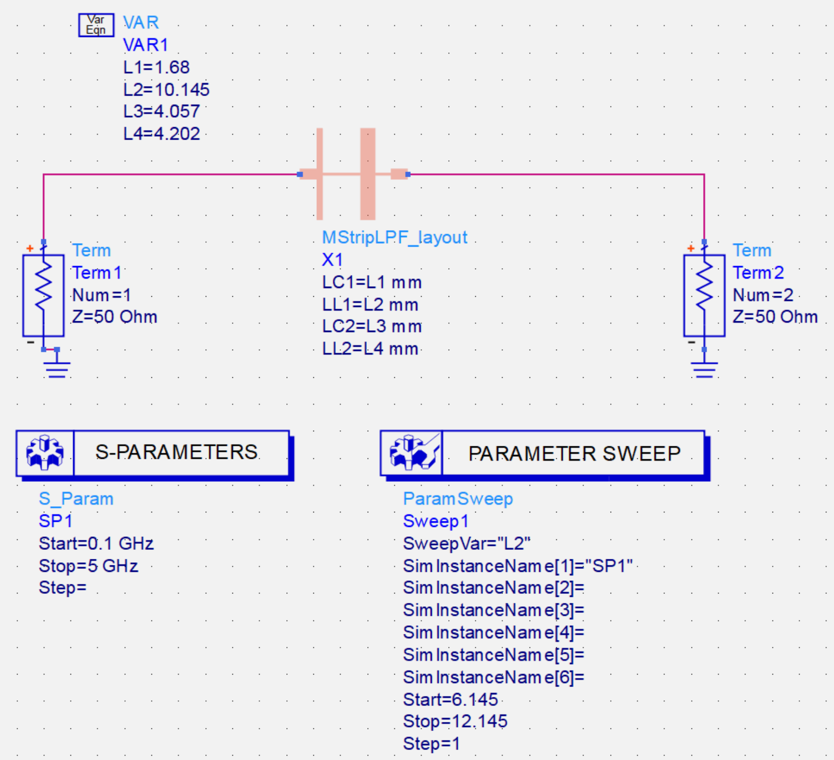

Figure 15. Schematic for the Parameter Sweep

Click the Simulate button and plot S(1,1) and S(2,1) in the Data Display window. This shows how the filter response changes with the length of the first inductive line. Add a line marker at 2 GHz to see the S-parameters at this point. This is shown in Figure 16.

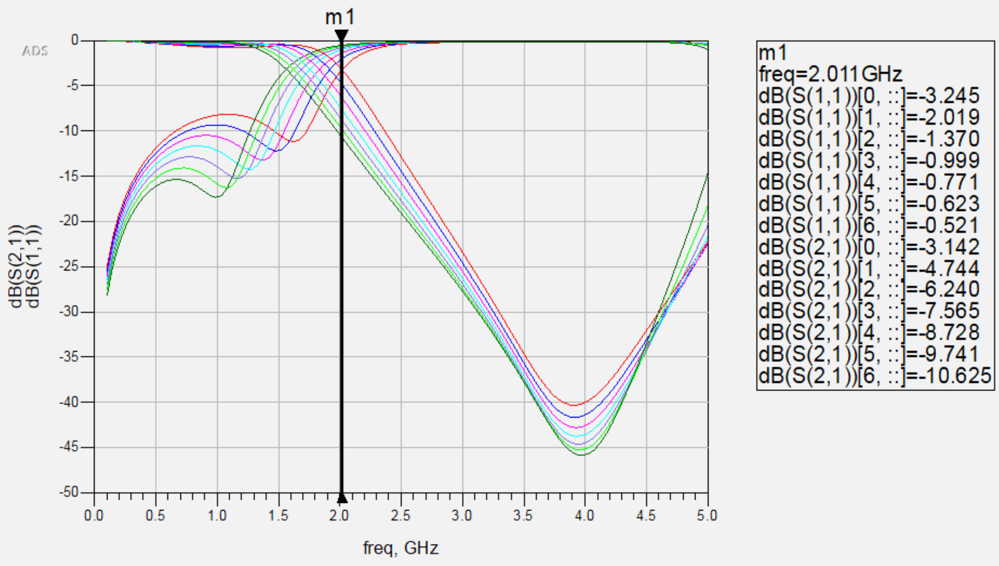

Figure 16. Plot showing the effect of the first inductive line length on the S-Parameters

From the plot, we can see that S(2,1) provides a 3 dB cutoff at 2 GHz for the first sweep value. Therefore, 6.145 mm seems to be the best value for L2.

