304-573A Microwave Electronics

Tutorial

Microstrip Design Using HP Advanced Design System (ADS)

Kar-Lun Tam

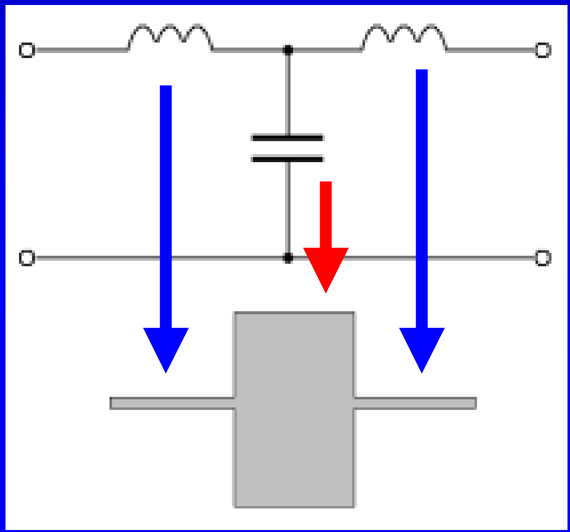
December, 1999.

What is covered in this tutorial?

- **In this tutorial, a simple low pass filter design using the HP Advanced Design System will be covered.**

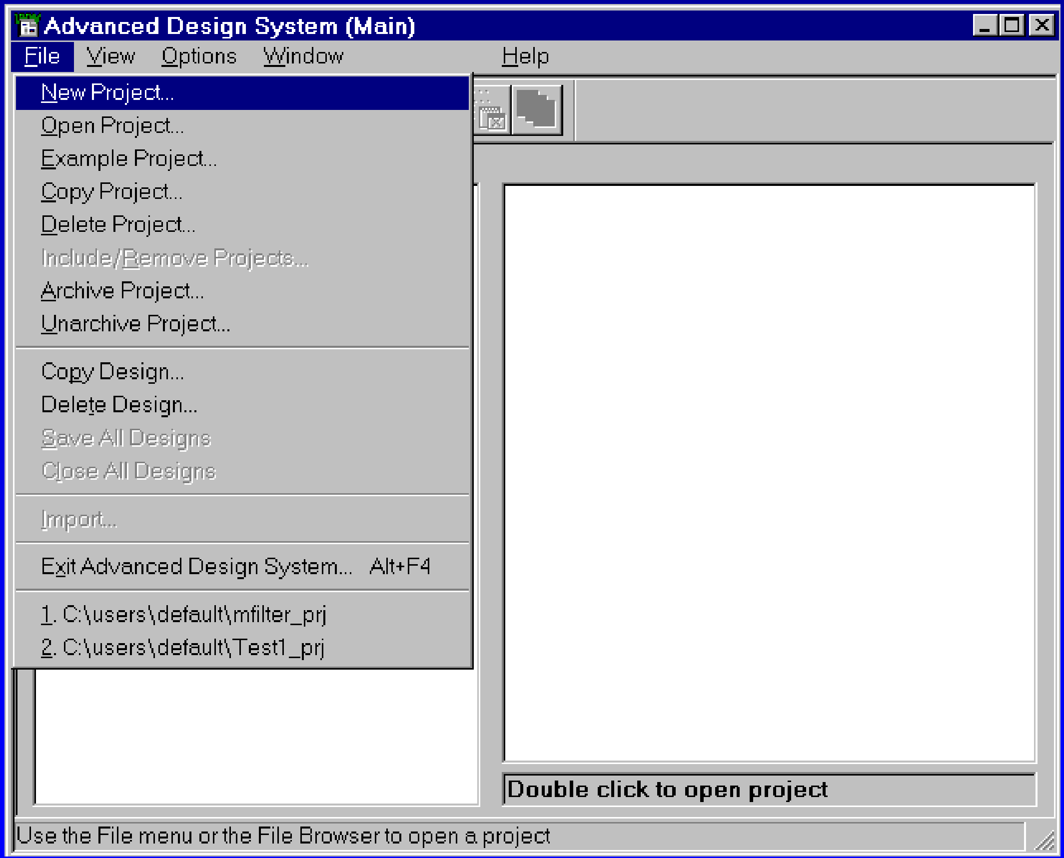
- **This includes the schematic simulations and layout simulations using HP Momentum.**

- **The low-pass filter design in this tutorial is a Butterworth prototype with 3 elements using microstrips (**ε_r=4, dielectric thickness = 0.5mm) **:**



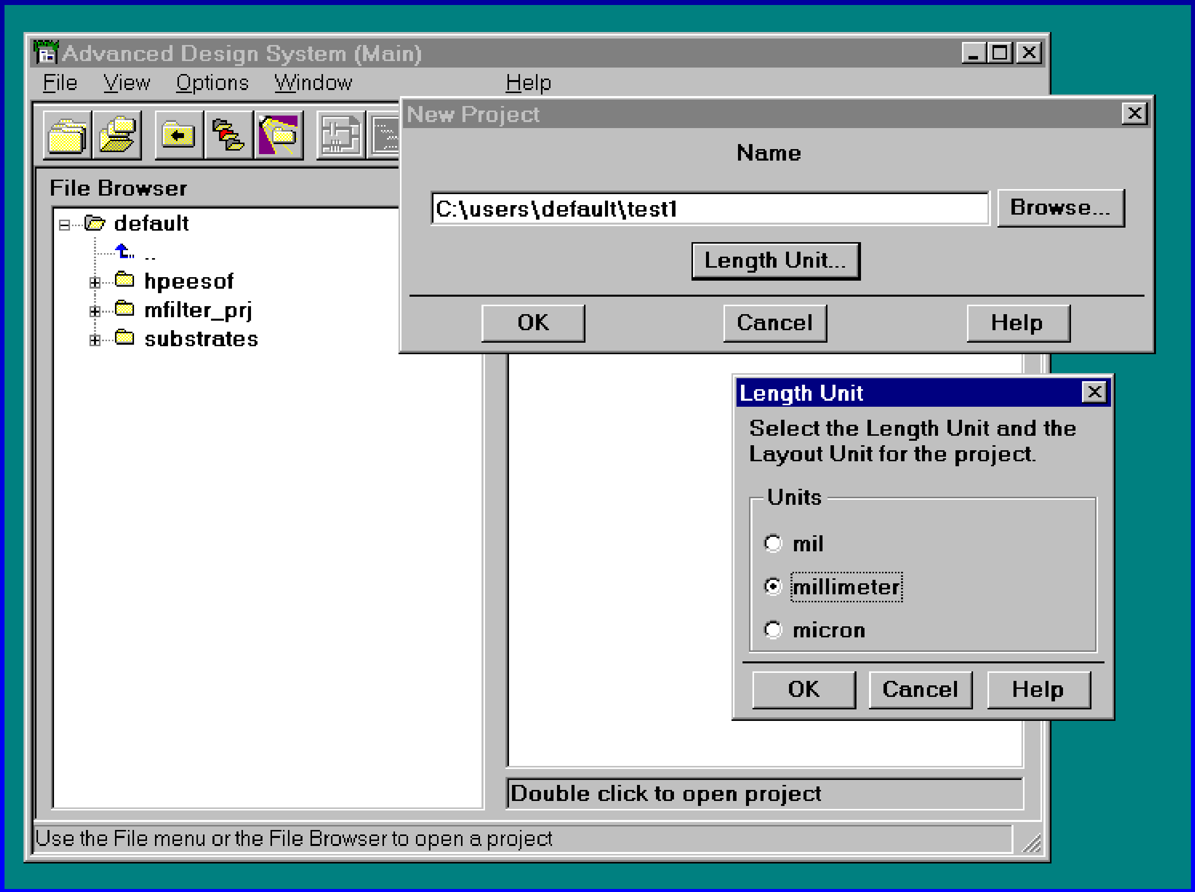
Starting a New Project

- **To start a new project, go the the File Menu of the HPADS window and select New Project.**



