Keysight EEsof EDA
Microwave Discrete and Microstrip Filter Design

Demo Guide

KEYSIGHT TECHNOLOGIES

Unlocking Measurement Insights

Theory

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 lowpass, highpass, 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 low pass 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 below.

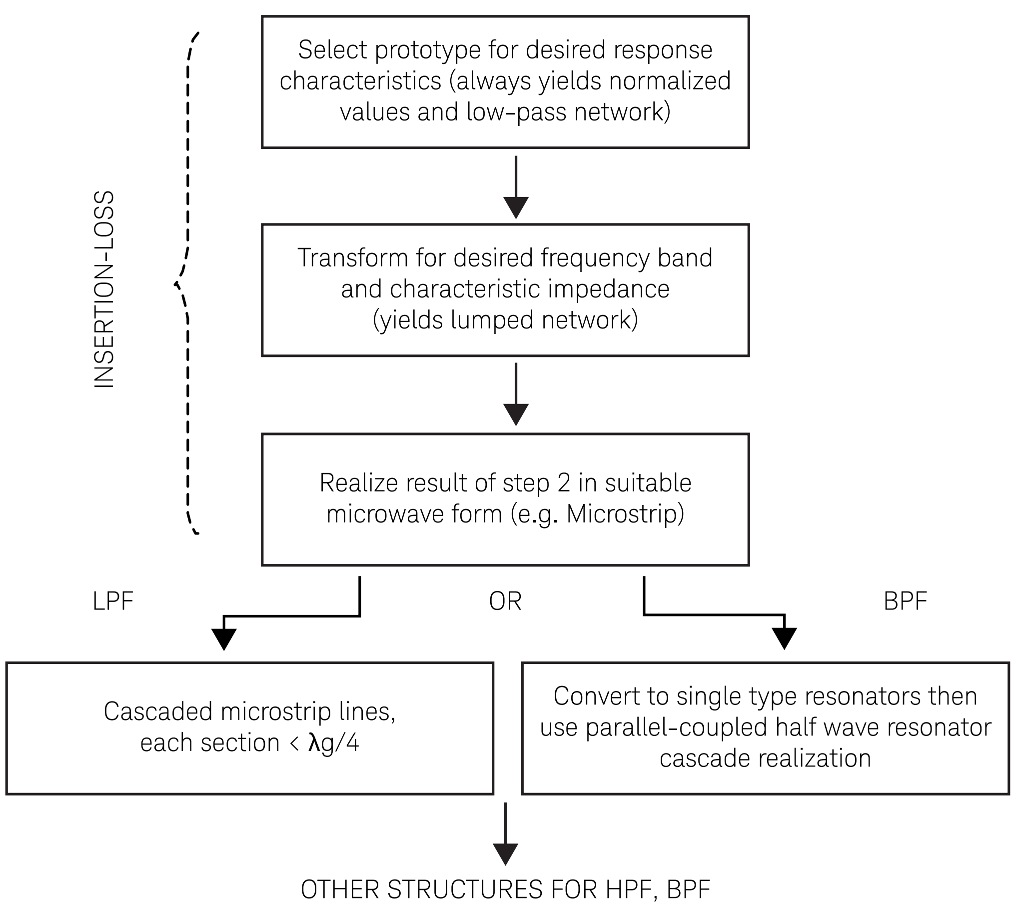


Figure 131.

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 proto type 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. Filter 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:

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

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

Chebyshev Approximation:

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

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

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

$\varepsilon = \{\text{Antilog}_{10} L_{Ar}/10\} - 1$ and L_{Ar} = 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 given equations.

Butterworth Approximation:

$g0 = 1$,

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

$g_{N+1} = 1$

Where, n is the order of the filter

Chebyshev Approximation:
The element values may be computed as follows

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

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

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

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

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

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

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

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

After computing the prototype values the prototype filter has to be transformed with respect to frequency and impedance to meet the specifications. The transformations can be done using the following equations.

For Lowpass filter:

After Impedance and frequency scaling:

$C'_k = C_k/R_0\,\omega_c$

$L'_k = R_0 L_k/\omega_c$ Where $R_0 = 50\ \Omega$

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

Length of capacitance section: $Z_l/R_0\ C_k$,
Length of inductance section: $L_k\ R_0/Z_h$

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

For bandpass filter:

Impedance and frequency scaling:

$L'_1 = L_1 Z_0/\omega_0 \Delta$

$C'_1 = \Delta/L_1 Z_0\ \omega_0$

$L'_2 = \Delta\ Z_0/\omega_0 C_2$

$C'_2 = C_2/Z_0\ \Delta\ \omega_0$

$L'_3 = L_3 Z_0/\omega_0\ \Delta$

$C'_3 = \Delta/L_3 Z_0\ \omega_0$

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

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 (LA(ω))
Type of approximation	: Butterworth
Order of the filter	: $LA(\omega) = 10\log_{10}\{1+\varepsilon\ (\omega/\omega_c)^{2N}$

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

Substituting the values of LA (ω), ω and ω_c, the value of N is calculated to be 4.

Prototype Values of the Lowpass Filter

The prototype values of the filter is calculated using the formula given by

$g_0 = 1$,

$g_k = 2\sin\{(2k-1)\pi/2N\}$ where k = 1,2,....N

and $g_{N+1} = 1$

The prototype values for the given specifications of the filter are

g1 = 0.7654 = C_1, g2 = 1.8478 = L_2, g3 = 1.8478 = C_3
& g4 = 0.7654 = L_4

Lumped Model of the Filter

The Lumped values of the Lowpass filter after frequency and impedance scaling are given by:

$C_k' = C_k / R_0 \omega_c$

$L_k' = R_0 L_k / \omega_c$ where R_0 is 50 Ω

The resulting lumped values are given by C_1 = 1.218 pF, L_2 = 7.35 nH, C_3 = 2.94 pF and L_4 = 3.046 nH

Distributed Model of the Filter

For distributed design, the electrical length is given by

Length of capacitance section (βLc) : $C_k Z_l / R_0$

Length of inductance section (βLi) : $L_k R_0 / Z_h$

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 βLc_1 = 0.153,
βLi_2 = 0.9239,

βLc_3 = 0.3695 and βLi_4 = 0.3827

Since $\beta = 2\pi/\lambda$, the physical lengths are given by

Lc_1 = 1.68 mm

Li_2 = 10.145 mm

Lc_3 = 4.057 mm and

Li_4 = 4.202 mm

Schematic Simulation Steps for Lumped Low Pass Filter

1. Open the Schematic window of ADS.
2. From the Lumped Components library select the appropriate components necessary for the lumped filer circuit. Click the necessary components and place them on the schematic window of ADS as illustrated.

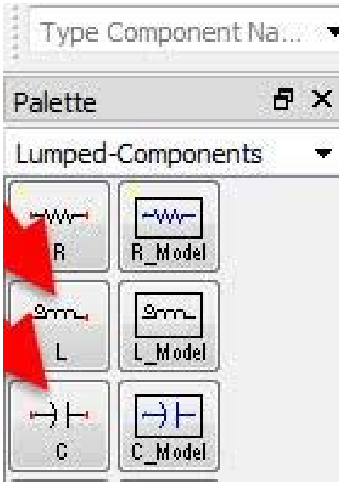


Figure 132.

