

Lecture X-2

Addendum by Thomas MacDonald

Coupled Line Bandpass Filter

In this supplementary lecture, we will study how to use Advanced Design System (ADS) 2009 to model a coupled line bandpass filter. This lecture will focus on use of the Libra model, MoM, and FEM capabilities of ADS.

Part I Getting Started

1. Problem statement

In this lecture, we will make be making extensive use of all of ADS's capabilities. First, we will explore the schematic capture feature of ADS. From there, we will build a model of a transmission line system based on the Libra models provided to us through ADS.

After the use of schematic capture, we will explore the layout capabilities of ADS. Through the layout tool, we are able to not only calculate the parameters through MoM but also through FEM. This is achieved by the Momentum engine and EMDS accordingly. EMDS is Agilent's electromagnetic simulator. This simulator has been succeeded by EMPro.

2. Starting ADS

Simply double-click the ADS 2009 64-bit edition icon on the desktop if it is present. If it is not then go to:

Start → All Programs → Advanced Design System 2009 →
Advanced Design System 2009 (64-Bit Simulations)

This is where things get tricky as ADS has a minor quirk associated with it. Click "Create a New Project". It should say as the name "M:\". Here is where you have to be careful. If you want to create a folder for your project you would add it here.

For example:

- M:\ADS\project1
- M:\project1

The trick here is that if you use the browse feature and want to use a folder you already created, say M:\ADS, the program will not allow you to do that because it wants to always create a folder for your project. Thus, ideally, you would use the Browse button to create your main ADS

collections folder. Then select ADS to report M:\ADS. This is achieved by highlighting your recently created ADS folder and then selecting choose. This should write “M:\ADS” in your Name box. Finally, alter the path to something that you will remember. In this case, M:\ADS\coupledLineFilter is what will be used.

Note:

Here is a great time to change the units you prefer to work in. If you want to work in millimeters, set that as ADS Standard Length Unit. If you prefer to work in mils, then select that. Make this easy for YOU.

Clicking OK will bring up the schematic capture window with an alert. ADS wants to help you with what you are simulating. If you are uncertain about how to do something, it can help you greatly with building a circuit or setting up a simulation. However, for this situation, we are professionals and as such do not need ADS’s help. Thus, select the radio button of “No help needed” and click Finish.

Your screen should look similar to this:

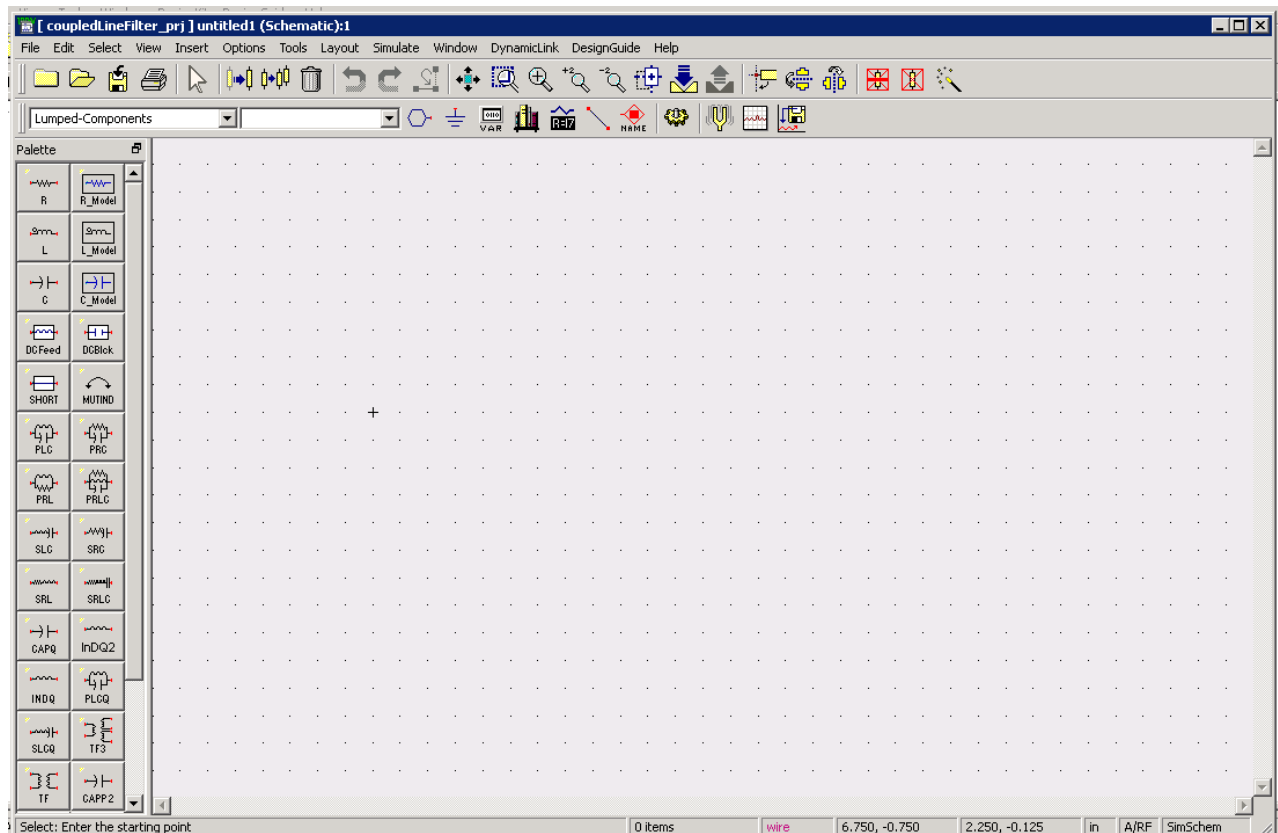
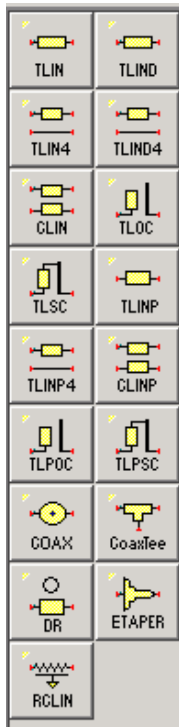


Figure 1. ADS Schematic Capture

Part II Ideal Coupled Transmission Lines

1. Navigating ADS

Select from the dropdown menu that says “Lumped Components” and scroll down to “TLines–Ideal”. You should see in the Palette something similar to this:



As can be seen in the figure to the left, there are multiple different types of ideal transmission lines that can be implemented by ADS.