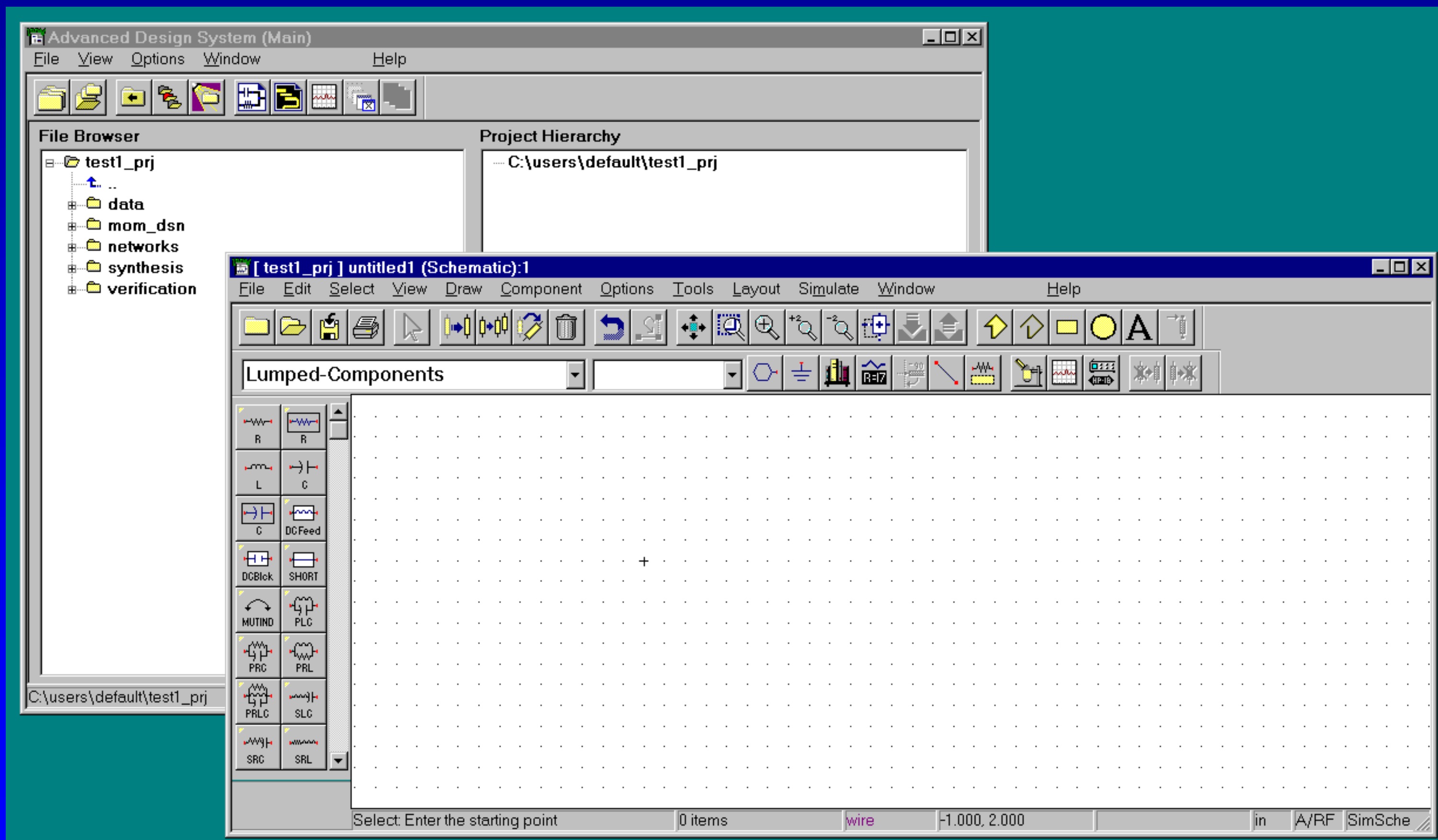
Starting a New Project (2)

- **Then, enter the name of the new project. Click Length Unit and set the length unit to milimeter. Press OK to close these two dialog boxes.**



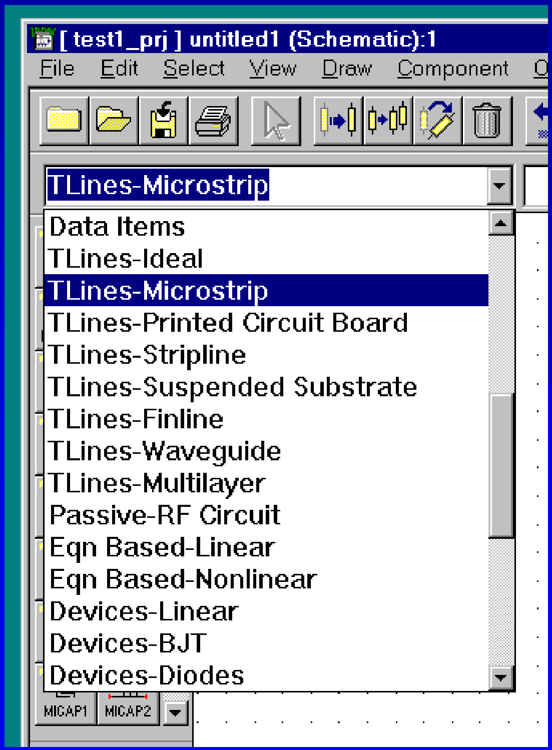
Schematic Editor

- **After a new project is created, a blank schematic window will pop up.**



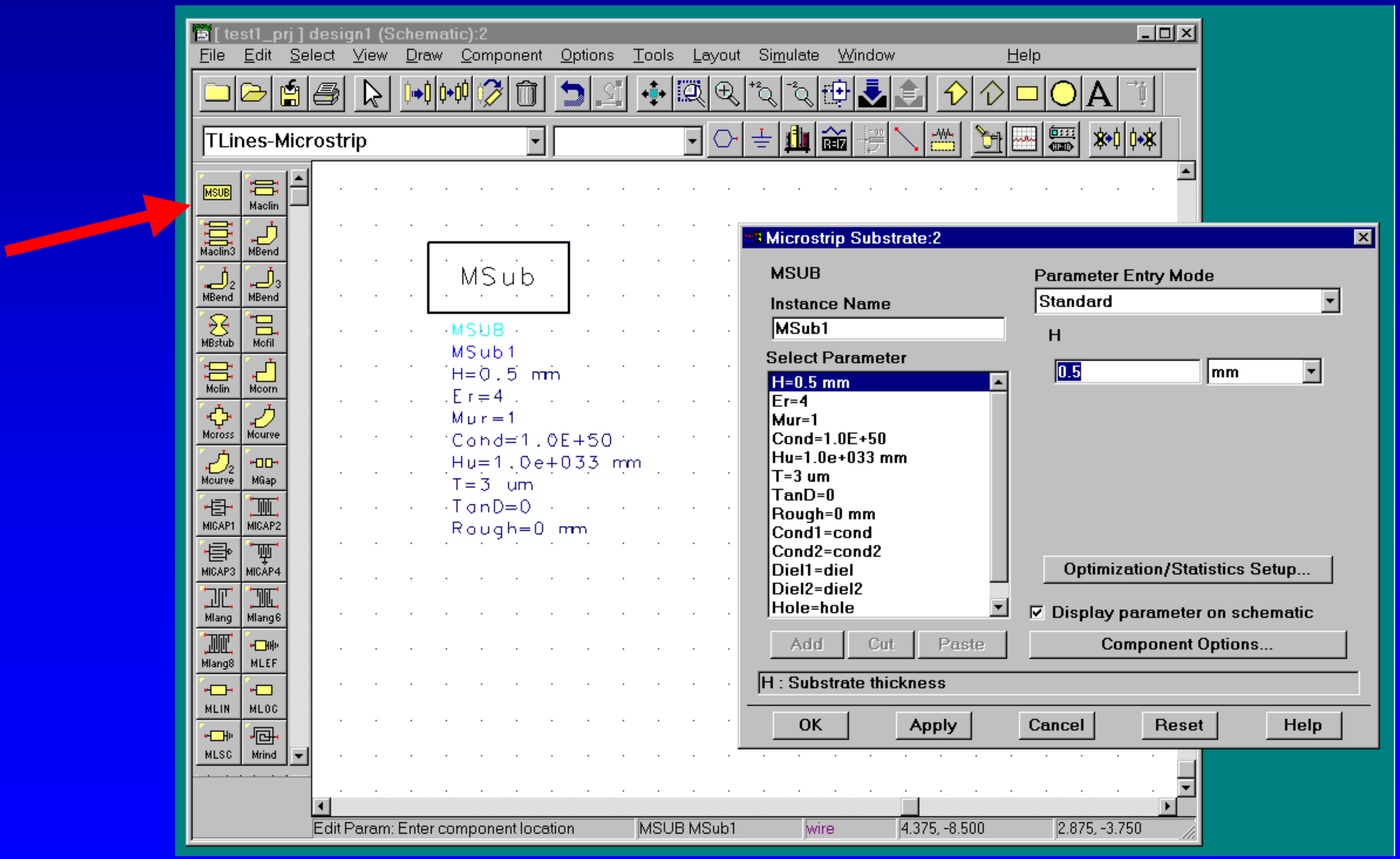
Schematic Editor Palette

- **On the left side of the schematic editor window, you can find a selection box in which you can choose your palette. Choose "Tlines-Microstrip" for microstrip components.**