3. Create the lumped model of the lowpass filter on the schematic window with appropriate lumped components and connect the circuit elements with wire . Enter the component values as calculated earlier.
4. Terminate both ports of the lowpass filter using terminations selected from the Simulation-S_Param library.
5. 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 enter Step Size = 49 MHz)

This completes the lumped model design of the filter as shown in the figure below.

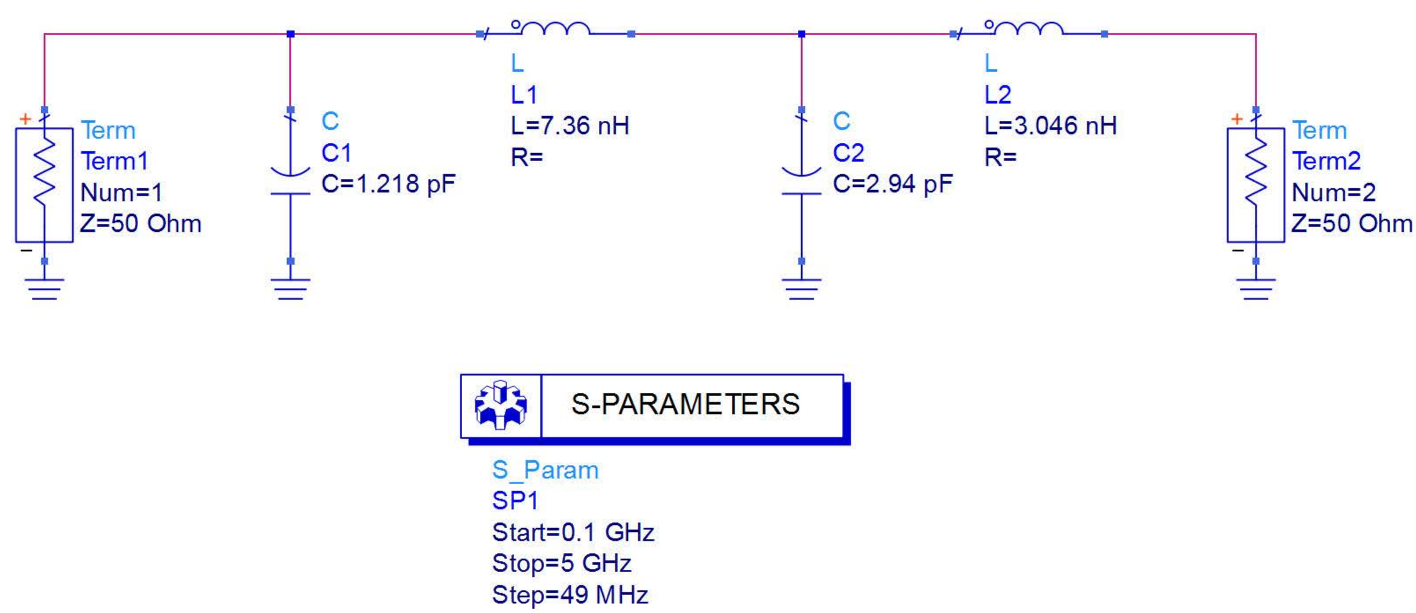


Figure 133.

6. Simulate the circuit by clicking **F7** or the simulation gear icon.
7. After the simulation is complete, ADS automatically opens the Data Display window displaying the results. If the Data Display window does not open, click **Window > New Data Display**. In the data display window, select a rectangular plot and this automatically opens the place attributes dialog box. Select the traces to be plotted (in our case S(1,1) & S(2,1) are plotted in dB) and click **Add>>**.
8. Click and insert a marker on S(2,1) trace around 2GHz to see the data display graph as shown below.

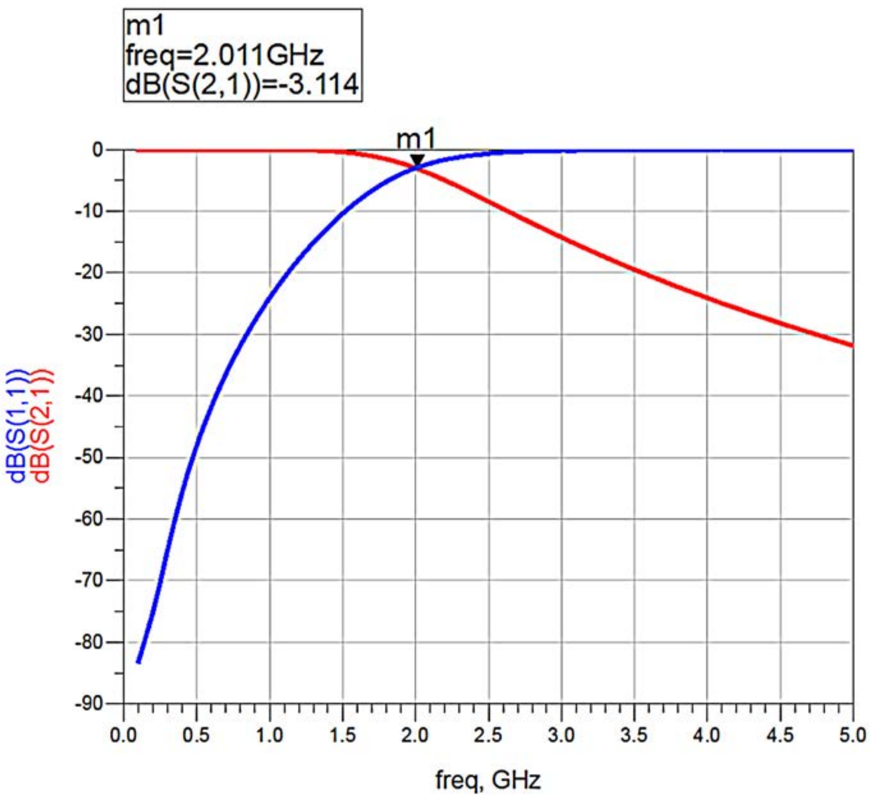


Figure 134.

Results and Observations

It is observed from the schematic simulation that the lumped model of the lowpass filter has a cutoff of 2 GHz and a roll off as per the specifications.

Layout Simulation Steps for Distributed Low Pass Filter

Calculate the physical parameters of the distributed lowpass filter using the design procedure given above. Calculate the width of the Z_l and Z_h transmission lines for the design of the stepped impedance lowpass filter. In this case Z_l = 10 Ω and Z_h = 100 Ω and the corresponding line widths are 24.7 mm and 0.66 mm respectively for a dielectric constant of 4.6 and a thickness of 1.6 mm.

Calculate the length and width of the 50 Ω line using the line calc (Tools->Line Calc->Start Line Calc) window of ADS as shown in figure below.