A few of these are:

- Name – Description – Parameters
- TLIN – Standard transmission line (Z_0 , E, F)
- TLIND – Delay specified transmission line (Z_0 , Delay)
- TLIN4 – 4 terminal transmission line model (Z_0 , E, F)
- CLIN – Coupled Transmission Lines (Z_e , Z_o , E, F)
- TLOC – Transmission line open circuit termination (Z_0 , E, F)
- TLSC – Transmission line short circuit termination (Z_0 , E, F)

In this case, Z_0 is the characteristic impedance of your system. Generally, this is 50 Ohms. 50 Ohms was selected in the early days of electrical engineering as a compromise between 30 and 77 Ohms. As 30 Ohms was calculated to be the best for power handling while 77 Ohms on the other hand was calculated to be the lowest loss. The geometric mean between the two is 48 Ohms. This was of course calculated for an air dielectric.

Figure 2. Ideal TLine Palette

E is the electrical length of the transmission line. A quarter wavelength line is 90 degrees long. Thus, you would enter 90 degrees if you needed a quarter wavelength line. ADS is built around using degrees. You can use radians if you choose, this is achieved by typing `deg(your_value_here)` instead of writing in degrees. F is the frequency used in Hertz. However, you can enter things like “1 GHz” if you so choose. Delay is measured in seconds, however, you can enter in an argument like “1 ns” and it will behave as such.

Z_e and Z_o are the even and odd mode impedances that can be calculated. The calculations for these impedances will be included later in the lecture. The difference between even and odd mode is beyond the scope of this lecture and needs its own lecture to be completely characterized.

To edit these, you may either edit directly by clicking on the value that you want to change as demonstrated here:

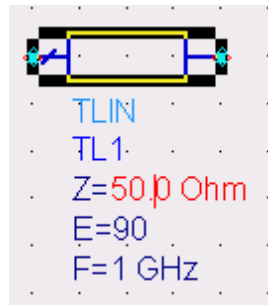


Figure 3. Transmission line parameter edit example

This is the optimal way to type in exactly what you want. If you happen to want a 75 GOhm characteristic impedance, you may type in 75 GOhm. However, this is by no means to guarantee that a 75 GOhm characteristic impedance will work.

On the other hand, you can double click the component to bring up the following menu:

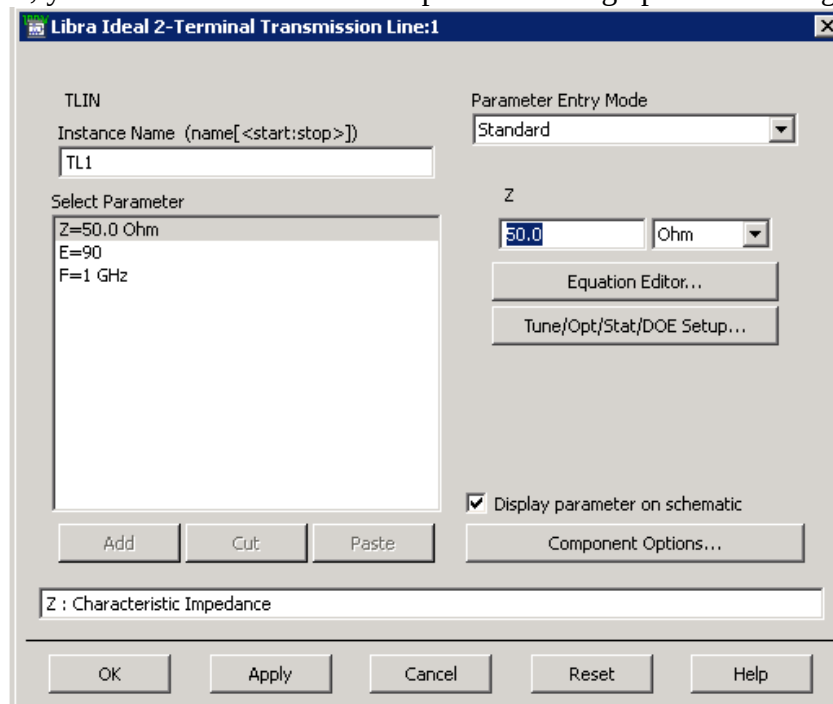


Figure 4. Transmission line parameter edit menu example

Here, instead of typing in “50 Ohm”, you would type 50 and then select a different unit from the drop down menu. This is an ideal place to write $\deg(\text{your_radian_value})$ inside of. Bear in mind that “your_radian_value” is literally the number that you want to enter. However, ADS does support variables.

2. Calculating Even and Odd Mode Impedances

As previously mentioned, even and odd mode impedances are a topic that need their own lecture to be justly explained. However, for even and odd mode impedances, the following always holds true:

$$Z_0 = \sqrt{Z_e * Z_o} \quad (1)$$

Thus, it can be seen that while the even and odd mode impedances may differ from the conventional characteristic impedance in appearance, they are still interrelated with one another. To calculate these impedances, we actually have to go through a process.

First we define the bandwidth,

$$BW = \frac{\omega_U - \omega_L}{\omega_0} \quad (2)$$

From this we have to defer to a table according to how to build the filter, included here are the g coefficients for Butterworth and 0.5 dB Chebyshev filters.