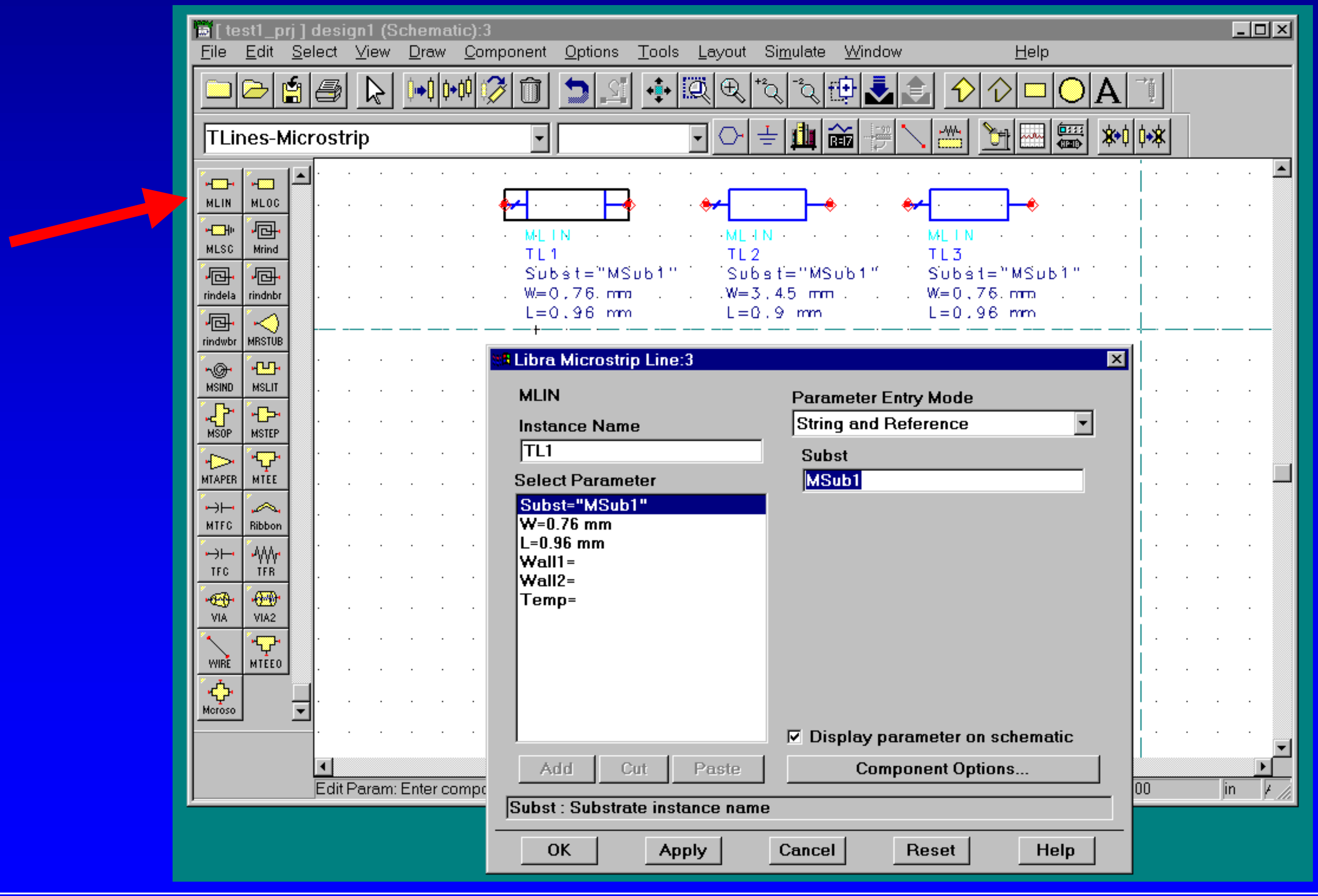
Adding the Substrate Model

- Choose the component MSUB from the top left hand corner of the palette to add the substrate model to this design. Double click the element in the schematic to change the values for H, Er, and T to those shown below.



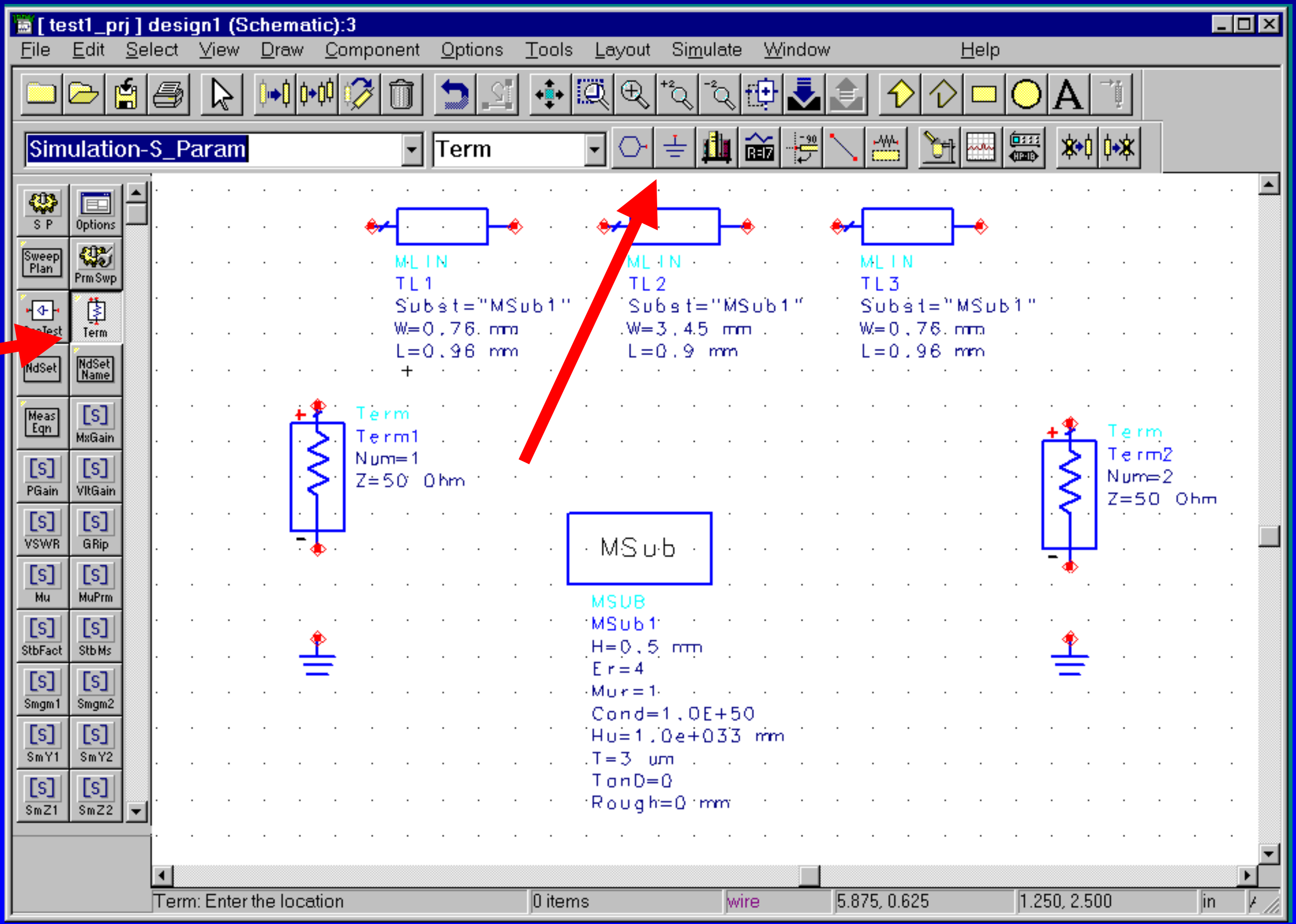
Adding Microstrip Lines

- Choose the component MLIN from the palette to add the three segments of the microstrip to this design. Change W(width) and L(Length) for each microstrip segment to those values shown below.