50 Ω Line input & output connecting line:
Width: 2.9 mm

Length: 4.5 mm

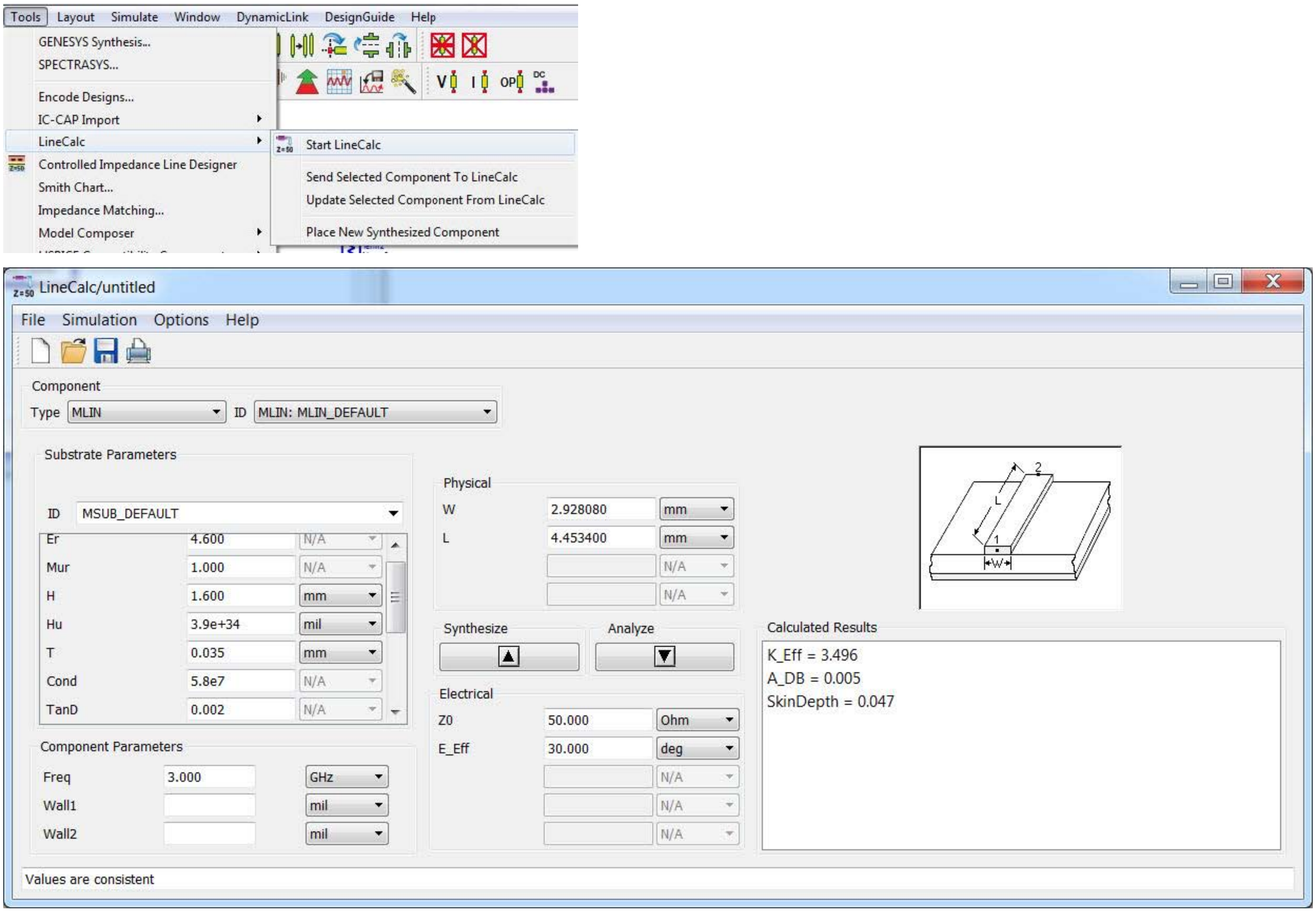


Figure 135.

Create a model of the lowpass filter in the layout window of ADS. The Model can be created by using the available library components **or** by drawing rectangles.

To create the model using library components, select the **TLines–Microstrip** library. Select the appropriate Microstrip line from the library and place it on the layout window as shown.

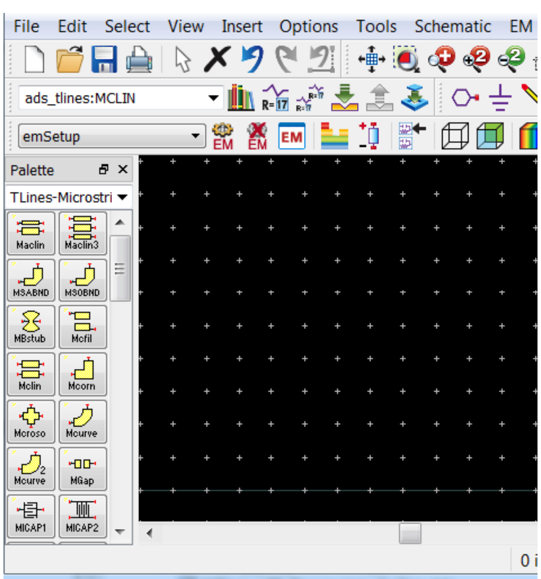


Figure 136.

Complete the model by connecting the transmission lines to form the stepped impedance lowpass filter as shown below based on the width & length calculations done earlier.

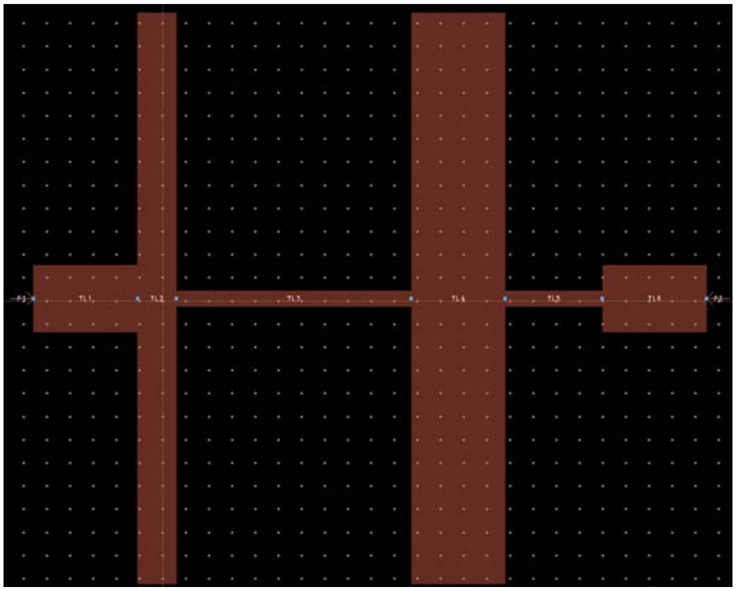


Figure 137.

Connect Pins at the input & output and define the substrate stackup and setup the EM simulation as described in the EM simulation chapter earlier. We shall use the following properties for the stackup:

Er=4.6
Height = 4.6 mm
Loss Tangent = 0.0023
Metal Thickness = 0.035 mm
Metal Conductivity = Cu (5.8E7 S/m)

In the EM setup window, go **to Options** > **Mesh** and turn on Edge Mesh.