N	g ₁	g ₂	g ₃	g ₄	g ₅	g ₆
1	2	1				
2	1.4142	1.4142	1			
3	1	2	1	1		
4	0.7654	1.8478	1.8478	0.7654	1	
5	0.6180	1.6180	2	1.6180	0.6180	1

Table 1. Maximally Flat (Butterworth / Binomial) Filter Coefficients

N	g ₁	g ₂	g ₃	g ₄	g ₅	g ₆
1	0.6986	1				
2	1.4029	0.7071	1.9841			
3	1.5963	1.0967	1.5963	1		
4	1.6703	1.1926	2.3661	0.8419	1.9841	
5	1.7058	1.2296	2.5408	1.2296	1.7058	1

Table 2. Chebyshev 0.5dB Filter Coefficients

Other types of filters exist, however, these are just samples of the coefficients used to calculate even and odd mode impedances. Next, a factor has to be calculated that makes the overall even and odd mode impedance calculations in a nicer form. These coefficients are defined as below:

$$J_{0,1} = \frac{1}{Z_0} \sqrt{\frac{\pi BW}{2g_0g_1}} \quad (3)$$

$$J_{i,i+1} = \frac{1}{Z_0} \frac{\pi BW}{2\sqrt{g_i g_{i+1}}} \quad (4)$$

$$J_{N,N+1} = \frac{1}{Z_0} \sqrt{\frac{\pi BW}{2g_N g_{N+1}}} \quad (5)$$

From these factors, we are able to calculate the even and odd mode impedances as given:

$$Z_o|_{i,i+1} = Z_0[1 - Z_0 J_{i,i+1} + (Z_0 J_{i,i+1})^2] \quad (6)$$

$$Z_e|_{i,i+1} = Z_0[1 + Z_0 J_{i,i+1} + (Z_0 J_{i,i+1})^2] \quad (7)$$

Armed with this information, we are now able to proceed with our simulations after being armed with these. For this example, we will create a 3rd order bandpass filter. This filter will be designed for 1.6 GHz as this is currently of interest. All of these lines have to be $\lambda/4$ long. That means we need to enter 90 degrees as the electrical length parameter.

3. Laying out the schematic

With ADS up and looking at schematic capture, while the toolset is current set for TLines – Ideal, we click the CLIN from the Palette and placed them until we have 4 CLINs. For visual reasons, please arrange them as shown below:

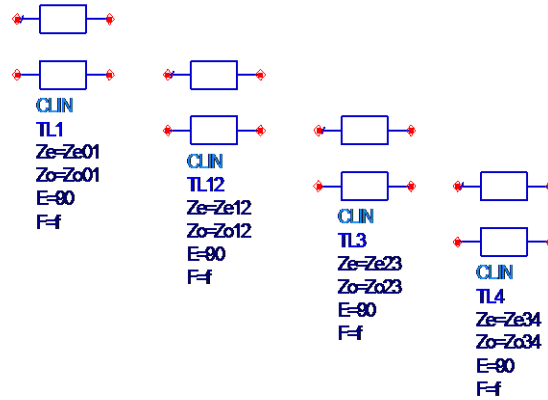


Figure 5. Preferred layout for coupled line filter

Now, to connect them, simply hit CTRL+W, which normally closes a program, and that will enable the wire connection mode. From there, simply click to connect the nodes. Please connect them in this fashion.

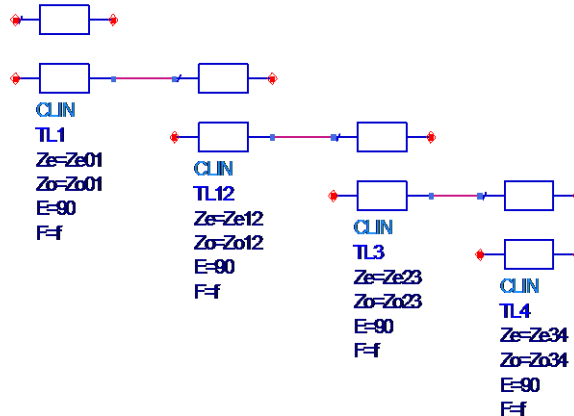


Figure 6. Preferred layout with wires connecting each element

As shown in the previous figures, instead of numbers, each of the variables used are parameterized. For simplicity, use these values instead:

	Odd Impedances	Even Impedances
Z01	38.30210065	100.9678075
Z12	39.97130504	67.7393234
Z23	39.97130504	67.7393234
Z34	38.30210065	100.9678075

Table 3. Odd and even mode impedances for the filter

4. Setting up the Simulation

Now that the basic structure has been set up, we have to establish the simulation configuration and the tests we want to run on our bandpass filter. Since the bandpass filter was designed for 1.6 GHz, the simulation should be able to account for this. By clicking the box where it says “TLines – Ideal” and selecting “Simulation – S_Param” allows for the first step to simulating this system.



The display should look similar to the Palette on the left. Select “SP” from the Palette and drop the box onto the workspace. It should look similar to this when placed:

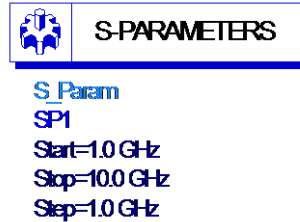


Figure 7. S-Parameter box for S-Parameter simulations

Figure 8. S_Param Palette

With the S-Parameter box now placed, there needs to be termination resistors as the system needs to know where to probe just like the HFSS simulations with the lumped port. In this case, it is important to understand that these termination resistors will act as your sources and your sinks for the circuit.

Select the box that says “Term” and looks like a resistor inside of a box. These will act as your input and output ports for the system. Place two terminating resistors on the schematic. To insert grounds, click the ground symbol that shares the same line as the “Simulation – S_Param” selection. Connect the grounds to each of the bottom of the terminating resistors.

Thus the schematic will look similar to this with some rearranging and changing parameters:

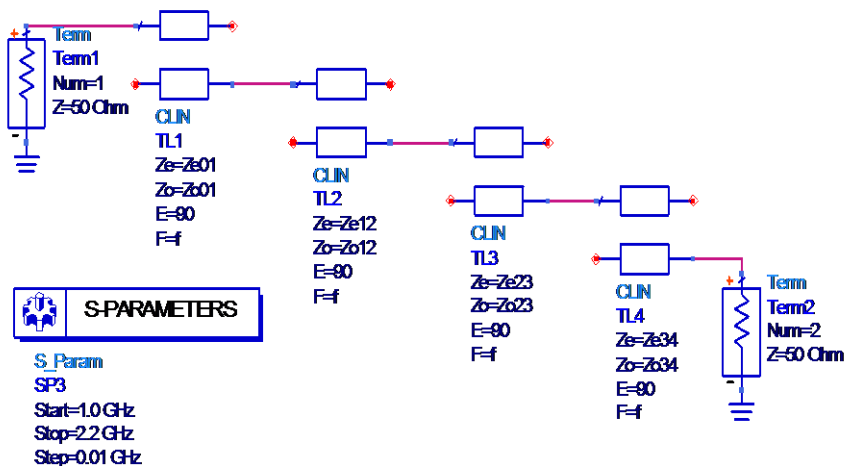


Figure 9. - Configured schematic that is ready for simulation

Be sure to change Start to 1.0 GHz, Stop = 2.2 GHz, and Step = 0.01 GHz. This will keep the plots from becoming too unwieldy.