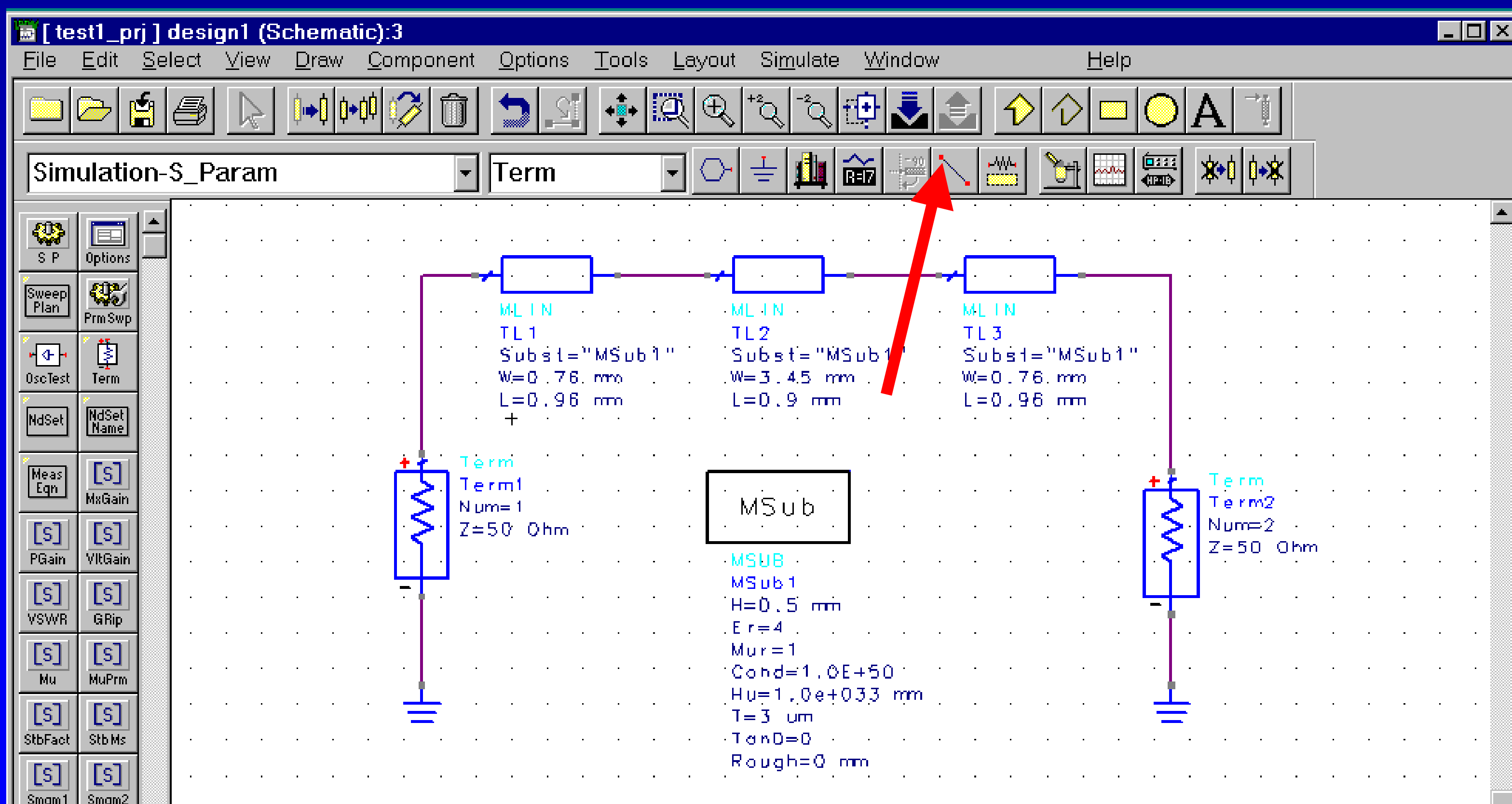
Adding Terminals and Grounds

- Choose the Simulation-S_Param palette and add two 50 ohm terminals to the schematic. Also, add two grounds to the schematic by choosing it from the toolbar.



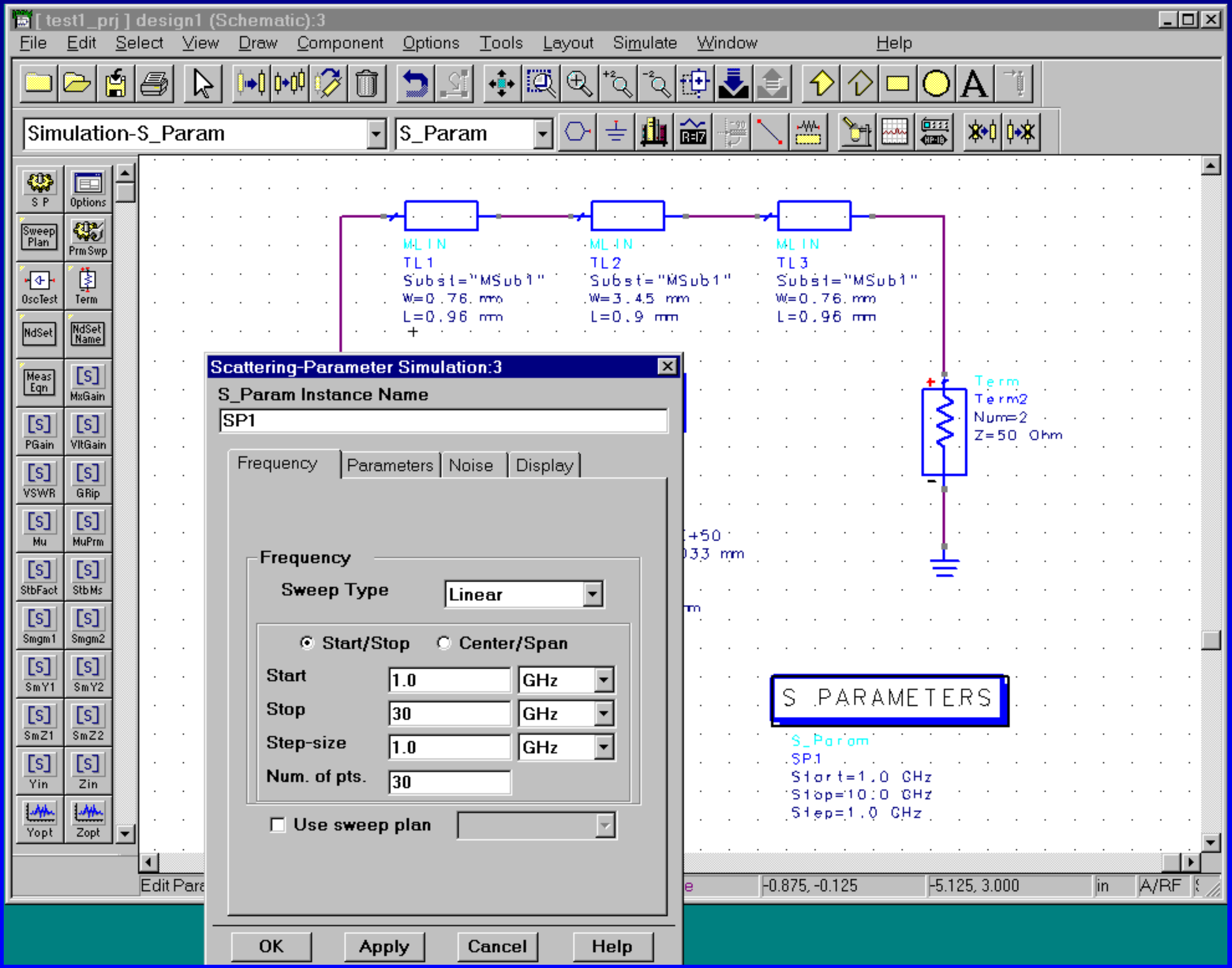
Connecting Them All Together

- Connect all the components together by choosing the connector wire from the toolbar.



