In this tutorial you will learn how to use ADS microstrip components to simulate a bandpass filter using coupled lines. You will do a network simulation and an electromagnetic simulation of the filter using **Momentum**.

1. Start ADS
2. Open an existing project: *File->Open Project …* or start a new project: *File->New Project->*
3. Start new design: *File->New Design*. Give the design an appropriate name like: *bandpass*
4. Pull down the **Component Palette List** and go to: *TLines-Microstrip*
5. Select *MSUB* and place it somewhere in the schematic window. Set the height of the substrate to H = 0.127 mm and the relative dielectric constant to Er = 2.2. Leave the other parameters as they are for now.
6. To calculate the width of a microstrip line, ADS has a special calculator that finds the line-widths for a particular impedance or vice versa. Go to *Tools ->LineCalc->Start Linecalc*
7. The default component type is MLIN, but you can also find coplanar waveguide (CPW) and other transmission lines. Stay with MLIN. In the **Substrate Parameters** box enter Er = 2.2 and H = 0.127 mm. Make sure you pick the right units in the drop-down box. In the **Electrical** box enter Zo = 50Ω. Leave E_Eff alone.
8. Click the *Synthesize* arrow. The calculator will show in the **Physical** box that the corresponding width is 0.386 mm. In the **Calculated Results** box you will see K_eff=1.875. This is the effective dielectric constant. You can now close LineCalc: *File->Exit* and click *No* in the dialog box.
9. In the **Schematic** window scroll down the components list on the left-hand side and pick **MLIN**. Place two of these parts in the window and set W = 0.386 mm and L = 15 mm.
10. Next select **Mcfil** from the components list and connect three of these together. For the first and third coupled line sections set W = 0.10 mm, S = 0.10 mm, and L = 11 mm. For the middle section set W = 0.15 mm, S = 0.10 mm, and L = 11 mm. Connect the two input and output 50 Ω lines to the coupled line sections as shown in Figure 1.
11. Now you will set up the s-parameter simulation. In the **Component Palette List** select *Simulation-S_Param*. Click on **SP** and place the box in the window. Set *Start* to 3.0 GHz, *Stop* to 7.0 GHz, and *Step* to 0.1 GHz. The next step is to put the signal sources in the network. Select the **Term** icon and place two sources at the input and output of the filter. Notice that the default source impedance is Z = 50 Ohm. Leave it at that.
12. Simulate the circuit by pressing the wheel icon or selecting *Simulate->Simulate*
13. A **Data Display Window** will appear automatically or if not, then initiate one to look at the results. Choose a rectangular display and place it in the window. Select **S(2,1)** and then *Add*. Click OK. The result should look like the bandpass response shown in Figure 2.
14. The filter response looks good from a network point of view. To further validate this design we need to perform an electromagnetic field simulation using the method-of-moments field solver **Momentum**, which is part of the ADS package. To do the field simulation you need to layout the filter structure just as you would build it on a printed circuit board. You can do this one of two ways: for simple designs you can use the automatic layout tool in ADS. For more complex designs you need to do the layout manually. Since this is a simple structure, we will use the automatic layout: *Layout->Generate/Update Layout*. Click *OK* in the dialog box. Click *OK* again. A layout window will pop up with the filter.

15. Before continuing, we need to set the correct units in the layout window. Click on *Options->Preferences*. Select the **Grid/Snap** tab and set the *Snap Grid Distance* to X = 1 and Y = 1. Set the *Snap* to *Pin, Vertex*, *Edge*, and *Grid.* Select the **Units/Scale** tab and set the *Length* field to mm. Select the **Layout Units** tab and set the units to mm.
16. Zoom in on the transition between the 50 Ω line and the first coupled line. Note that the 50 Ω line is shorting the coupled lines. To rectify this problem shift the line upwards. First determine how much you have to move it upwards by using the ruler: *Insert->Measure*. The result is 0.143 mm. Select the transmission line by clicking on the left mouse button and then *Edit->Move->Move Relative* type X = 0 and Y = 0.143. Do not type the units. The result should look like **Figure 3**. Now, go to the transmission line at the other end of the filter and move it as well.
17. To place the ports, click on the **Port** icon and place one at the input and another at the output of the filter.
18. Now we need to specify the substrate parameters: *Momentum->Substrate->Create/Modify.* Change the **Substrate Layer Name** to *Rogers5880*. Change the **Thickness** to 0.127 mm and the **Permittivity (Er)** to 2.2. Click *Apply*. Select the **Metallization Layers** tab. In the *Substrate Layers* box select the dotted line above *Rogers5880* and then select *Strip*. Click *Apply* and then *OK*.
19. Click *Momentum->Substrate->Precompute* and set the **Minimum Frequency** to 1 GHz and the **Maximum Frequency** to 30 GHz.
20. To set the Mesh discretization frequency: *Momentum->Mesh->Setup*. For the **Mesh Frequency** type 5 GHz. Click *OK*.
21. Next set the simulation frequency range: *Momentum->Simulation->S-Parameters*. Set the **Sweep Type** to Linear, the **Start** and **Stop** frequencies to 3 GHz and 7 GHz respectively, and the **Frequency Step** to 0.1 GHz. Click *Add to Frequency Plan List*, and finally click *Simulate.* Momentum will now start simulating your filter.
22. After the simulator is done, a window will pop up automatically with all the s-paramters. In **Figure 4** you will see S21. It is quite close to the network simulation in **Figure 2**, thus validating the design.