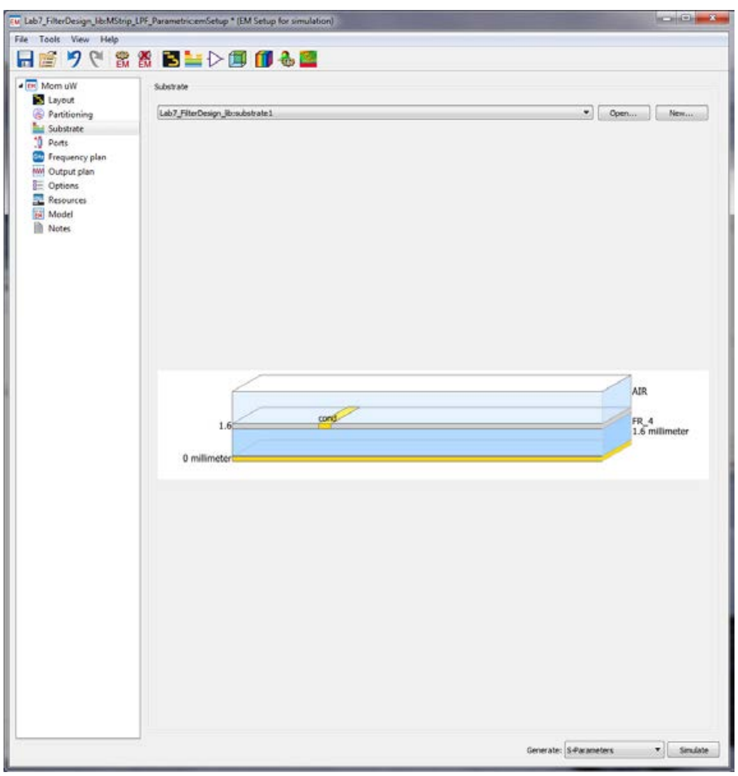


Figure 138.

Click the **Simulate** button and observe the S11 and S21 response.

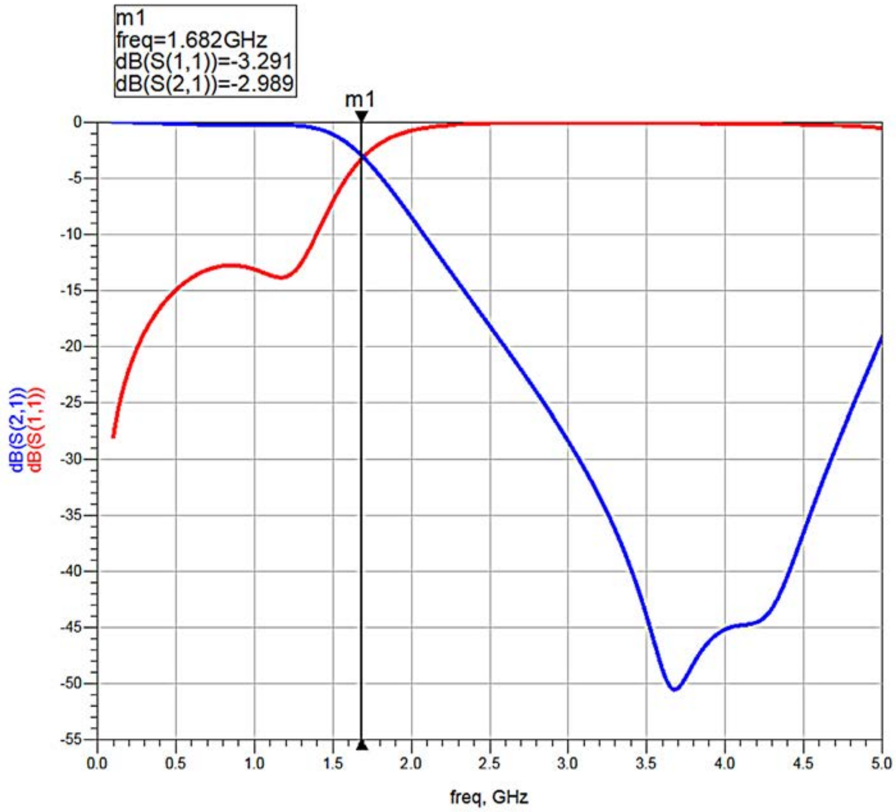


Figure 139.

It can be noted that the 3 dB cut-off has shifted to 1.68 GHz instead of 2 GHz as our theoretical calculations doesn't allow accurate analysis of open end effect and a sudden impedance change in the transmission lines, hence the lengths of the lines needs to optimized to recover the desired 2 GHz cutoff frequency specifications.

This optimization can be carried out using the Momentum simulator in ADS or by performing a parametric sweep on the lengths of Capacitive and Inductive lines.

Parametric EM Simulations in ADS

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

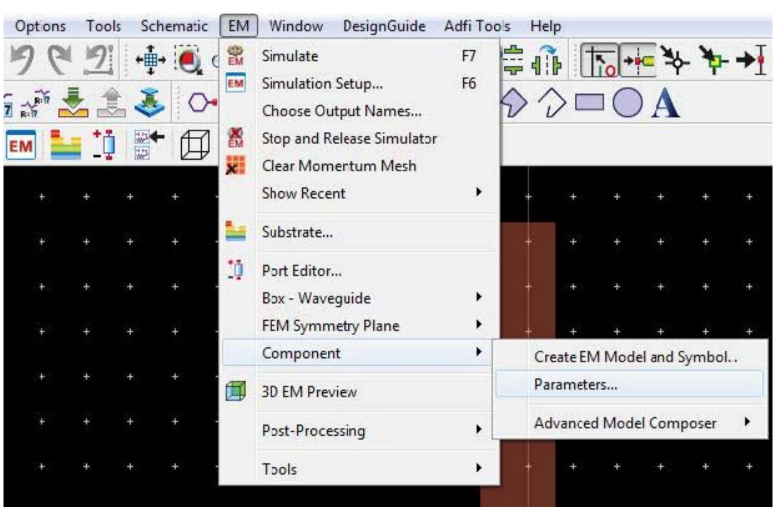


Figure 140.

In the parameter pop-up window, define 4 variables for capacitive and inductive lines and enter their nominal values alongwith the corresponding units and choose **Type** = **Subnetwork** as these parameters will be associated with Microstrip library components, which has parameterized artwork. If we are trying to parameterize the polygon/rectangle based components, then we can select the Nominal/Perturbed method which requires additional attention to the way components get parameterized.

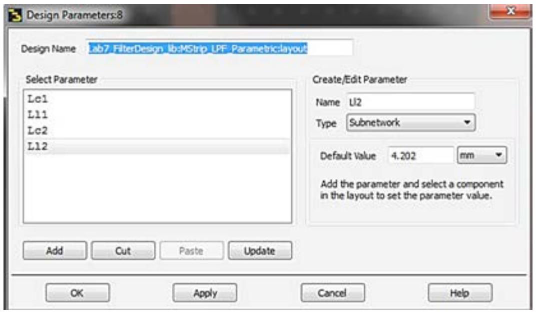


Figure 141.