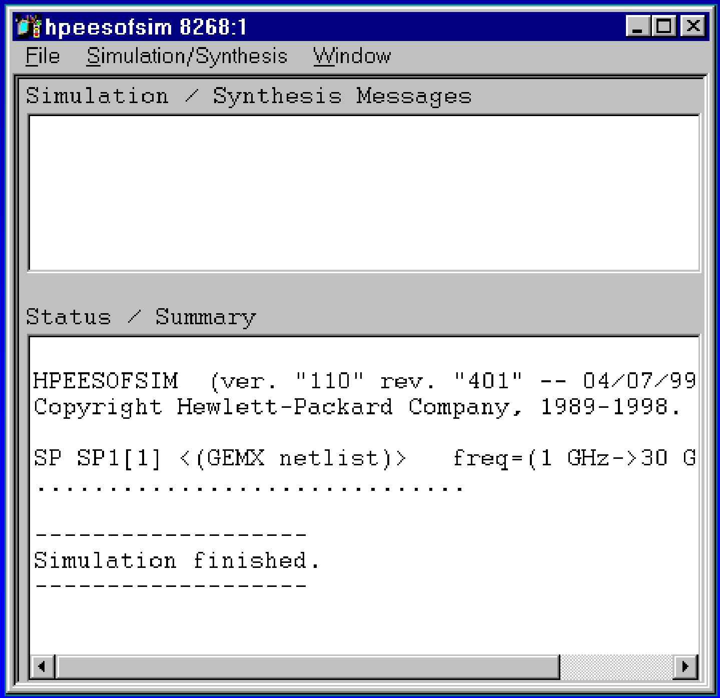
Define Simulation Type

- Choose the SP block from the Simulation-S_Param palette, and enter the S-paramter simulation parameters as follows.



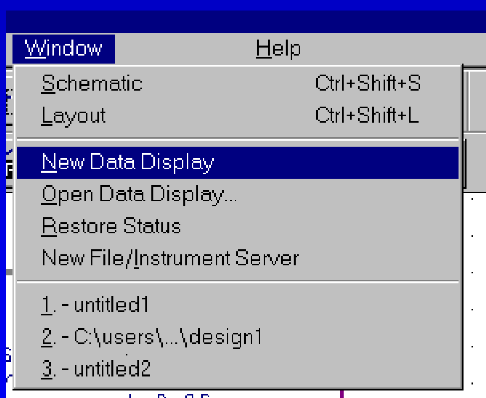
Running a Simulation

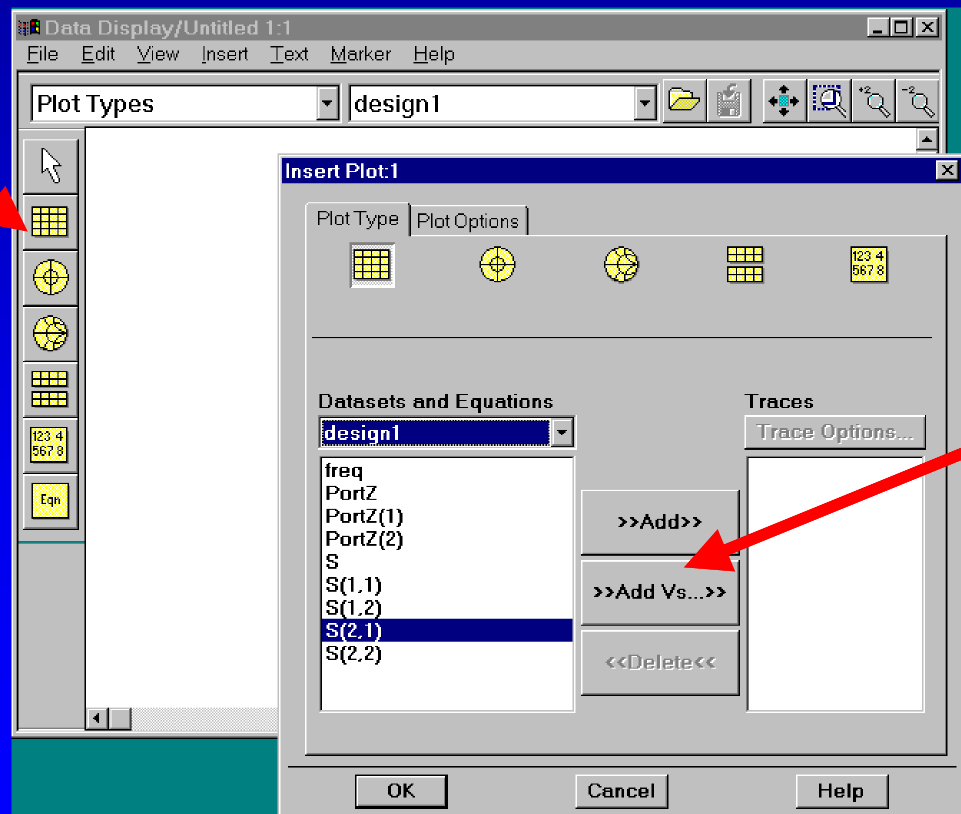
- Go back to the schematic view. Save the design as Design1. After the design is saved, press F7 on the keyboard to initiate a simulation. When the simulation is finished, the following message will be displayed in the simulation window.



Viewing Simulation Results (1)

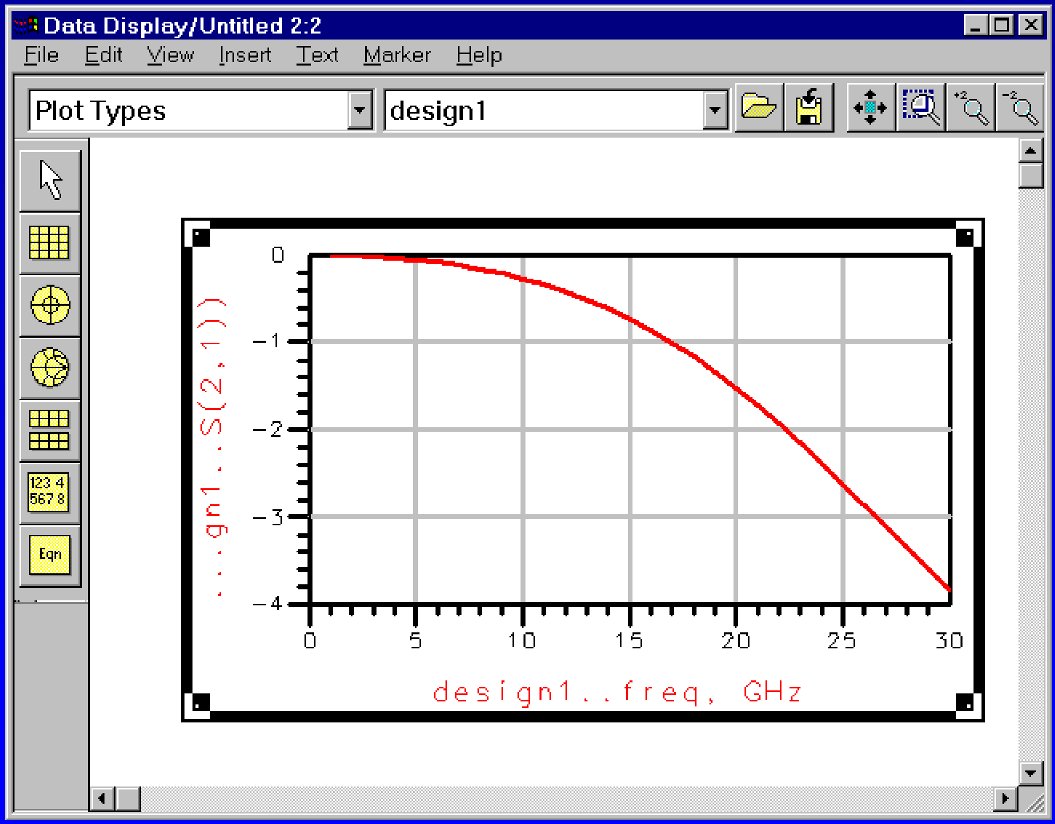
- Go to the schematic view. Choose New Data Display from the Window menu. A Data Display window will then pop up. Click the rectangular plot icon in the Data Display window. Make sure the dataset is coming from “design1”. Choose S(2,1) then click “Add Vs..” In the popup windows that follow, choose dB and the second variable as “freq”.