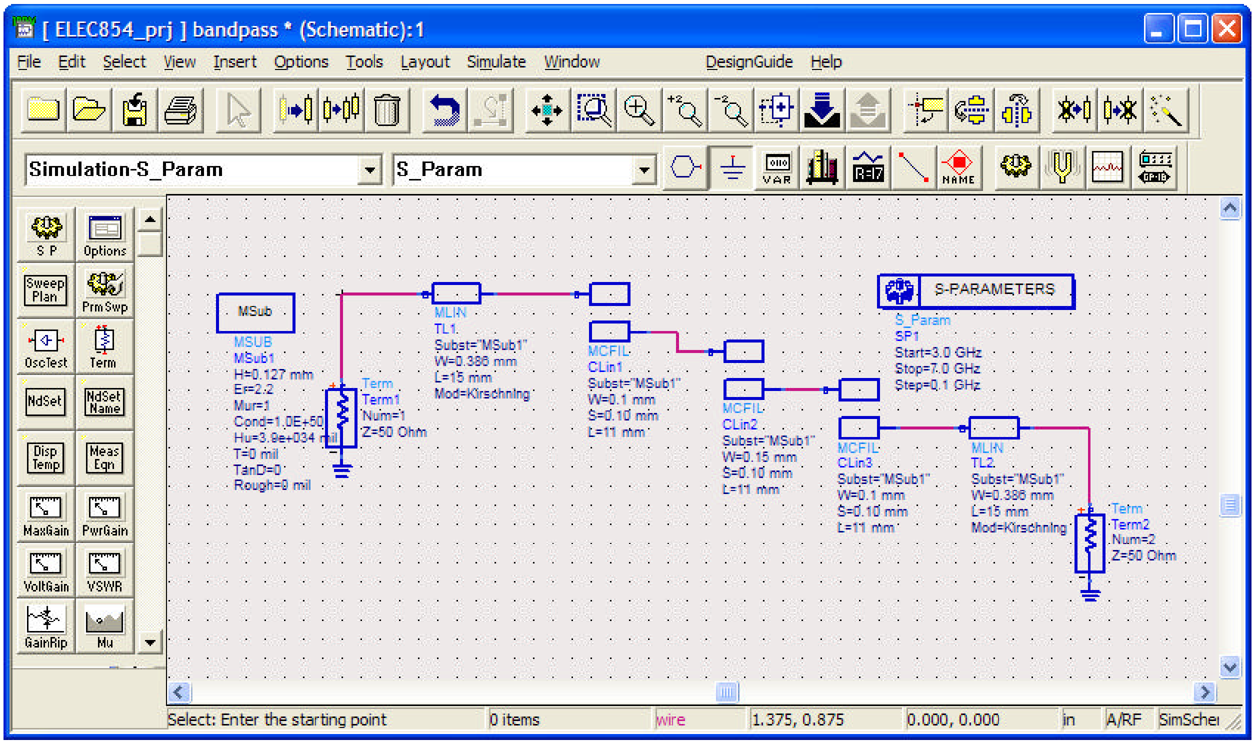

Figure 1 Bandpass Filter Schematic

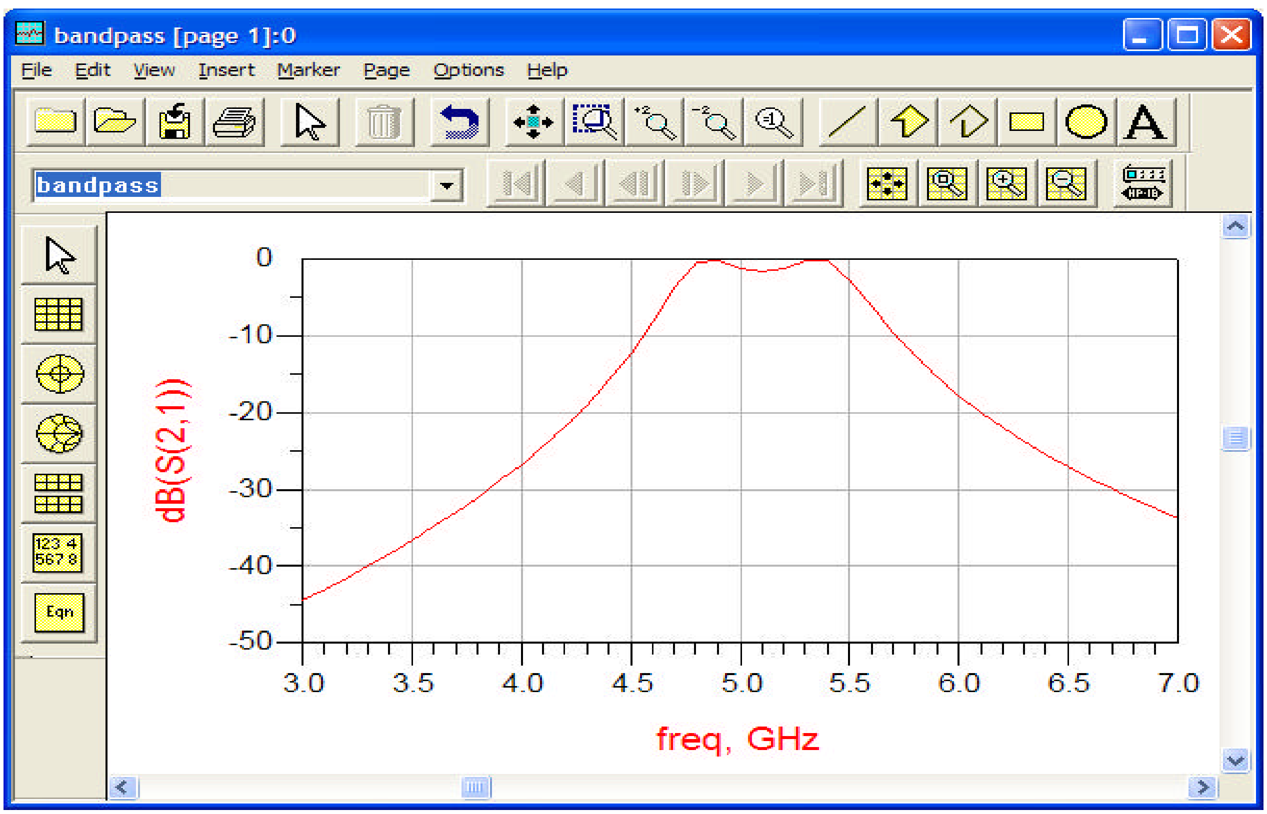