In the schematic, disable the Parameter Sweep, and set **L2 = 6.145**. Perform the simulation again to see the filter response. The result is shown in Figure 17.

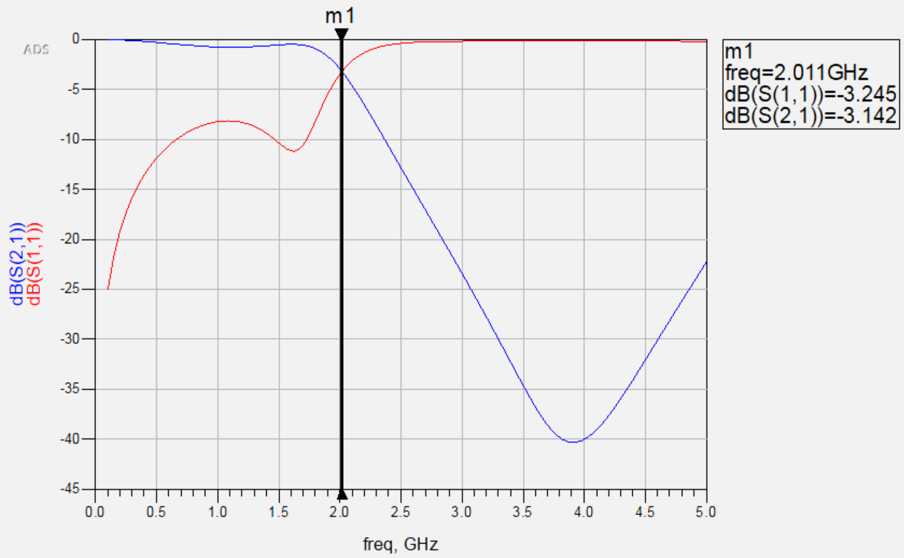

Figure 17. Plot showing the S-Parameters with the first inductive line length redefined

Results and Observations

Using theoretical values, the cutoff frequency for the distributed (stepped impedance) lowpass filter was about 1.68 GHz, far below the specification of 2 GHz. Further optimization was needed to meet the specification. By performing a parametric sweep, the first inductive line length was optimized. This optimization showed significant impact to the 3 dB cutoff frequency by reducing the length of the first inductive line. This raised the cutoff frequency to about 2 GHz. As this met the specification, no further optimization was needed.

Simulation of a Lumped and Distributed Bandpass Filter Using ADS

Typical Design

Upper Cutoff Frequency (f_{c1})	: 1.9 GHz
Lower Cutoff Frequency (f_{c2})	: 2.1 GHz
Ripple in Passband	: 0.5 dB
Order of the Filter	: 3
Type of Approximation	: Chebyshev

Prototype Values of the Filter

The prototype values of the filter for a Chebyshev approximation are calculated using the formulas:

$$\beta = \ln\left(\coth \frac{L_{Ar}}{17.37}\right) \text{ where } L_{Ar} \text{ is ripple in the passband}$$

$$\gamma = \sinh\left(\frac{\beta}{2n}\right)$$

$$a_k = \sin\left[\frac{(2k-1)\pi}{2n}\right], \ k = 1, 2, 3, \ldots, n$$

$$b_k = \gamma^2 + \sin^2\left(\frac{k\pi}{n}\right), \ k = 1, 2, 3, \ldots, n$$

$$g_1 = \frac{2a_1}{\gamma}$$

$$g_k = \frac{4a_{k-1}a_k}{b_{k-1}g_{k-1}}, \ k = 2, 3, \ldots, n$$

$$g_{n+1} = 1 \text{ for } n \text{ odd}$$

$$= \coth^2\left(\frac{\beta}{4}\right) \text{ for } n \text{ even}$$

The prototype values for the given specifications of the filter are:

$g_1 = 1.5963 = C_1$

$g_2 = 1.0967 = L_2$

$g_3 = 1.5963 = C_3$

$g_4 = 1.0000$

Lumped Model of the Filter

The lumped values of the bandpass filter after frequency and impedance scaling are given by:

$$L'_1 = \frac{L_1 Z_0}{\omega_0 \Delta}$$

$$C'_1 = \frac{\Delta}{L_1 Z_0 \omega_0}$$

$$L'_2 = \frac{\Delta Z_0}{\omega_0 C_2}$$

$$C'_2 = \frac{C_2}{Z_0 \Delta \omega_0}$$

$$L'_3 = \frac{L_3 Z_0}{\omega_0 \Delta}$$

$$C'_3 = \frac{\Delta}{L_3 Z_0 \omega_0}$$

Where

Δ is the fractional bandwidth $\Delta = (\omega_2 - \omega_1) / \omega_0$

$Z_0 = 50\ \Omega$

The resulting lumped element values are calculated to be:

$L_1^{'} = 63$ nH

$C_1^{'} = 0.1004$ pF

$L_2^{'} = 0.365$ nH

$C_2^{'} = 17.34$ pF

$L_3^{'} = 63$ nH

$C_3^{'} = 0.1004$ pF

The geometry of the lumped element bandpass filter is shown in Figure 18.

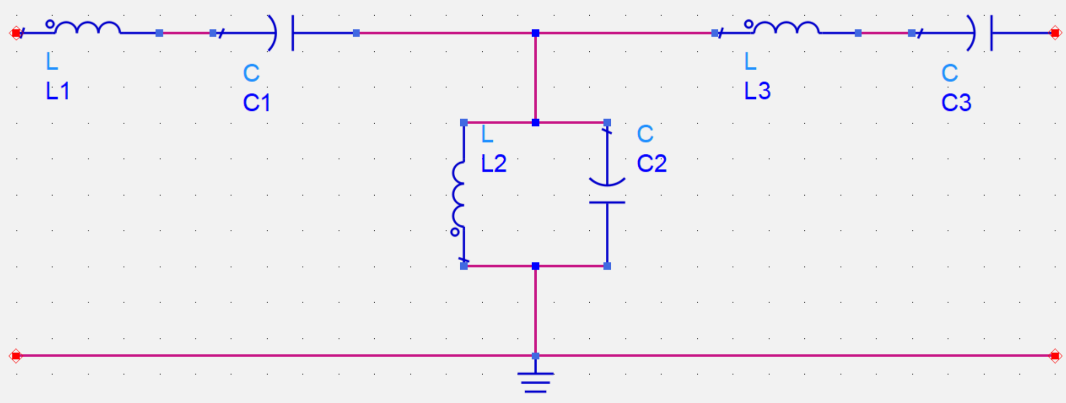

Figure 18. Geometry of lumped element bandpass filter

Distributed Model of the Filter

Calculate the value of J from the prototype values:

$$Z_0 J_1 = \sqrt{\frac{\pi\Delta}{2g_1}}$$

$$Z_0 J_n = \frac{\pi\Delta}{2\sqrt{g_{n-1} g_n}} \quad \text{for } n = 2, 3, \ldots, N$$

$$Z_0 J_{N+1} = \sqrt{\frac{\pi\Delta}{2 g_N g_{N+1}}}$$

Where

Δ is the fractional bandwidth $\Delta = (\omega_2 - \omega_1) / \omega_0$

$Z_0 = 50\ \Omega$

The values of odd and even mode impedances can be calculated as follows

$$Z_{0e} = Z_0\left[1 + JZ_0 + (JZ_0)^2\right]$$

$$Z_{0o} = Z_0\left[1 - JZ_0 + (JZ_0)^2\right]$$

Schematic Simulation Steps for the Lumped Bandpass Filter

Open a new schematic window and construct the lumped element bandpass filter, as shown in Figure 19. These values were calculated earlier and is the same geometry as the circuit in Figure 18. Setup the S-Paramater simulation from 1.0 GHz to 3.0 GHz with steps of 5 MHz.

