

## S-Parameter Simulation: Momentum

In the second part of this tutorial, the Microstrip line will be analyzed using the full-wave simulator, Momentum, which is based on the Method-of-Moments (MoM) numerical method. Unlike the schematic simulator which used equations to obtain S-parameters and other electrical properties of the components under test, a full-wave simulation actually solves Maxwell's equation for the design. As a result, a full-wave simulation is more accurate and should always be preformed for devices that have bends or close gaps or when the user wishes to model the device as accurately as possible. In addition, the schematic simulator does not account for radiation, therefore radiating structures such as antennas designed using Agilent ADS can only be characterized by Momentum.

### Simulation Setup

To perform a full-wave simulation on the Microstrip line from Part I, open up the project and from the schematic window go to Layout>Generate/Update Layout. A series of windows will appear, click on Okay for all of them; once this has been done a Momentum window will appear showing the layout of the Microstrip line as depicted in Fig. 1.

### Microstrip Transmission Line Design Values

- Operational Frequency: 1 GHz to 5 GHz
- Substrate: 2.2 Permittivity and Height of 1.57 mm
- Characteristic Impedance: 50 Ohm
- Electrical length: 90 degrees at 2.4 GHz.

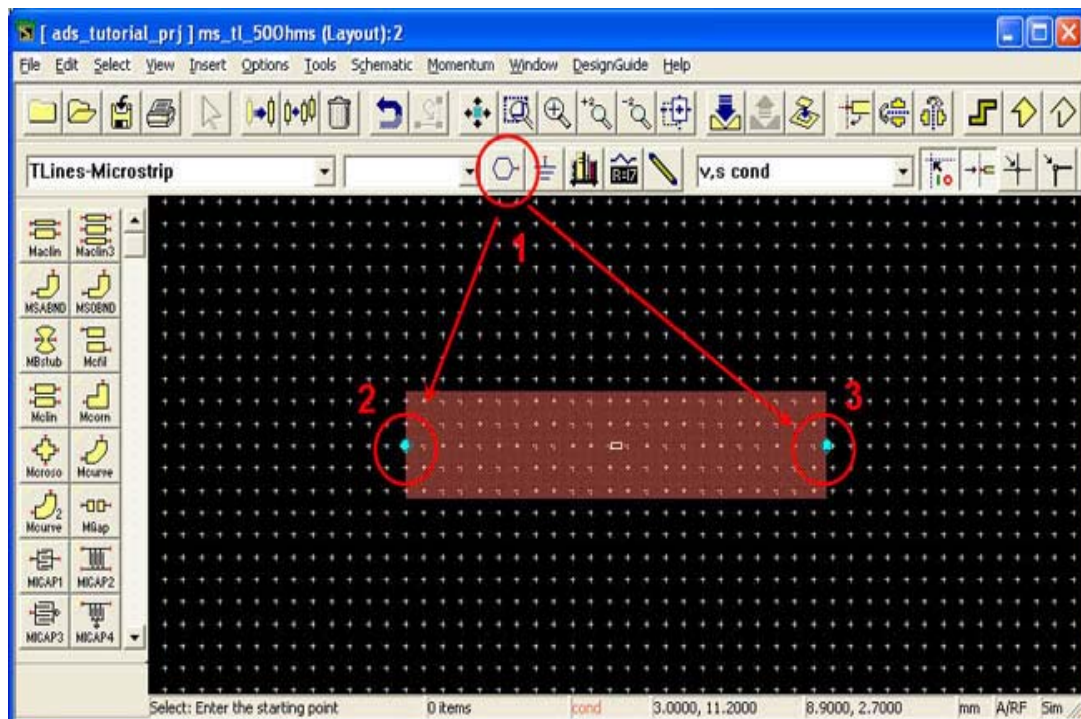


Figure 1. Momentum window showing Microstrip line.