Figure 2 Plot of Bandpass Filter Response

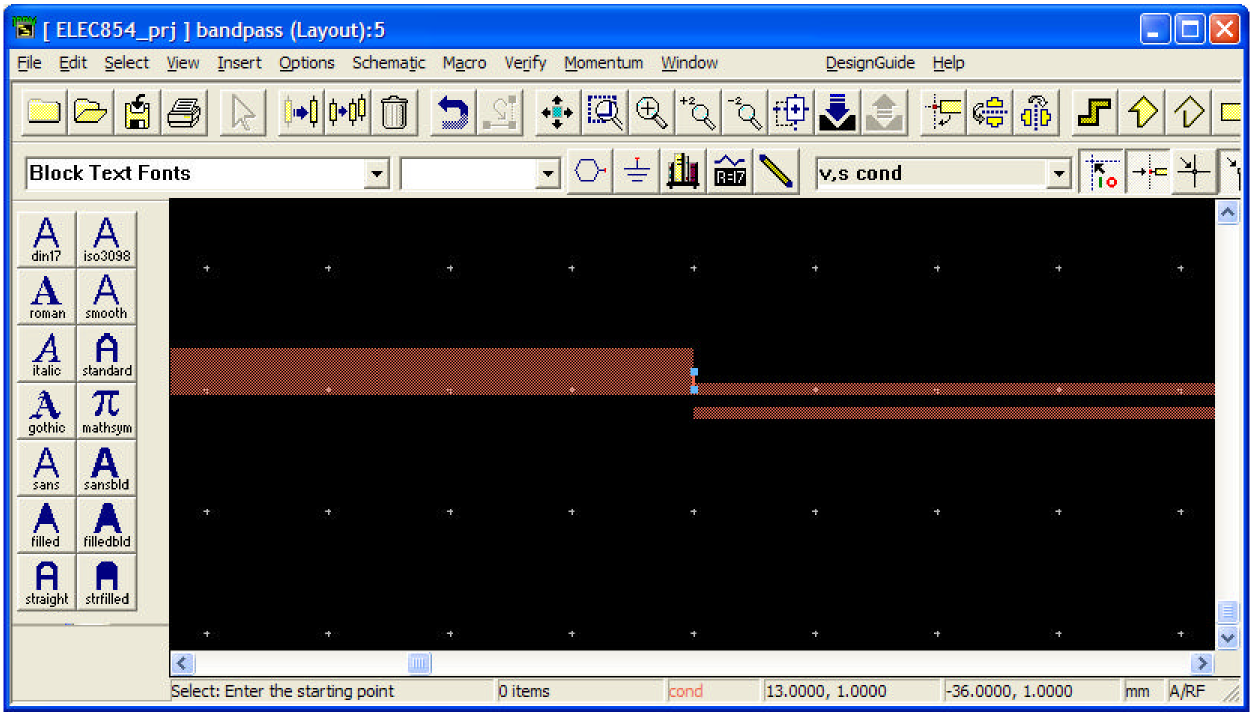