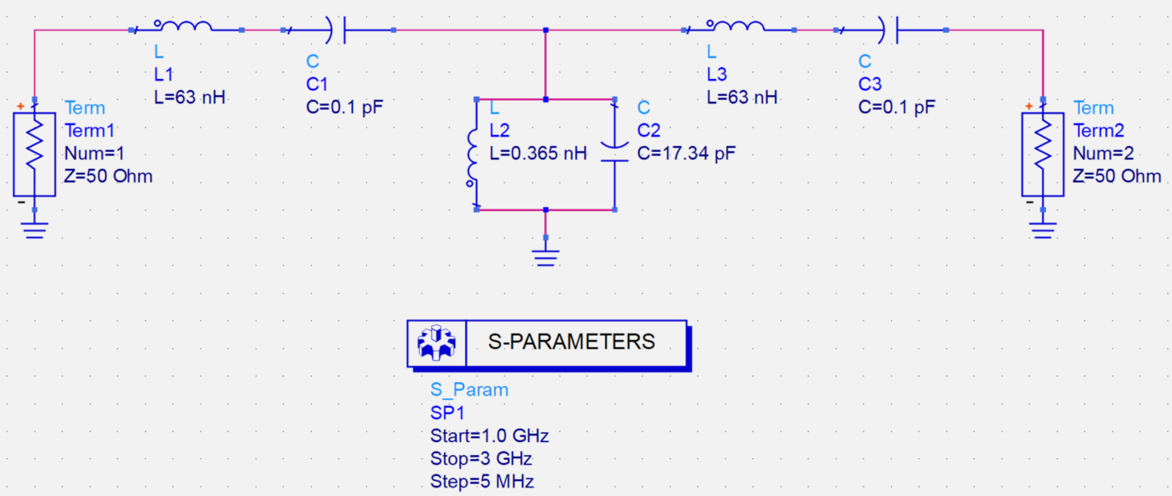

Figure 19. Lumped element bandpass filter schematic

Click the **Simulate** button and plot S(1,1) and S(2,1) in the data display window. Add markers for the lower and upper cutoff frequencies. The results are shown in Figure 20.

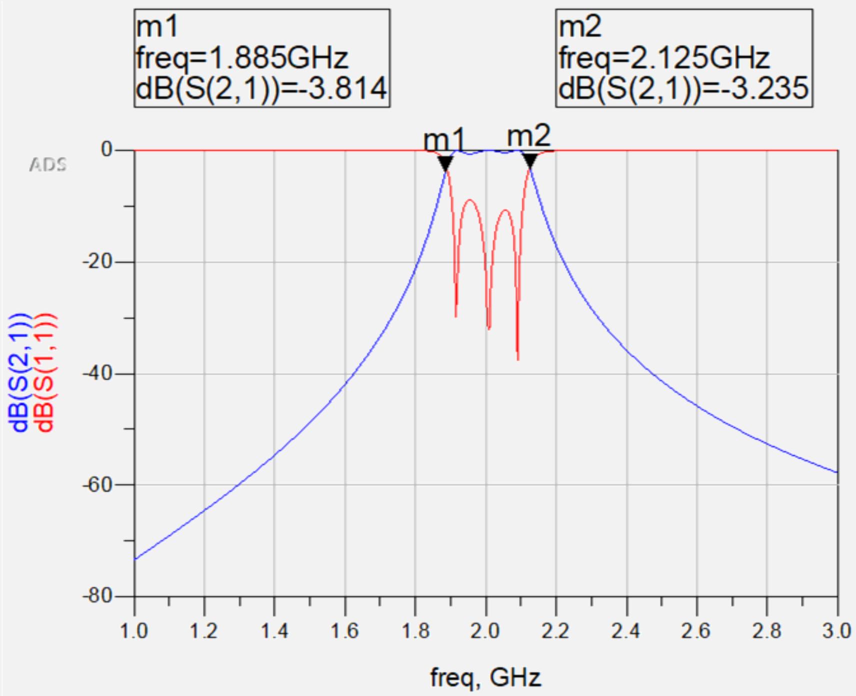

Figure 20. Lumped element bandpass filter S-Parameters

Results and Observations

The results are satisfactory for the lumped element design. The simulation shows that the design is in agreement with the specified lower and upper cutoff frequencies. The filter also has an appropriate roll off for a Chebyshev filter. However, to get more accurate results, the circuit should be simulated and optimized with the vendor component libraries. It may also be helpful to perform a yield analysis to determine how the results are impacted by the tolerance of the lumped components.

Layout Simulation Steps for the Distributed Bandpass Filter

Calculate the even mode and odd mode impedance values (Z_{0e} and Z_{0o}) of the bandpass filter using the design procedure given above. Synthesize the physical parameters (length, width, and spacing) for the coupled lines for a substrate thickness of 1.6 mm and dielectric constant of 4.6. This can be done using LineCalc. Make sure to change **Component Type** to **MCLIN**. This is shown in Figure 21.