Once the parameters have been added in the list, double-click the respective components and insert the corresponding variable names, please note that no units need to be defined here as we have already defined units in the variable parameter list. An example of one component has been shown below:

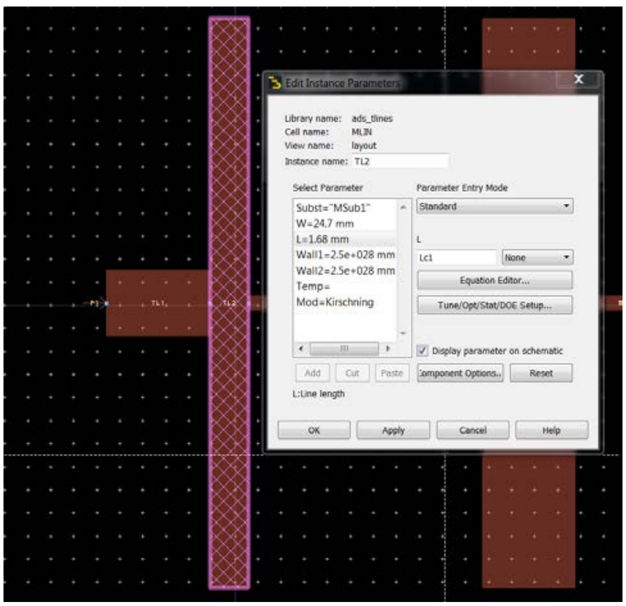


Figure 142.

After defining all the parameter values in the desired layout components we can create an EM model and symbol that can then be used for parametric EM cosimulation in the schematic. To create a parametric model and symbol for the layout, click the "**EM > Component > Create EM Model and Symbol**" option.

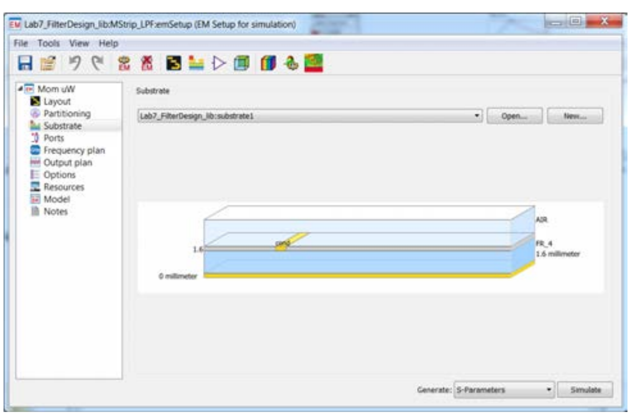


Figure 143.

Once done, observe the main ADS window where the name of the emmodel and symbol are displayed below the layout cell name as shown below:

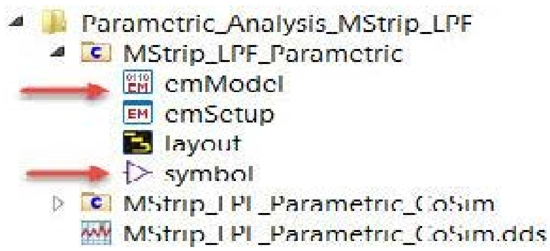


Figure 144.

Open a new schematic cell and drag and drop the emModel component to place it as subcircuit. You will notice the defined parameters being added to the emModel component, which can then be swept using the regular Parameter Sweep component in the ADS schematic as shown below. In this case, we have defined variables L1-L4 and assigned it to the emModel component. To start with, we sweep the length of L2 (1st inductive line) from 6.145 to 12.145 in steps of 1.

At this stage, we can decide to setup optimization and then optimize the layout component variables like any other circuit optimization, but please note that EM optimization will take longer as compared to circuit based optimization but produces more an accurate response as the EM simulation will be performed for every combination.

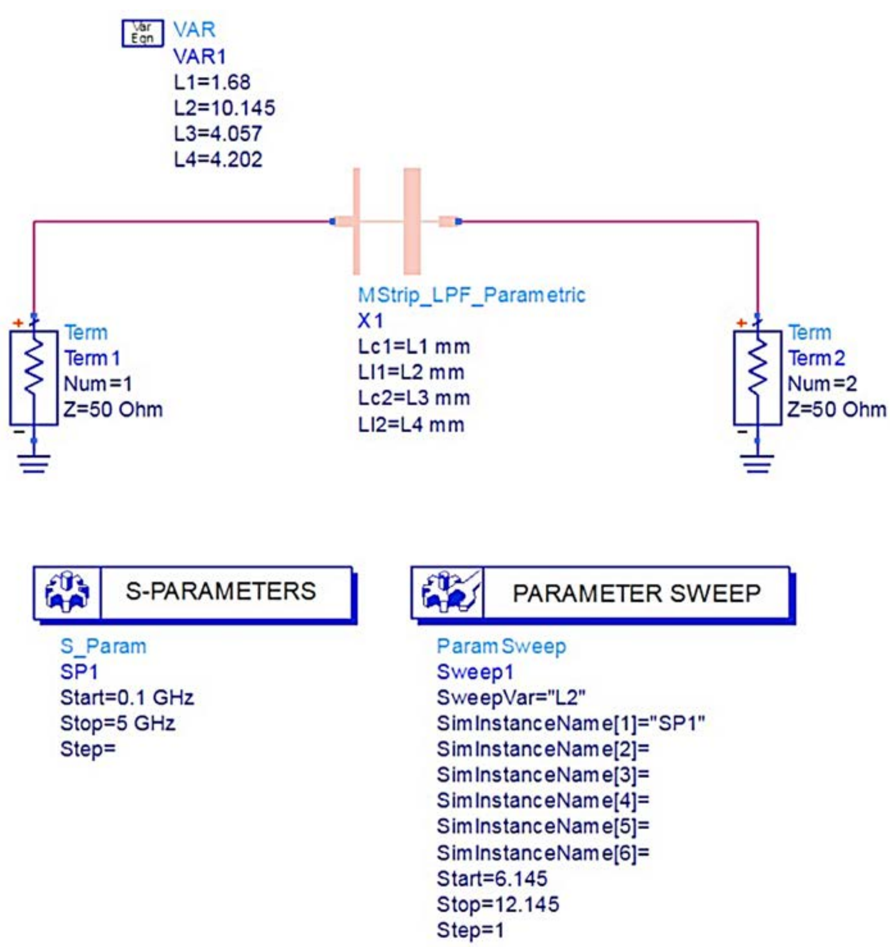


Figure 145.

Click the **Simulate** icon and plot the graph in the data display window to see how the filter response changes with the length of the 1st inductive line.

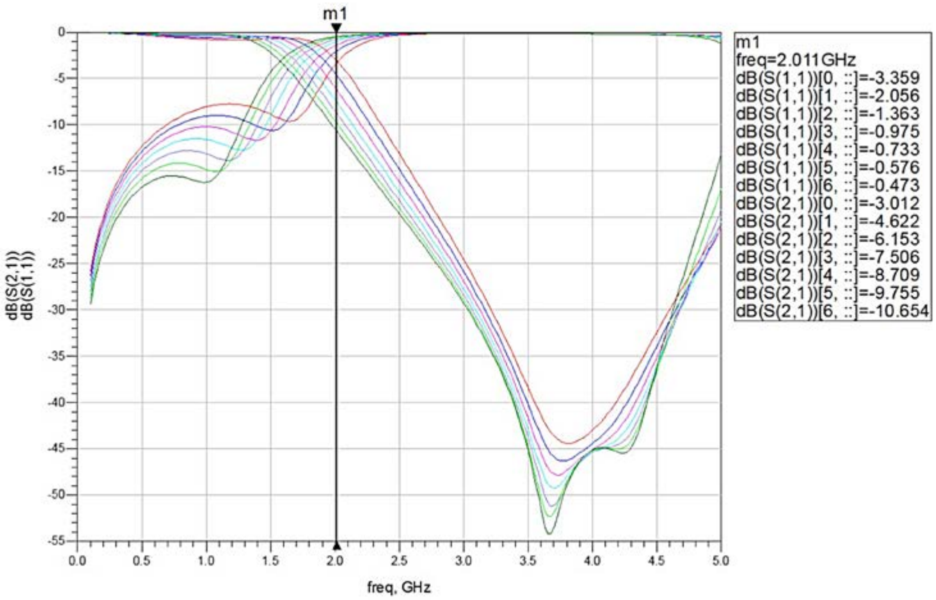


Figure 146.