5. Parameterization of the Simulation – OPTIONAL

The simulation will work wonderfully the way it is. However, if one wants to parameterize their system. This section presents a way of doing it.

Next to the ground button on the toolbar is a button with a box that says “VAR”, it looks like this:



Figure 10. Variable button

This generates a block that we can enter in variables. The best way to modify this block is by double clicking it.

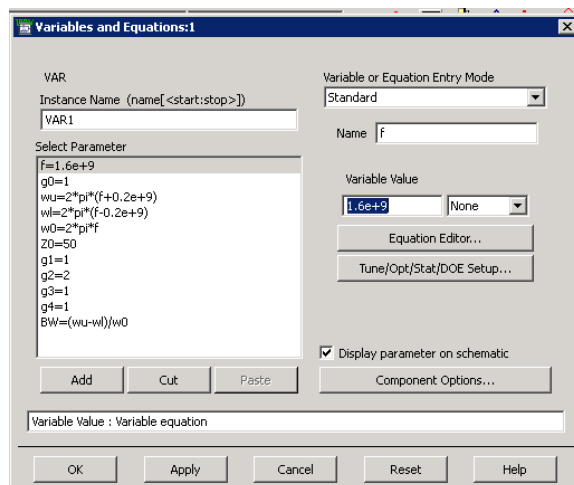


Figure 11. Variable edit menu

This interface is somewhat intuitive as you can type in your own variable into the Name field then press the “Add” button. This will allow you to edit the variable. To save a change of the variable, click another variable in the list as this will cause it to update and save the change or press “OK”. “Apply” will also work in this case.

As can be seen in the previous figure, this program also supports equations. With a little ingenuity, one can create a schematic that looks similar to this:

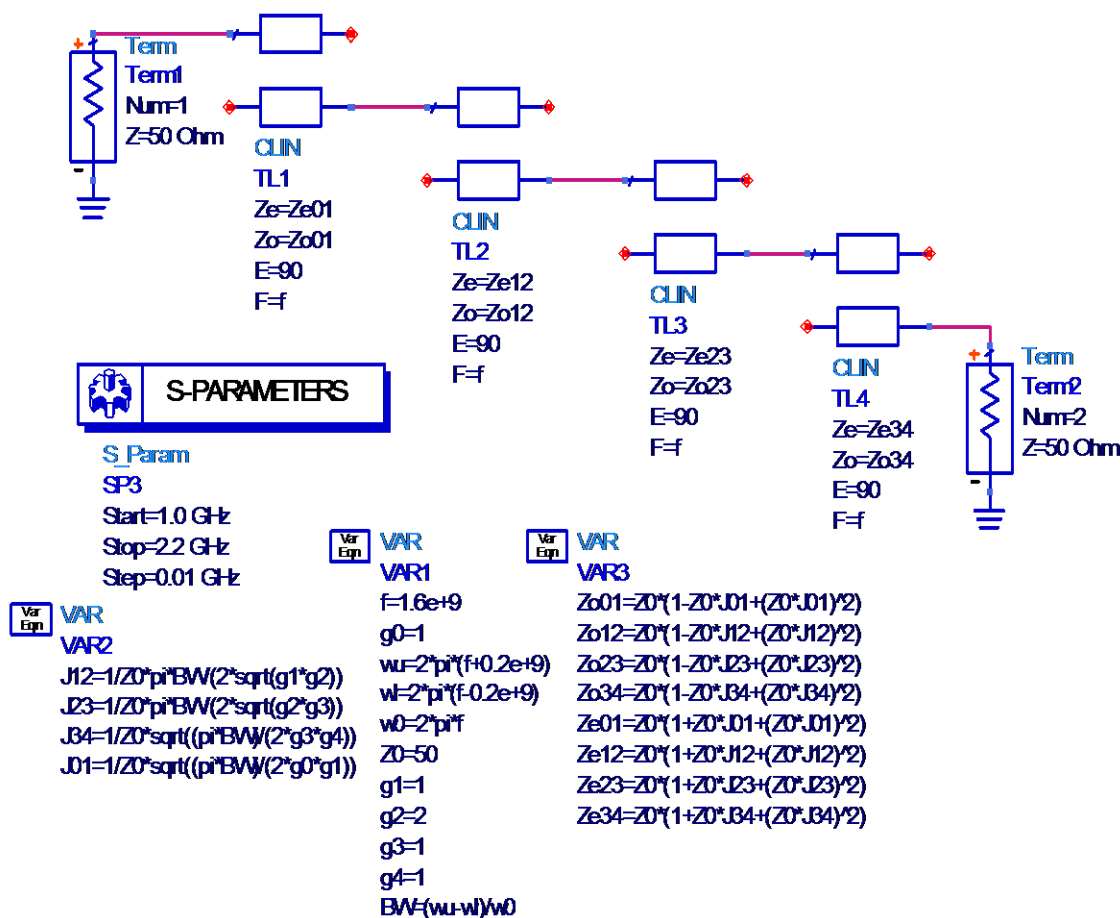


Figure 12. Fully Parameterized Circuit

6. Running the Simulation and Analyzing the Data

There are two ways to proceed from this point. Either press the shortcut key of F7 to run your simulation or select Simulate (from the toolbar) and press Simulate. This will generate a warning but that is only because there are some nodes that are left open. However, ADS solves this problem and still produces the right answer.

Once the simulation is completed, a new window will pop up. This window allows for great plotting ease. From the Palette provided, select the rectangular plot button.