Figure 3 Bandpass Filter Layout

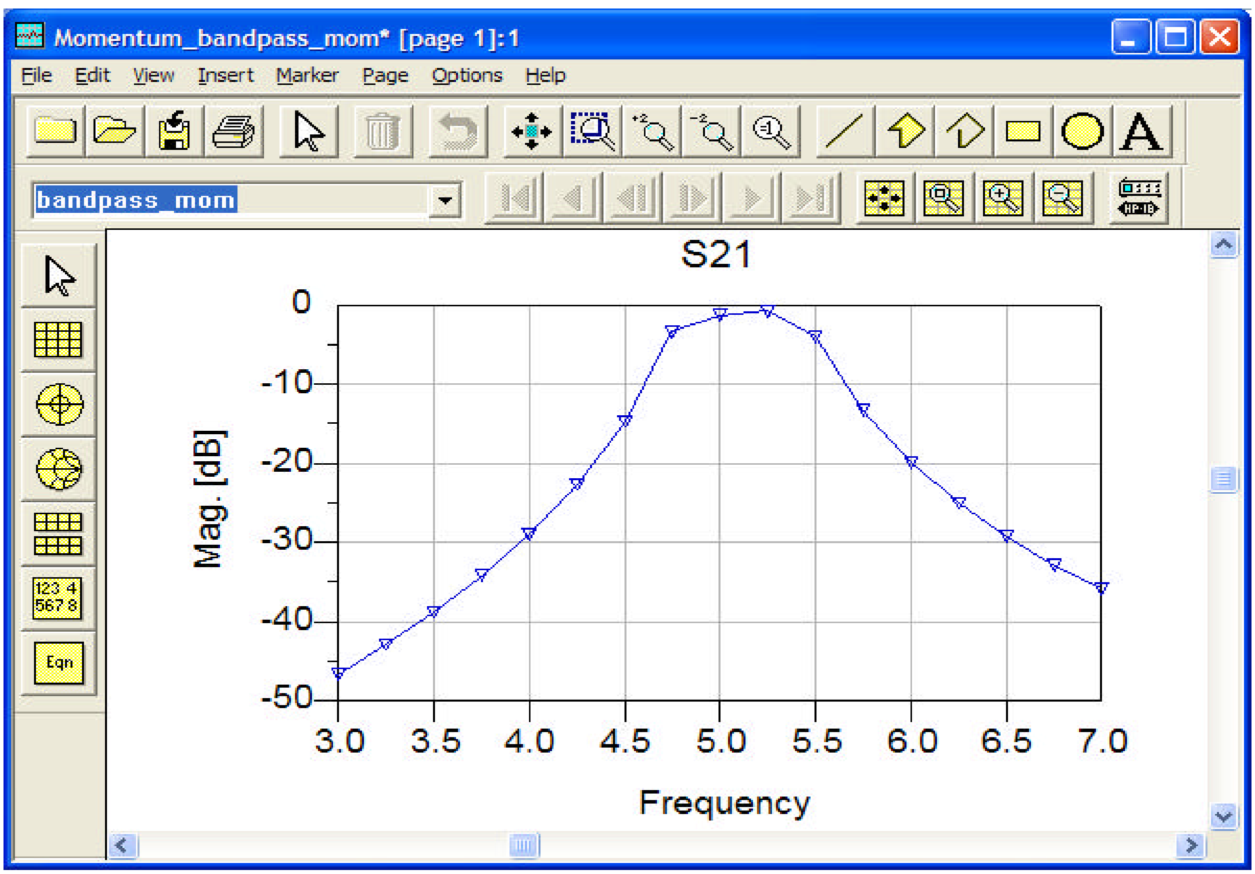

Figure 4 Bandpass Filter Momentum Simulation Results