From the data display, we can see that the 1st sweep value of L2 is providing a 3 dB cutoff at *2 GHz i.e. L2=6.145 mm* seems to be the correct value.

Disable the parameter sweep and change the value of *L2 = 6.145 mm* and perform the simulation again to see the filter response. The circuit can be EM optimized if better return loss is expected from the circuit.

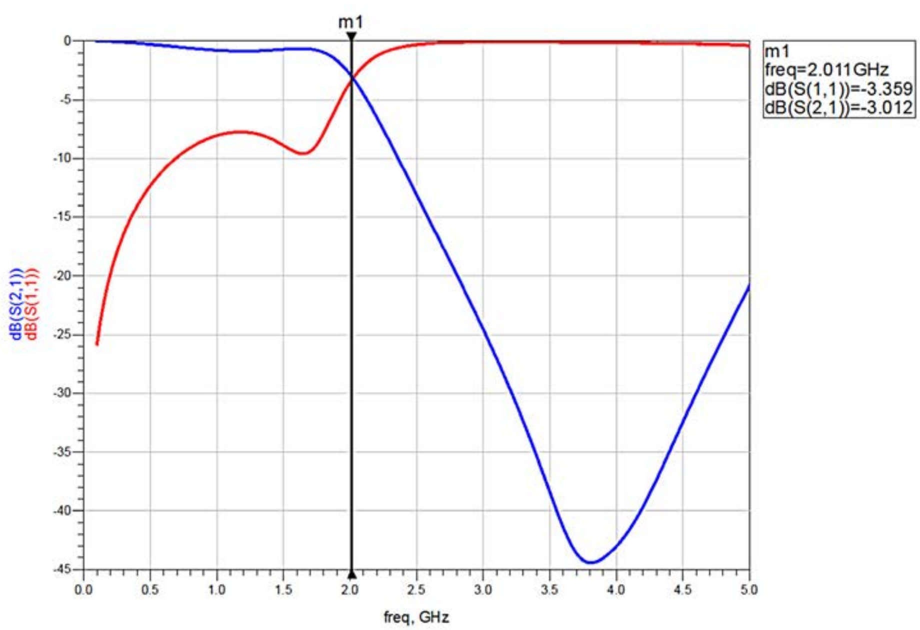


Figure 147.

Results and Observations

It is observed from the layout simulation that the Lowpass filter has a 3 dB cutoff frequency of 2 GHz after parametric EM analysis.

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 is calculated using the formulae given above.

The prototype values for the given specifications of the filter are
g1 = 1.5963, g2 = 1.0967 & g3 = 1.5963

Lumped model of the filter

The Lumped values of the Bandpass filter after frequency and impedance scaling are given by:

$L'_1 = L_1 Z_0/\omega_0 \Delta$

$C'_1 = \Delta / L_1 Z_0 \omega_0$

$L'_2 = \Delta Z_0/\omega_0 C_2$

$C'_2 = C_2 / Z_0 \Delta \omega_0$

$L'_3 = L_3 Z_0/\omega_0 \Delta$

$C'_3 = \Delta / L_3 Z_0 \omega_0$ where, Z_0 is 50 Ω

$\Delta = (\omega_2 - \omega_1)/\omega_0$

The resulting lumped values are given by:

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 the next figure.

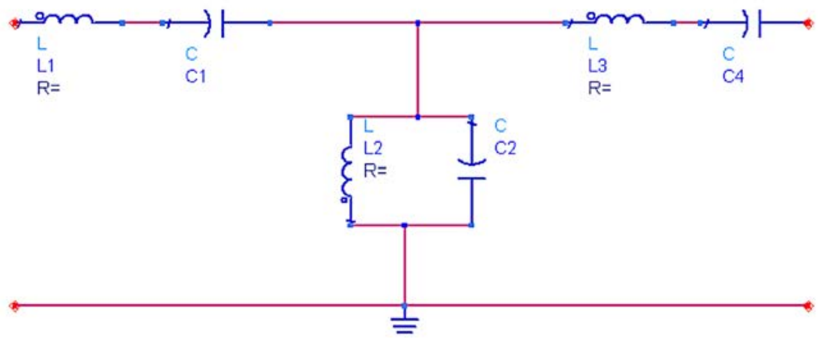


Figure 148.

Distributed Model of the Filter

Calculate the value of j from the prototype values as follows:

$$\left[Z_0 \; j_1 = \sqrt{\frac{\pi\Delta}{2 g_1}} \right]$$

$$Z_0 \, j_n = \frac{\pi\Delta}{2\sqrt{g_{n-1} g_n}} \qquad \text{For } n = 2, 3....N,$$

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

Where, $\Delta = (\omega_2 - \omega_1)/\omega_0$

Z_0 = Characteristic Impedance = 50 Ω

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

$$Z_{0e} = Z_0[1 + jZ_0 + (jZ_0)^2]$$

$$Z_{0o} = Z_0[1 + jZ_0 + (jZ_0)^2]$$

Schematic Simulation Steps for the Lumped Bandpass Filter

Open the Schematic window of ADS and construct the lumped bandpass filter as shown below. Setup the S-Parameter simulation from 1 GHz to 3 GHz with steps of 5 MHz (401 points).

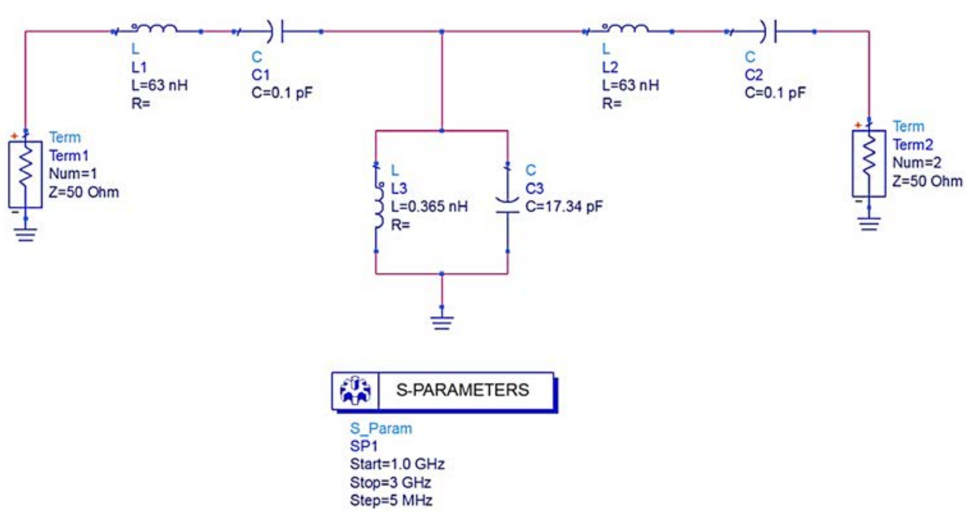


Figure 149.