Figure 13. Rectangular Plot

Place the plot in the white workspace. This will bring up a menu. Select S(2,1) and click “>>Add>>”. Select the radio button for dB. Press OK and OK again. Your workspace should now include the following plot:

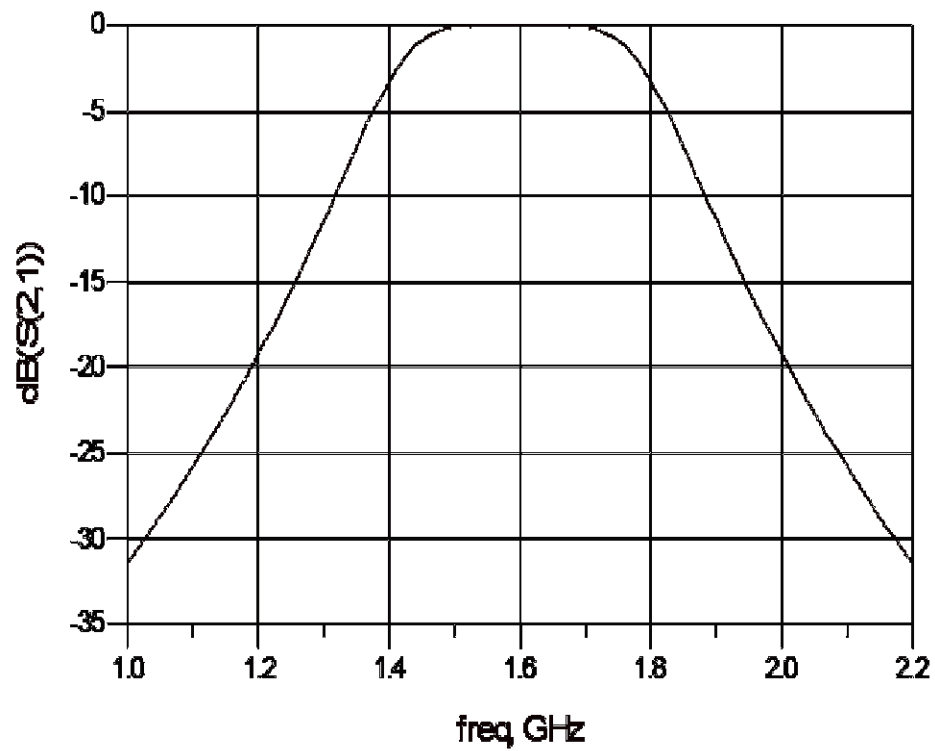


Figure 14. Filter plot for our circuit

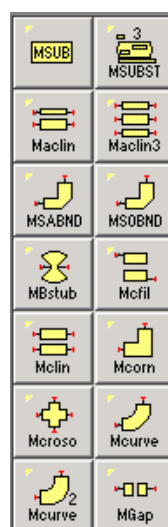
It works! Genius!

Part III Coupled Transmission Lines

1. Dielectric Differences – Alumina

With Part II, we focused on an ideal transmission line that allowed us to rapidly vary and test how our schematic would operate under basic principles. In this section, we will focus on expanding that from the ideal to the slightly more real. This will be achieved using the microstrip technology presented in ADS.

We begin with selecting “TLines-Microstrip” from the dropdown menu where “TLines-Ideal”, “Lumped Components”, and “Simulation-S_Param” all appear.



As can be seen on the left, there are various elements that can be placed for microstrip simulations. These include:

- Name – Description – Parameters
- MSub – Characterization of the dielectric substrate
- Maclin – Asymmetric coupled microstrip lines (W1, W2, S, L)
- Maclin3 – Asymmetric coupled microstrip lines (W1, W2, W3, S1, S2, L)
- Mcfil – A coupled line filter element (W, S, L)
- Mcln – Two equally sized coupled lines (W, S, L)
- Mlin – Microstrip transmission line (W, L)

All the W parameters deal with the actual width of each trace line. S deals with the spacing between strips. L is the length of each trace.

Figure 15. Toolbar for TLine-Microstrip



Figure 16. MSub Parameters

By placing the MSub block, we can alter a lot of various settings for the system. For this simulation, alumina will be used instead of FR4. Alumina’s chemical composition is Al_2O_3 . It has an exceptionally high thermal conductivity ($30 \frac{W}{m \cdot K}$). Alumina when other trace elements are present can be classified as sapphire or even ruby depending on the concentration of the trace elements. Alumina’s dielectric constant for microstrip structures is 9.6. TanD is approximately equal to 0.0002 in this instance.

Please set your variables to be equal to these. Notice that T=1.4 mils, that is effectively a 1 oz. copper trace. Also, the board will be 62 mils thick as we have previously done.

2. Determining additional microstrip parameters

ADS includes an exceptionally powerful tool that will be utilized as a part of this lecture. This tool is the LineCalc application. It allows for precise calculations of transmission lines to achieve optimal results without consulting a chart or graph to create these simulations. Start LineCalc by going up to the toolbar and selecting:

Tools → LineCalc → Start LineCalc

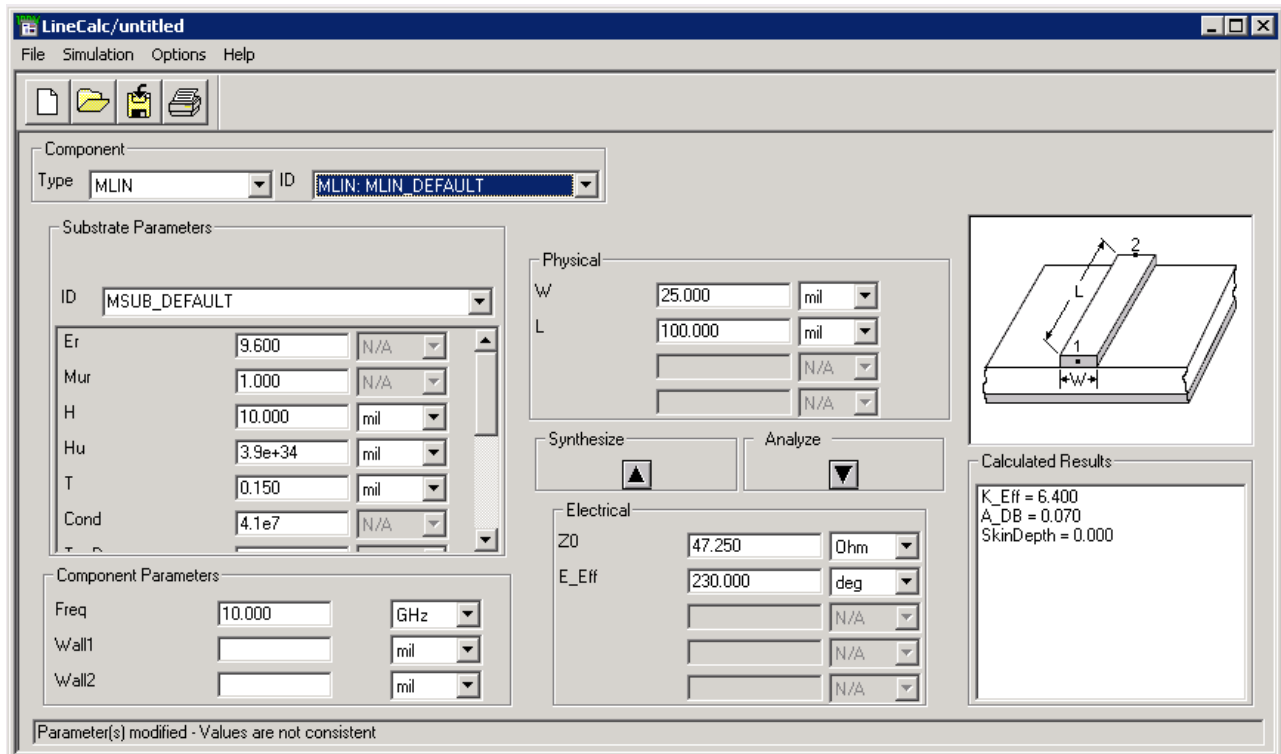


Figure 17. LineCalc start up menu

Change the type from MLIN to MCLIN. This will have ADS do some minor preparation calculations but that should not alarm you. By changing the parameters on the left to match the MSub that we have defined previously will establish a baseline for the microstrip line calculations. Be sure to change Freq (the frequency) to 1.6 GHz as each of these lines has to be calculated for a value of 1.6 GHz.

	Odd Impedances	Even Impedances
Z01	38.30210065	100.9678075
Z12	39.97130504	67.7393234
Z23	39.97130504	67.7393234
Z34	38.30210065	100.9678075

Table 4. Reminder of odd and even mode impedances

If you make a mistake with the odd and even impedances, the solution **WILL NOT CONVERGE**. Also, make sure that E_Eff is set to 90 degrees as that is the important parameter for the coupled microstrip lines. You do not have to set Z_0 since Z_0 is set by Z_e and Z_o . Once the

parameters are entered, select “Synthesize” to yield the values that need to be entered for two of the pairs of mutually coupled microstrip lines.

	Microstrip Pair 1 & 4	Microstrip Pair 2 & 3
W	0.588801 mm	1.217930
S	0.303590 mm	0.776120
L	19.817400 mm	18.900100

Table 5. The values that should be extracted from LineCalc

Close the LineCalc tool.

3. Continuing the set up of the schematic

Now that we are armed with the microstrip sizes that are necessary to make this project work, we can place our microstrip Mfil segments. Place two segments and set their parameters to match Microstrip Pair 1 and Microstrip Pair 2’s parameters, one each. Select the Mfil piece that has the same parameters as Microstrip Pair 2, copy and paste it. This is achieved either by pressing CTRL+C and then CTRL+V or right-click the component, select copy, then follow a similar process for paste.

Also place the SP block and termination resistors connected to ground appropriately. If done correctly, your schematic should look like this:

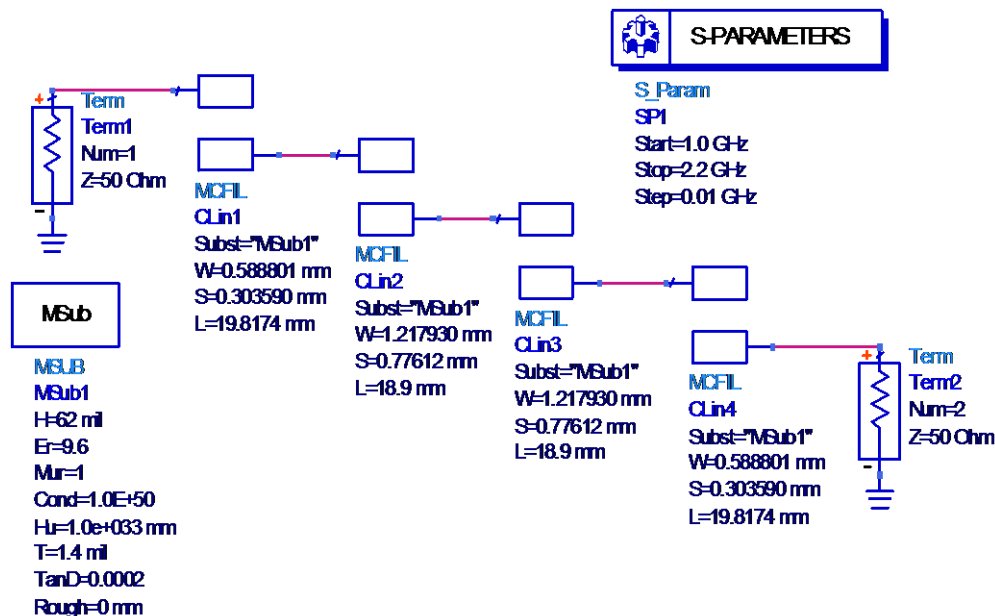


Figure 18. Microstrip Bandpass Filter Setup

For a refresher on how to add the SP block, see Part II. When this is simulated, it will yield a plot that looks like the following:

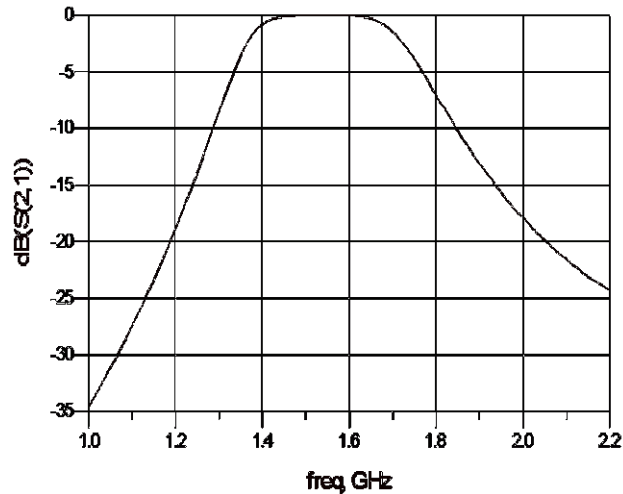


Figure 19. Plot of the Mfil variant's S-Parameters

With the ideal case, our results were perfectly symmetrical about the point of interest. However, for that case, we also use the ideal version of mutually coupled lines. Thus, just as we did with the Mfil, we can do the same with MClín in order to compare to our initial results for a filter. Thus, we will have a circuit that looks like this:

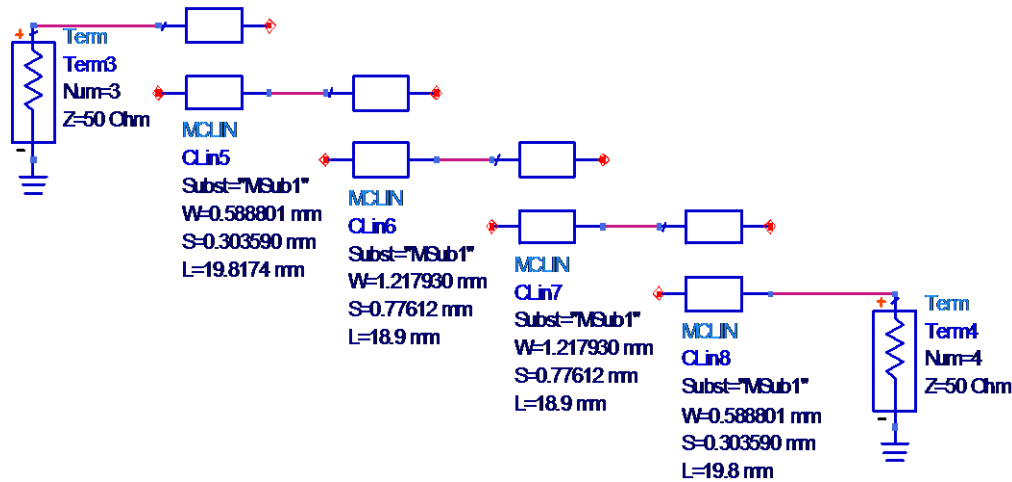


Figure 20. Mfil schematic modified for MClín components instead

While this should not make a major difference, it is a direct evolution from the ideal Clín configuration. Since this circuit is not touching the other circuit in any capacity, both can be run by ADS simultaneously to provide overlapping plots.

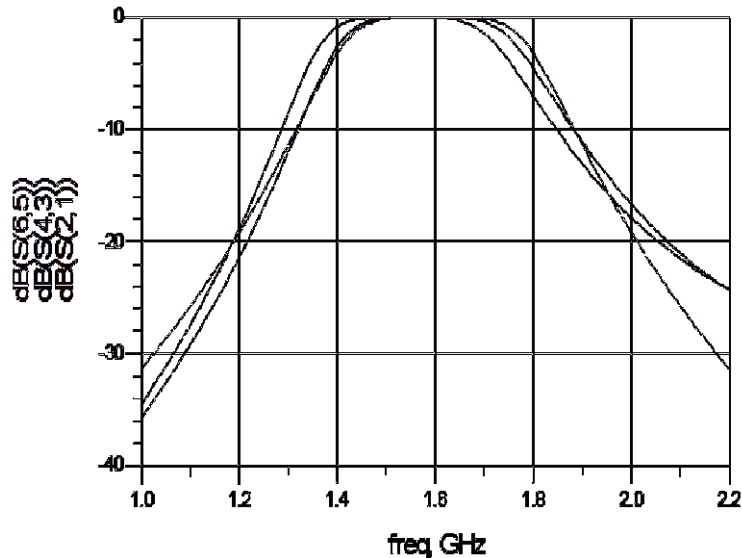


Figure 21. MFil (Red), MClin (Blue), Ideal (Pink) Filter Plots

Finally, we arrive at the genius result. For a filter, using the standard MClin component provides a result that is closer to the ideal while MFil moved the center of the passband. This is an interesting result as there was extra care taken when inputting the variables into LineCalc.

4. Momentum and RF Momentum

The fun begins! Since we have successfully implemented our models in the schematic capture tool provided by ADS, we may now move to the Method of Moments (MoM) and Finite-Element Modeling (FEM). Both are supported by ADS in the form of Momentum (MoM) and EMDS (FEM). With the schematic that was built in Part III, Section 3, we can now export this directly to the layout tool provided by ADS.

Copy and paste the MLin schematic into a new schematic. EDMS in ADS is exceptionally picky, thus you should only run one of these schematics at a time. Momentum is not nearly as picky about the situation and appears to handle the circuit exceptionally well. It is also faster than the FEM variant.

From the toolbar:

Layout → Generate / Update Layout

Click OK

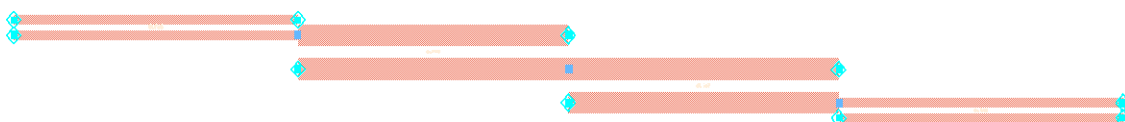


Figure 22. Filter generated by the layout tool in ADS

Immediately, from looking at the layout that was generated one can see that there is a slight problem with the layout. The two end pieces are effectively touching the same trace. This will

throw off the calculations of the FEM and MoM calculators. Thus, move the traces until the fat trace lines up with the inner edge of the thin piece of transmission line.