The physical parameters of the coupled lines for the given values of Z_{0o} and Z_{0e} are shown below:

Substrate Parameters

Thickness: 1.6 mm

Dielectric constant: 4.6

Frequency: 2 GHz

Electrical length: 90 degrees

Section 1: Z_{0e} = 70.61 Ω, Z_{0o} = 39.24 Ω

Width = 2.290 mm

Spacing = 0.491 mm

Length = 20.723 mm

Section 2: Z_{0e} = 56.64 Ω, Z_{0o} = 44.77 Ω

Width = 2.803 mm

Spacing = 1.804 mm

Length = 20.255 mm

Section 3: Z_{0e} = 56.64 Ω, Z_{0o} = 44.77 Ω

Width = 2.803 mm

Spacing = 1.804 mm

Length = 20.255 mm

Section 4: Z_{0e} = 70.61 Ω, Z_{0o} = 39.24 Ω

Width = 2.290 mm

Spacing = 0.491 mm

Length = 20.723 mm

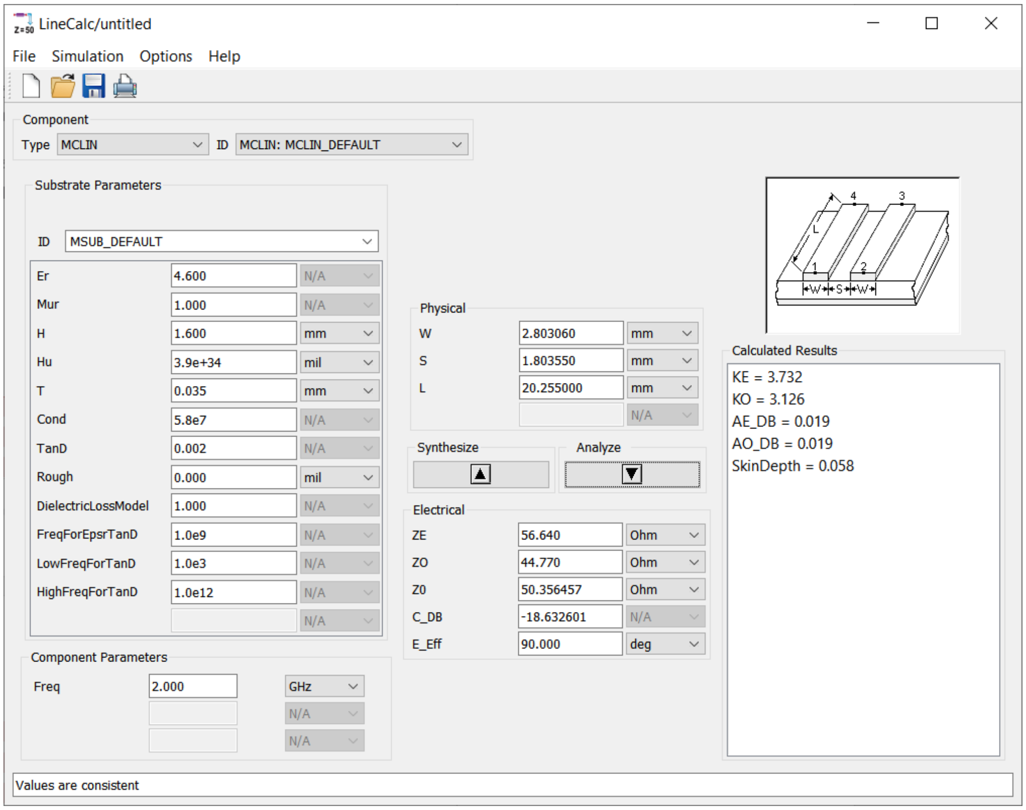

Figure 21. Using LineCalc to determine parameters

Calculate the width of the 50 Ω line using LineCalc. Make sure to reset **Component Type** to **MLIN**. The length of the line is chosen to be 5 mm for simplicity. The parameters for the 50 Ω line should be:

Width = 2.918 mm

Length = 5 mm

Now that the parameters have been determined, create a model of the distributed bandpass filter in a new layout window. The easiest way to do this is to use the library components. Select the **TLines-Microstrip** library. Use the **MCFIL** components for the coupled line sections and the **MLIN** components for the 50 Ω lines. Connect Pin 1 to the input and Pin 2 to the output. The finished layout is shown in Figure 22.