Click the **Simulate** icon to observe the graph as illustrated:

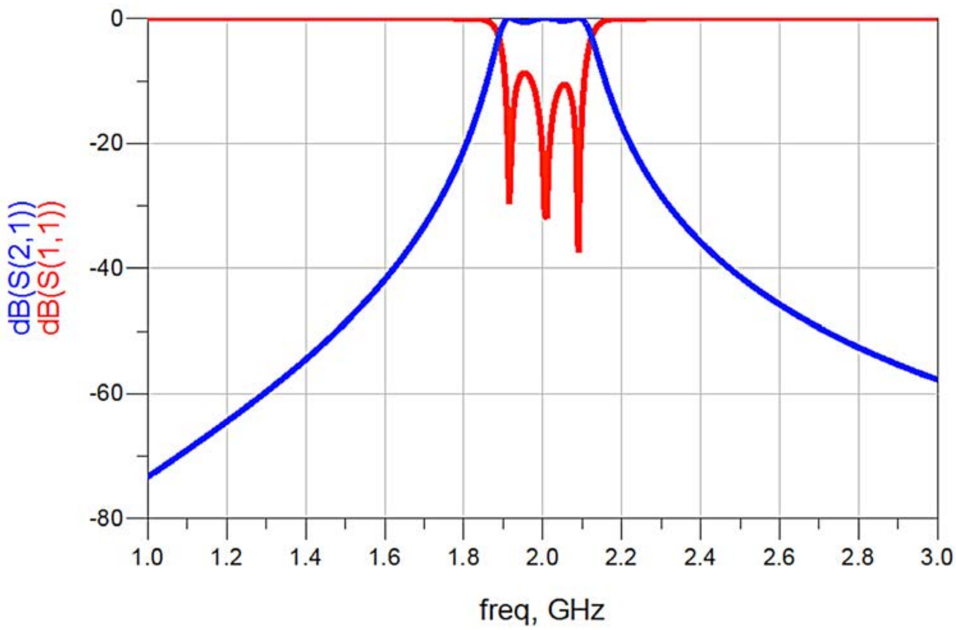


Figure 150.

Results and Observations

It is observed from the schematic simulation that the lumped model of the bandpass filter has an upper cutoff at 1.9 GHz, lower cutoff at 2.1 GHz and a roll off as per the specifications.

Layout Simulation Steps for the Distributed Bandpass Filter

Calculate the odd mode and even mode impedance values (Zoo & Zoe) of the bandpass filter using the design procedure given above. Synthesize the physical parameters (length & width) for the coupled lines for a substrate thickness of 1.6 mm and dielectric constant of 4.6.

The physical parameters of the coupled lines for the given values of Zoo and Zoe are given as follows:

Substrate thickness: 1.6 mm
Dielectric constant: 4.6
Frequency: 2 GHz
Electrical length: 90 degrees

Section 1: Zoo = 36.23, Zoe = 66.65
Width = 2.545
Length = 20.52
Spacing = 0.409

Section 2: Zoo = 56.68, Zoe = 44.73
Width = 2.853
Length = 20.197
Spacing = 1.730

Section 3: Zoo = 56.68, Zoe = 44.73
Width = 2.853
Length = 20.197
Spacing = 1.730

Section 4: Zoo = 36.23, Zoe = 66.65
Width = 2.545
Length = 20.52
Spacing = 0.409

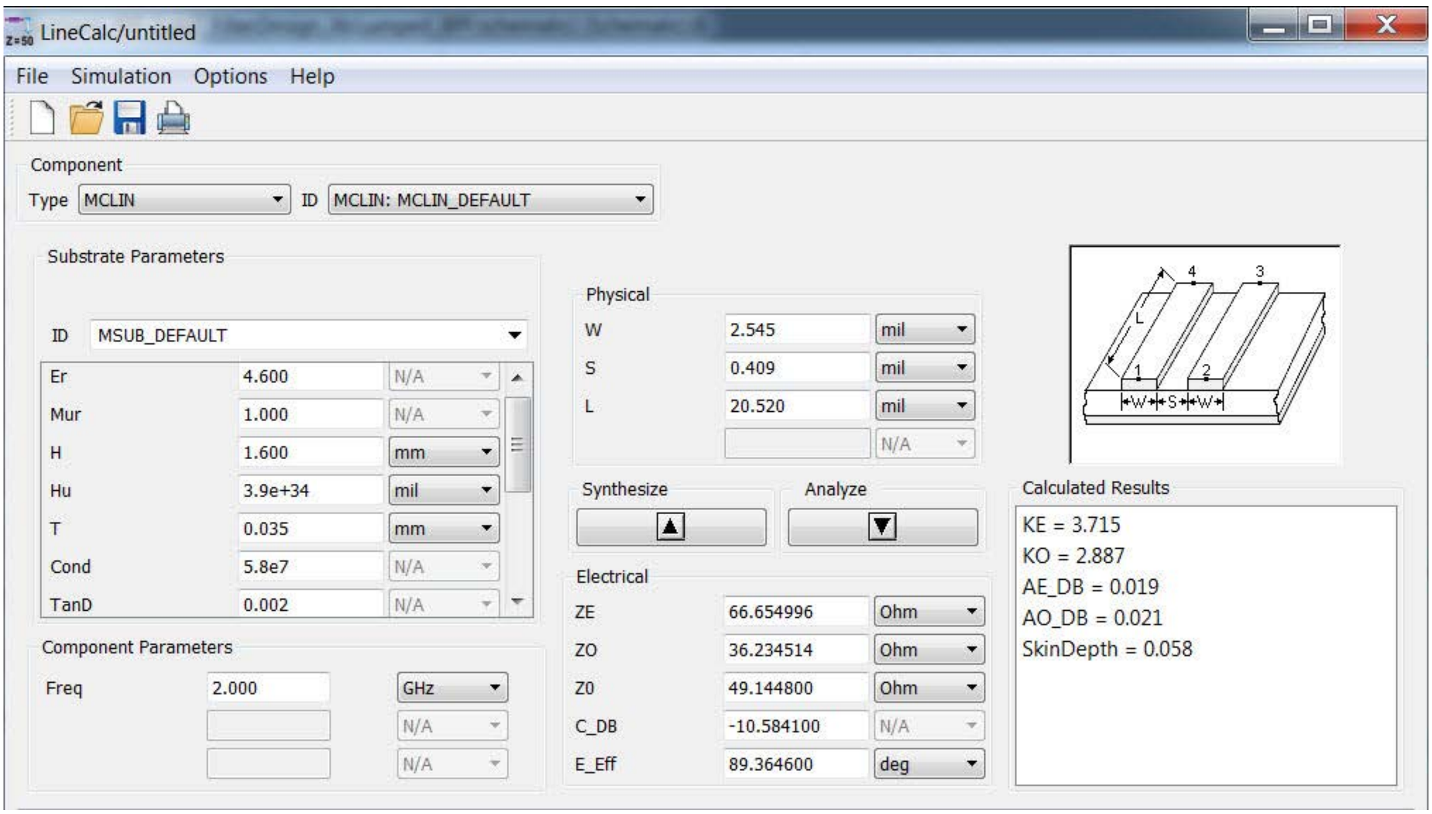


Figure 151.