Ports have to be added to the Microstrip line; to do this:

- 1.) Click on the port icon labeled 1 on in Fig. 1,
- 2.) Click on the first port labeled as 2 in Fig. 1
- 3.) Click on the second port labeled as 3 in Fig. 1.

The substrate has to be defined for the momentum simulation; to do this go to *Momentum>Substrate>Update From Schematic*. Next, go to *Momentum>Mesh>Setup* and for the mesh frequency enter the highest frequency of the simulation (5 GHz) and then go to *Momentum>Mesh>Precompute*; the mesh will be applied to the Microstrip line. The precompute function is useful because it allows you to view the mesh and decide whether the mesh is fine enough before running the simulation.

Before we can simulate, the simulation parameters have to be setup. To do this, go to *Momentum>Simulation>S-Parameters* and enter the values shown in Fig. 2; an adaptive sweep will be performed. It is important that you press *Update* after entering your frequency range and # of sample points. Press the *Simulate* button to run the simulation. After simulation is complete, a data window will appear and you can plot the desired data. Fig. 3 shows the full-wave results for S11 and S21 magnitude and the phase of S21. The results agree well with the schematic simulation. However, for more complex designs, the schematic and full-wave will show larger discrepancies.

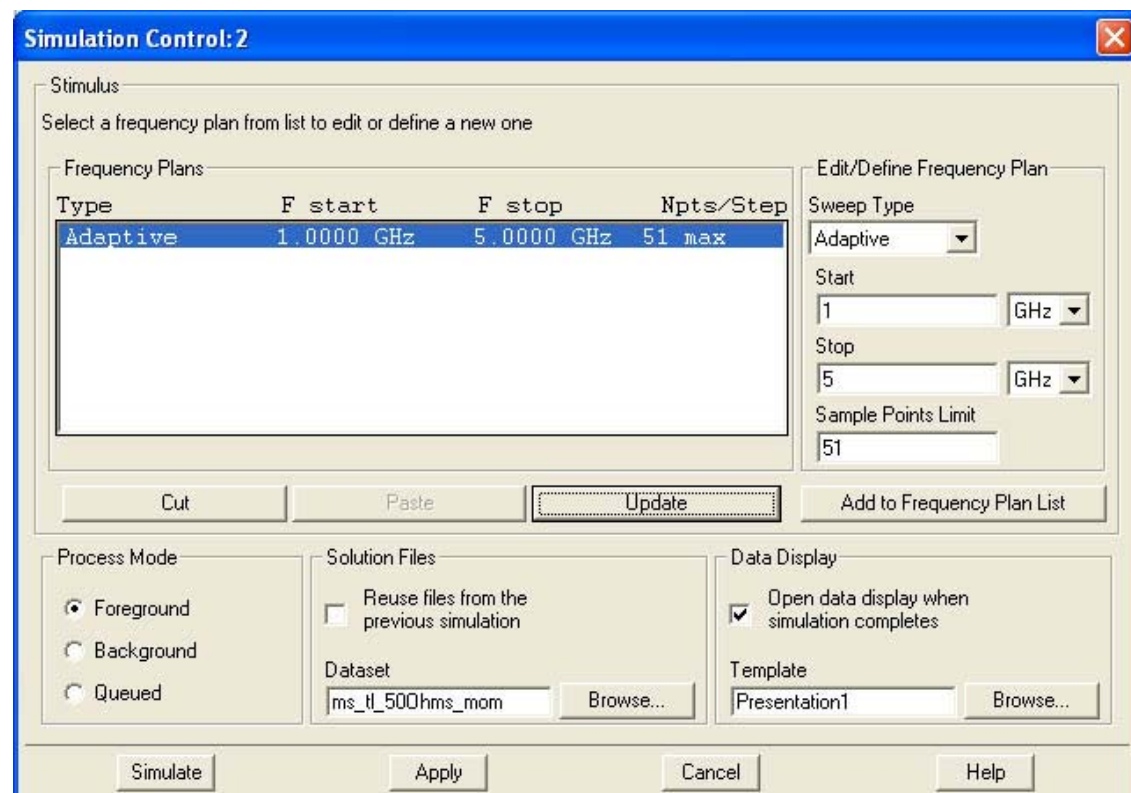


Figure 2. Simulation setup window.