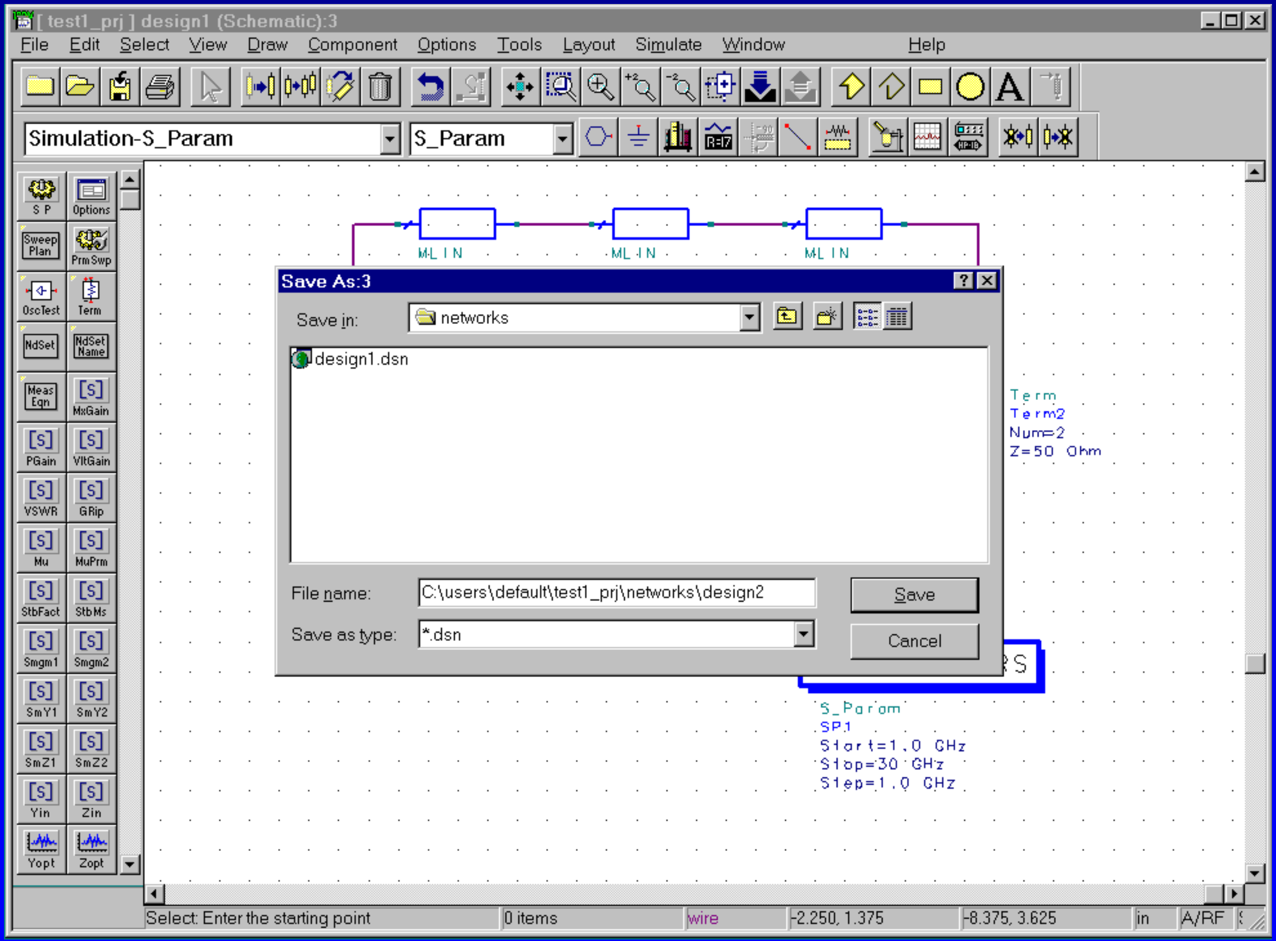
Viewing Simulation Results (2)

- After clicking ok in the dialog box shown in the previous page, the following S(2,1) plot should be displayed in the Data Display window.



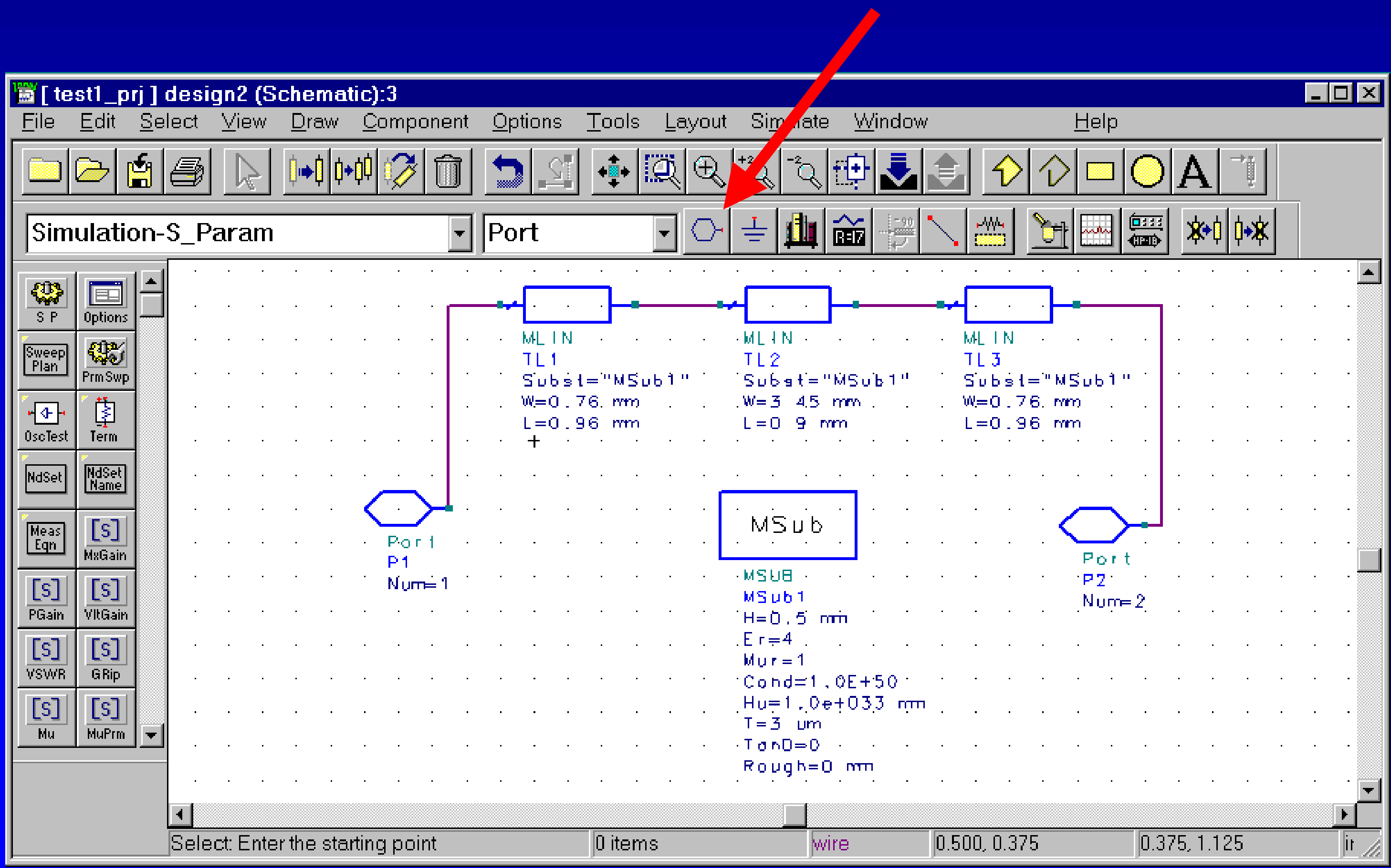
Preparing for Layout Simulations (1)

- Close the Data Display window. Now save the schematic to a new name called “design2”.