Calculate the length and width of the 50 Ω line using the **linecalc** window of ADS as done earlier.

- 50 Ω Line:
- Width: 2.9 mm
- Length: 5 mm

Create a model of the bandpass filter in the layout window of ADS. The Model can be created by using the available library components or by drawing rectangles.

To create the model using library components select the MCFIL from TLines–Microstrip library. Select the appropriate kind of Microstrip line from the library and place it on the layout window as shown in Figure 152.

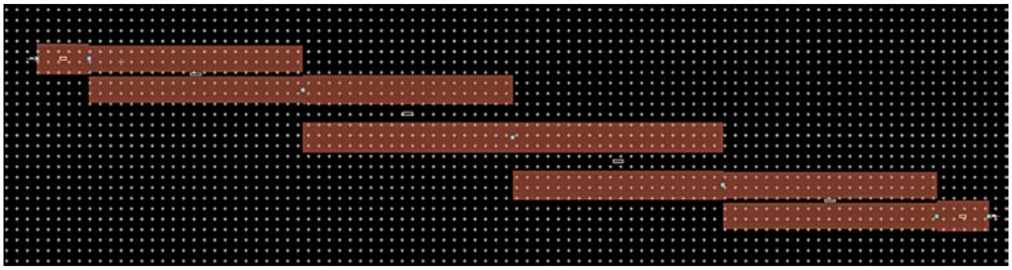


Figure 152.

Setup the EM simulation using the procedure defined earlier for 1.6 mm FR4 dielectric and perform a Momentum simulation from 1 GHz to 3 GHz and don't forget to turn on Edge Mesh from **Options > Mesh** tab of the **EM Setup** window.

Once the simulation finishes, plot the S11 and S21 response of the BPF as shown below:

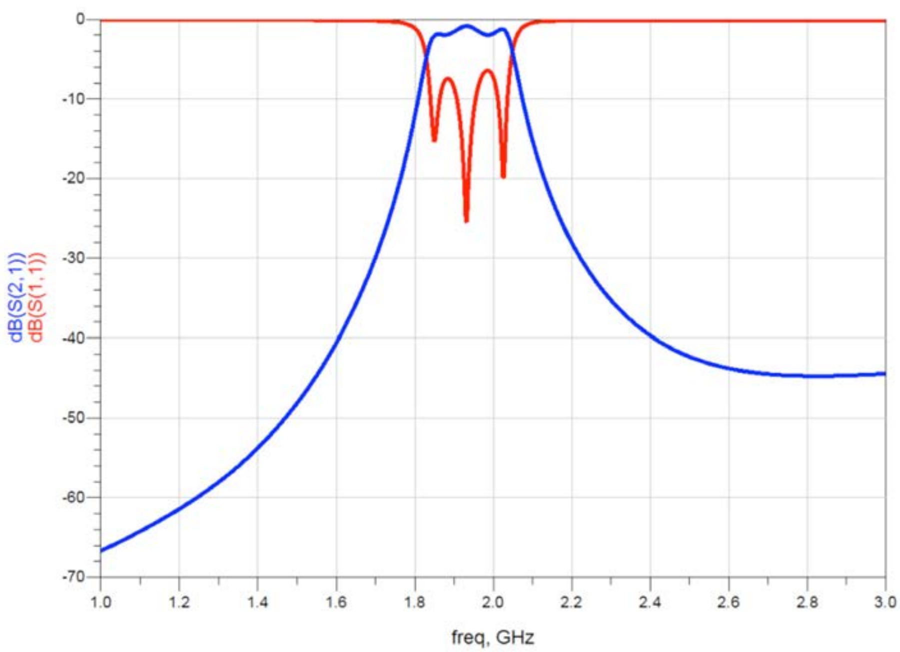


Figure 153.

Results and Observations

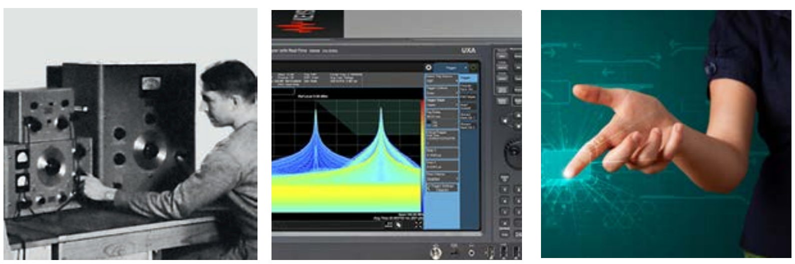
The results are good in the lumped element filter but the circuit needs to be simulated and probably needs to be re-optimized with the Vendor component libraries and we need to perform a Yield analysis simulation to take note of the performance variation, which may be caused due to tolerances of the lumped components.

For the distributed filter design, we can further optimize the design using the circuit simulator or Momentum EM simulator to obtain better bandpass filter characteristics, if desired, as the EM simulation is showing little degraded performance for fhs BPF.

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

Evolving

Our unique combination of hardware, software, support, and people can help you reach your next breakthrough. We are unlocking the future of technology.



From Hewlett-Packard to Agilent to Keysight

myKeysight

myKeysight
www.keysight.com/find/mykeysight
A personalized view into the information most relevant to you.

For more information on Keysight Technologies' products, applications or services, please contact your local Keysight office. The complete list is available at:
www.keysight.com/find/contactus

Americas	
Canada	(877) 894 4414
Brazil	55 11 3351 7010
Mexico	001 800 254 2440
United States	(800) 829 4444
Asia Pacific	
Australia	1 800 629 485
China	800 810 0189
Hong Kong	800 938 693
India	1 800 11 2626
Japan	0120 (421) 345
Korea	080 769 0800
Malaysia	1 800 888 848
Singapore	1 800 375 8100
Taiwan	0800 047 866
Other AP Countries	(65) 6375 8100
Europe & Middle East	
Austria	0800 001122
Belgium	0800 58580
Finland	0800 523252
France	0805 980333
Germany	0800 6270999
Ireland	1800 832700
Israel	1 809 343051
Italy	800 599100
Luxembourg	+32 800 58580
Netherlands	0800 0233200
Russia	8800 5009286
Spain	800 000154
Sweden	0200 882255
Switzerland	0800 805353
	Opt. 1 (DE)
	Opt. 2 (FR)
	Opt. 3 (IT)
United Kingdom	0800 0260637

For other unlisted countries:
www.keysight.com/find/contactus
(BP-06-08-16)

KEYSIGHT TECHNOLOGIES
Unlocking Measurement Insights

Published in USA, June 16, 2016
5992-1624EN
www.keysight.com