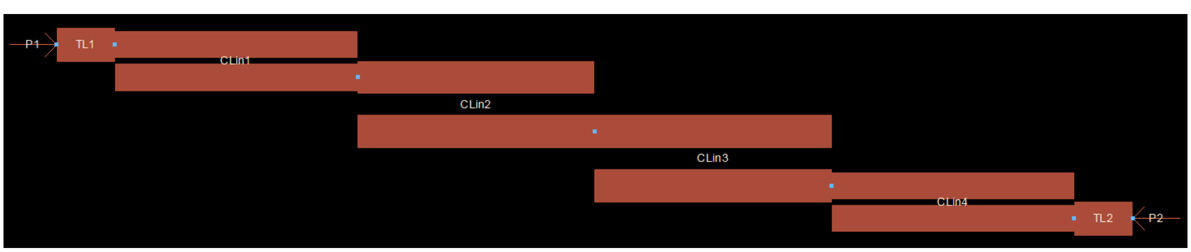

Figure 22. Layout of the distributed bandpass filter

Setup the EM simulation using the procedure defined earlier for 1.6 mm FR4 dielectric. Define the frequency plan as 1 GHz to 3 GHz with 101 number of points. Don't forget to turn on Edge Mesh under the **Options > Mesh** tab of the EM Setup window.

Once the simulation finishes, plot S(1,1) and S(2,1). Add markers to the -3dB bandwidth points to determine the lower and upper cutoff frequencies. The plot is shown in Figure 23. Note that the markers are not at -3 dB. That is because the passband ripple varies between approximately -1.0 dB and -1.5 dB, which will slightly impact the definition of the lower and upper cutoff frequencies.

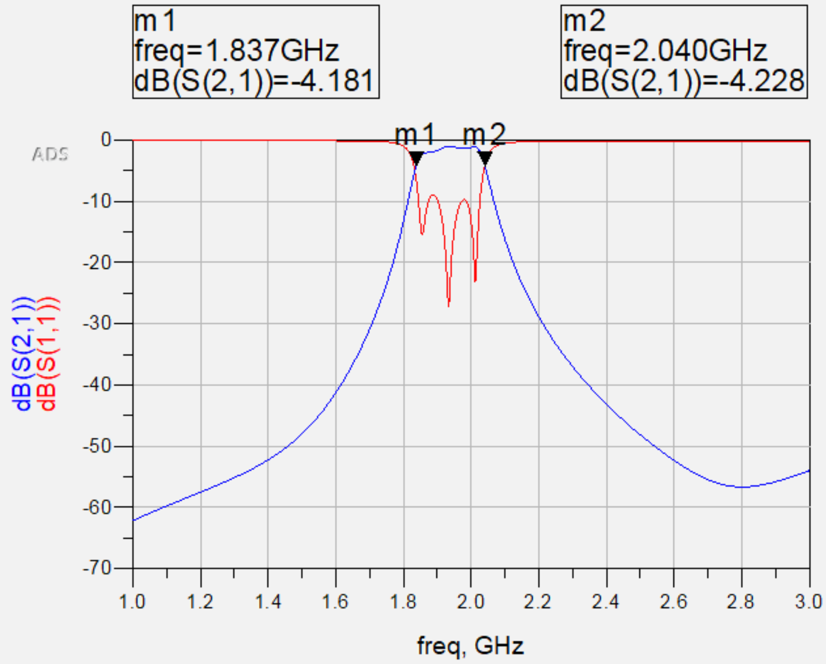

Figure 23. Plot of S-Parameters for distributed bandpass filter

Results and Observations

For the distributed design, the filter does not meet the specifications and the coupled lines need to be optimized (length, width, spacing). The odd and even impedance values were calculated from standard formulas using ideal assumptions and do not consider any electromagnetic or parasitic effects. The optimization can be done using a variety of tools inside ADS.

Conclusion

Congratulations! You have completed Microwave Discrete and Microstrip Filter Design. Check out more examples at www.keysight.com/find/eesof-ads-rfmw-examples.