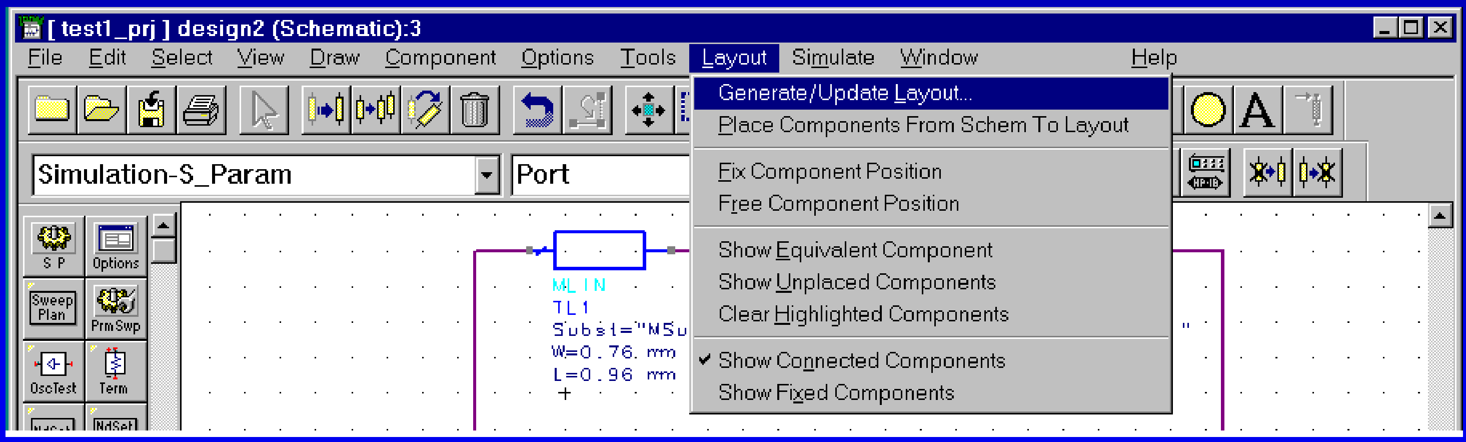
Preparing for Layout Simulations (2)

- In the schematic of "design2", delete the 50-ohm terminals and the S-Parameter block. Add two ports to the schematic as shown:



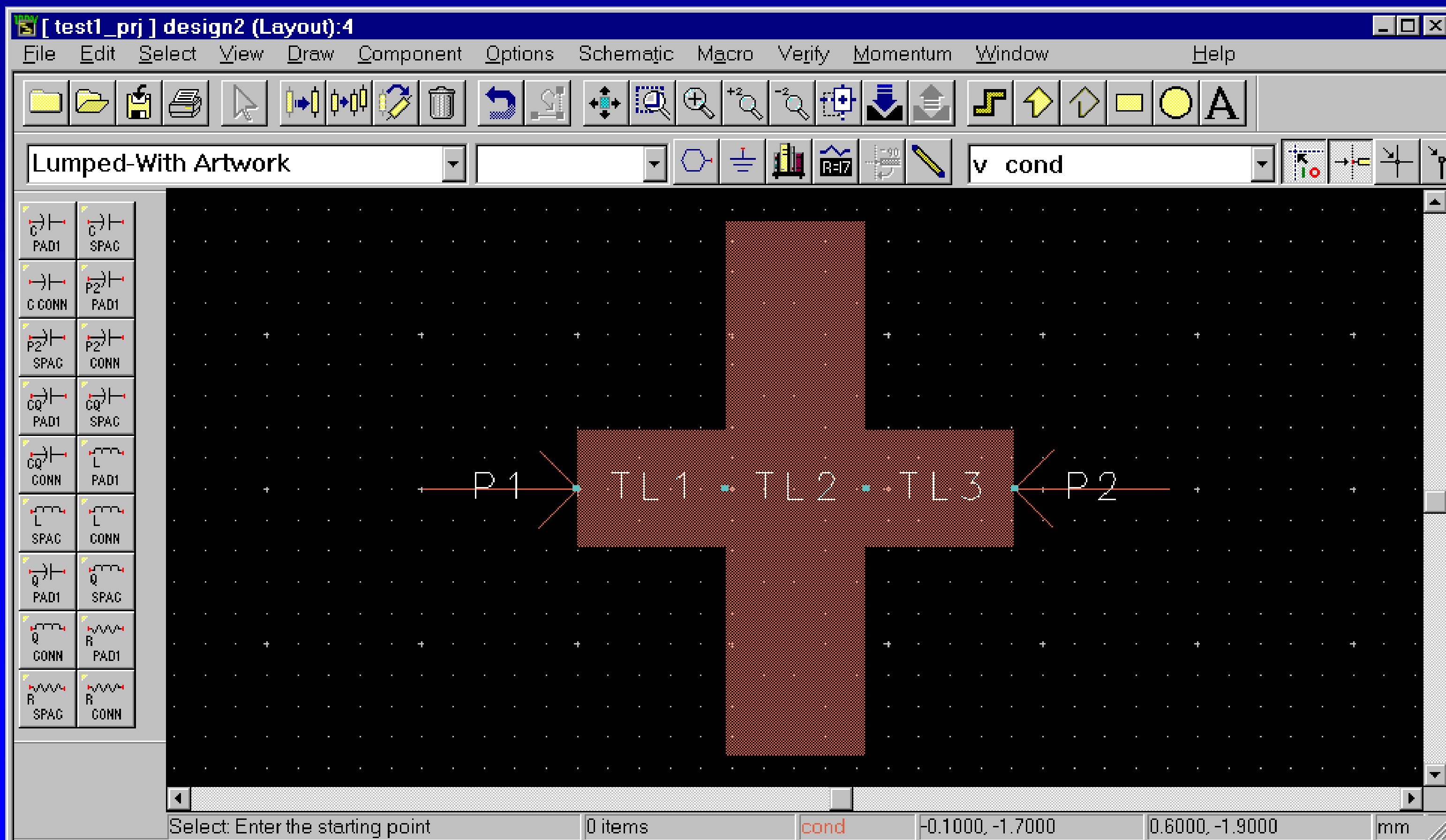
Automatic Layout Generation (1)

- The design is ready for automatic layout generation. Now go to the Layout menu, select Generate/Update Layout. Click OK in the dialog boxes that follow until you see the layout.



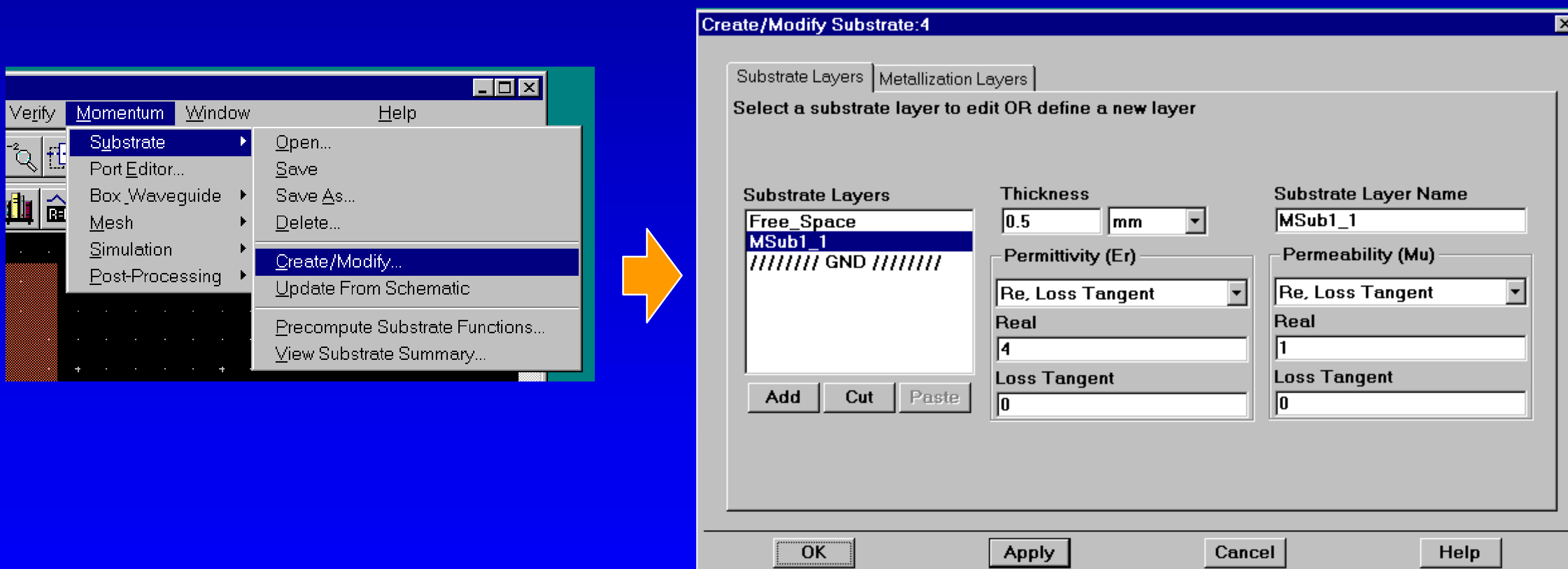
Automatic Layout Generation (2)