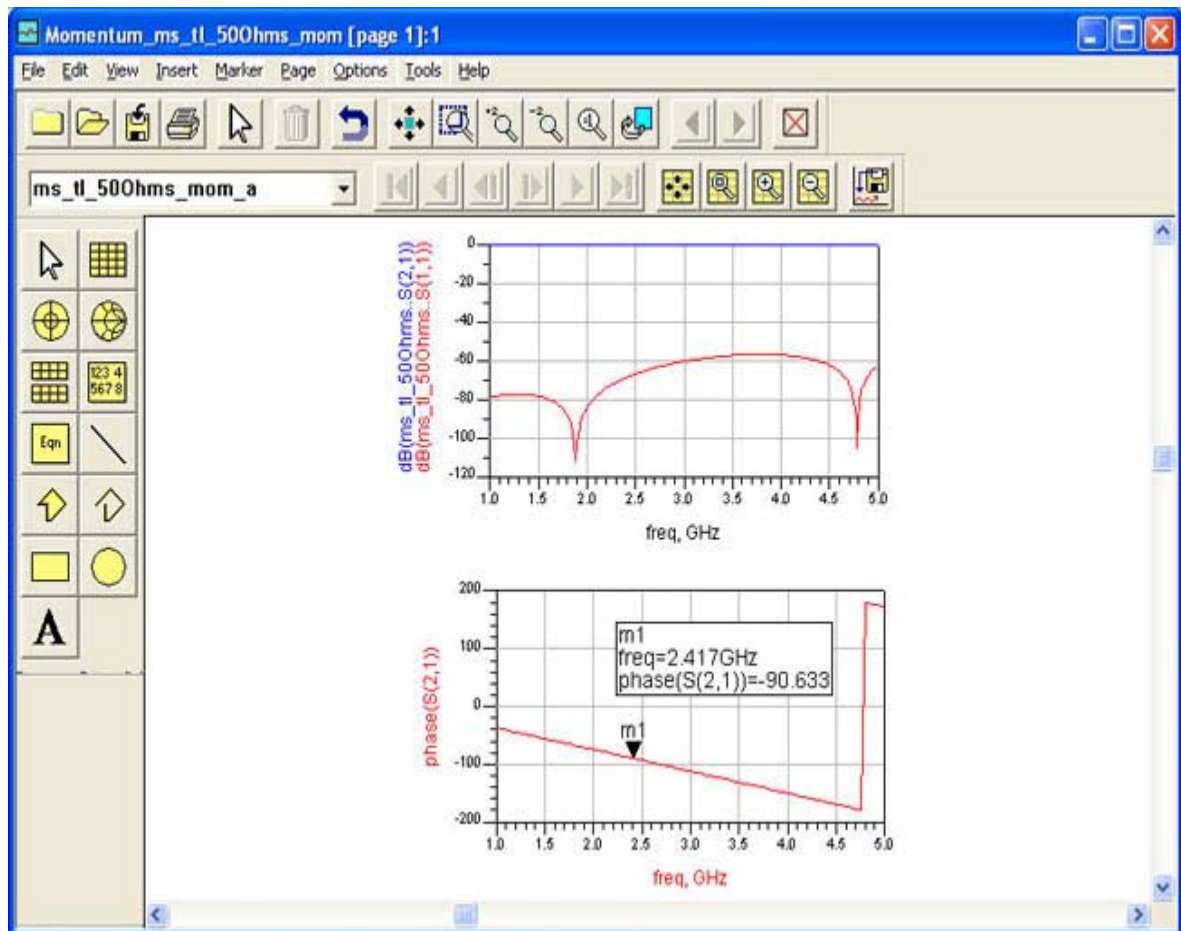


Figure 3. Plot of Momentum results.