The last important piece before simulating is that we need to define the ports in this example since the termination resistors are no longer present. To define a port, click the button next to the ground symbol.

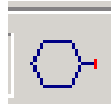


Figure 23. Port symbol used by the layout tool in ADS

Momentum versus RF Momentum – What's the difference?

Simply put, RF momentum ignores radiation.

*****DO NOT MISS THIS STEP*****

When loading a schematic into a layout, make sure to update the substrate that your solution is solved on. ADS does NOT automatically update this. To do this starting with the toolbar:

Momentum → Substrate → Update from Schematic

Also, it is useful to precompute the substrate. This is done by:

Momentum → Substrate → Precompute

For this comparison, normal Momentum will be used. So, let's get started with the toolbar.

Momentum → Simulation → S Parameters...

A new menu will pop up that will allow for a frequency sweep plan, thus instead of doing an adaptive sweep first set it for linear and place it at the same spacing as used before. That is to say, Start = 1 GHz, Stop = 2.2 GHz, and 0.01 GHz as the spacing. Click the Add button. Press the Simulate button and wait. When it is done, you should arrive at a similar solution to this one:

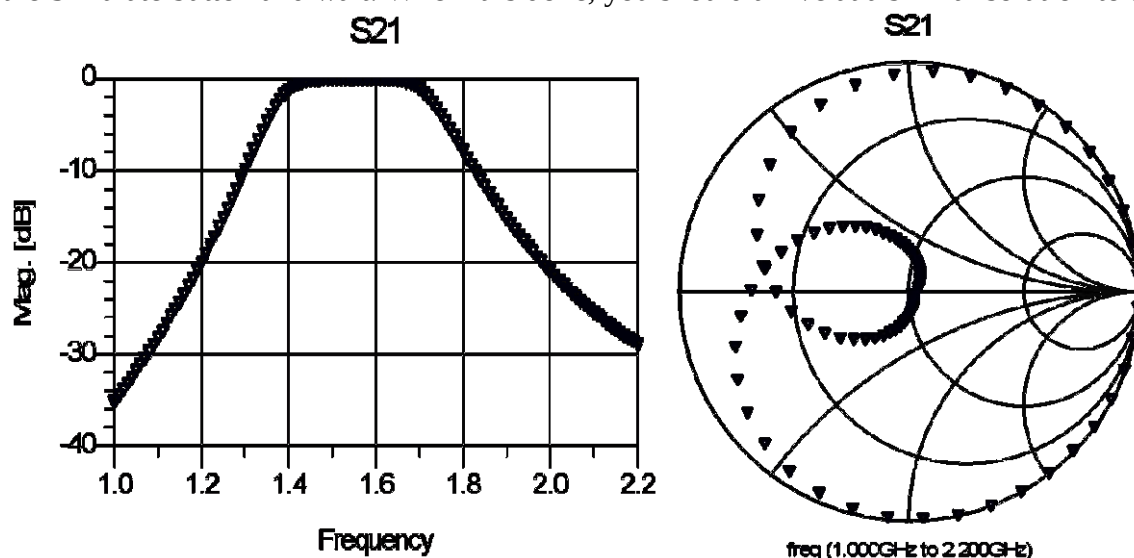


Figure 24. S-Parameter plot of S21

Genius! Look at what Momentum has produced. It almost completely traces what the MClint lines in the simulation provided. This is a wonderful result as it appears to agree full-heartedly with the models used by ADS's normal circuit solver.

5. EMDS

To simulate using the built-in FEM solver is a very similar technique when compared to Momentum.

*******DO NOT MISS THIS STEP*******

**When loading a schematic into a layout, make sure to update the substrate that your solution is solved on. ADS does NOT automatically update this. To do this starting with the toolbar:
EMDS → Substrate → Update from Schematic**

The FEM solver in ADS is much slower than Momentum. It is not recommended to repeat the same refined step process as performed in Momentum since the MoM calculation was solved within 5 minutes of being started. EMDS can take a better part of a day if values are chosen poorly.

To start EMDS, look at the toolbar and then:

EMDS → Simulation → S-Parameters

For this particular run, to produce these results, leave the step blank while setting the Start = 1 GHz and Stop = 2.2 GHz. Also, switch from the linear mode to adaptive. This will allow EMDS to get the most out of the areas of interest. Click the Add button to add this to the frequency plan. Then press the button Simulate button and you are good to go.

Unfortunately, at the time of writing this, the EMDS results are not reporting anything that seems to be useful.

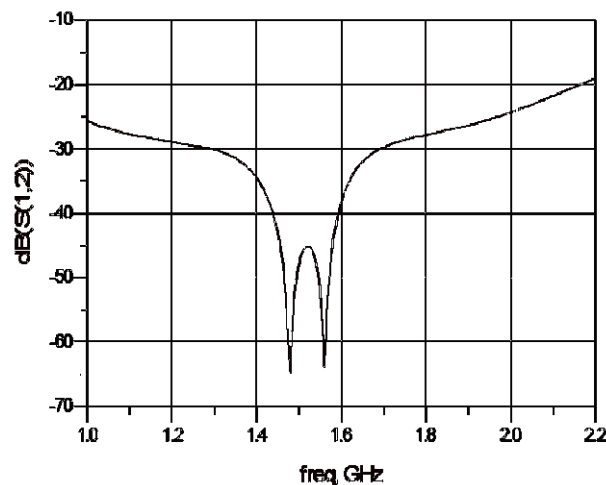


Figure 25. Plot of S-Parameters of Port 1 versus Port 2

HW#9 Exercises

1. Recreate this circuit using HFSS. For ease of calculation, use infinitely thin conductors (Perfect E rectangles). This changes the width and height of each conductor as follows:

- Microstrip Pair 1 & 4
 - $W = 0.65 \text{ mm}$
 - $S = 0.23 \text{ mm}$
 - $L = 19.2 \text{ mm}$
- Microstrip Pair 2 & 3
 - $W = 1.26 \text{ mm}$
 - $S = 0.72 \text{ mm}$
 - $L = 18.7 \text{ mm}$

Good luck!