- The generated layout should look like the following:



Checking Substrate Parameters (1)

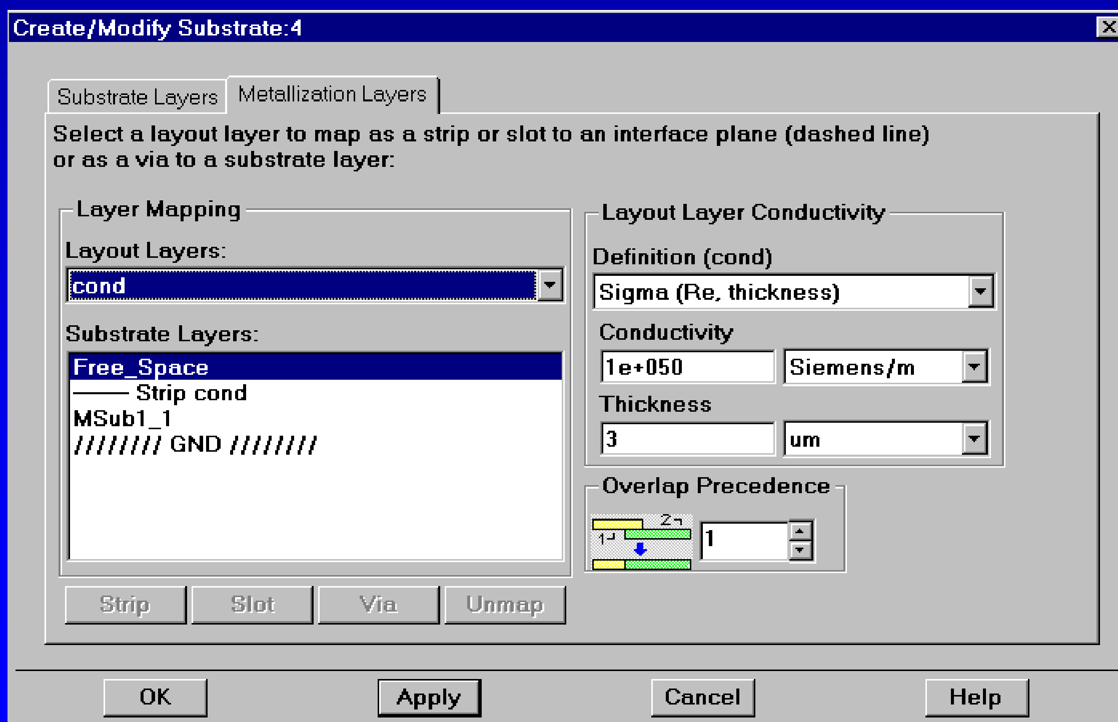
- In the Layout view, go to the Momentum menu, choose Substrate -> Create/Modify to display the substrate parameters defined in the layout. In the Substrate Layers Tab, check if the Thickness and the Permittivity are the same as that in the schematic.



Checking Substrate Parameters (2)

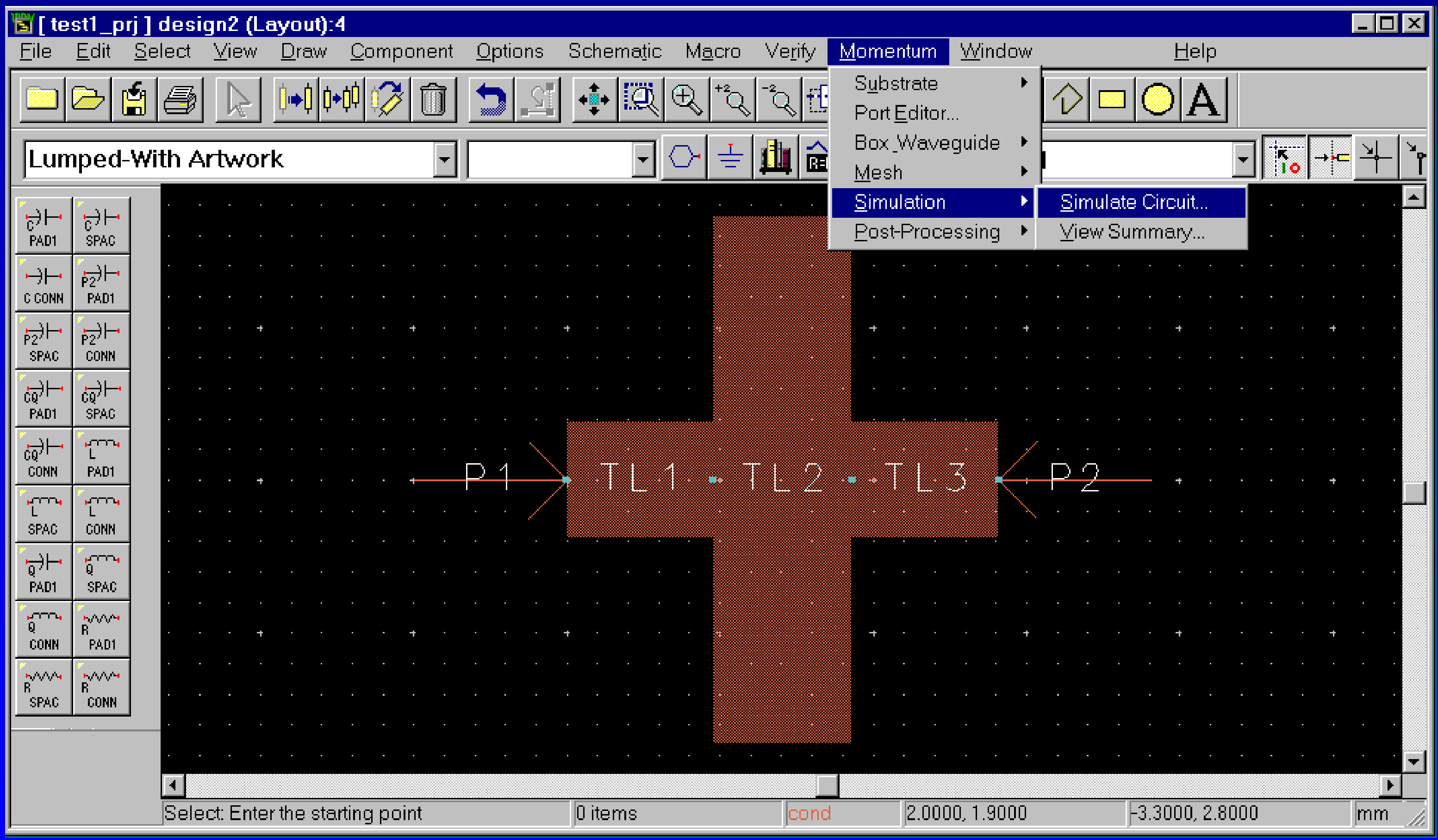
- In the Metallization Layers Tab, check if the thickness of the metal layer agrees with that in the schematic (3um).

- Click OK to go back to the Layout view.



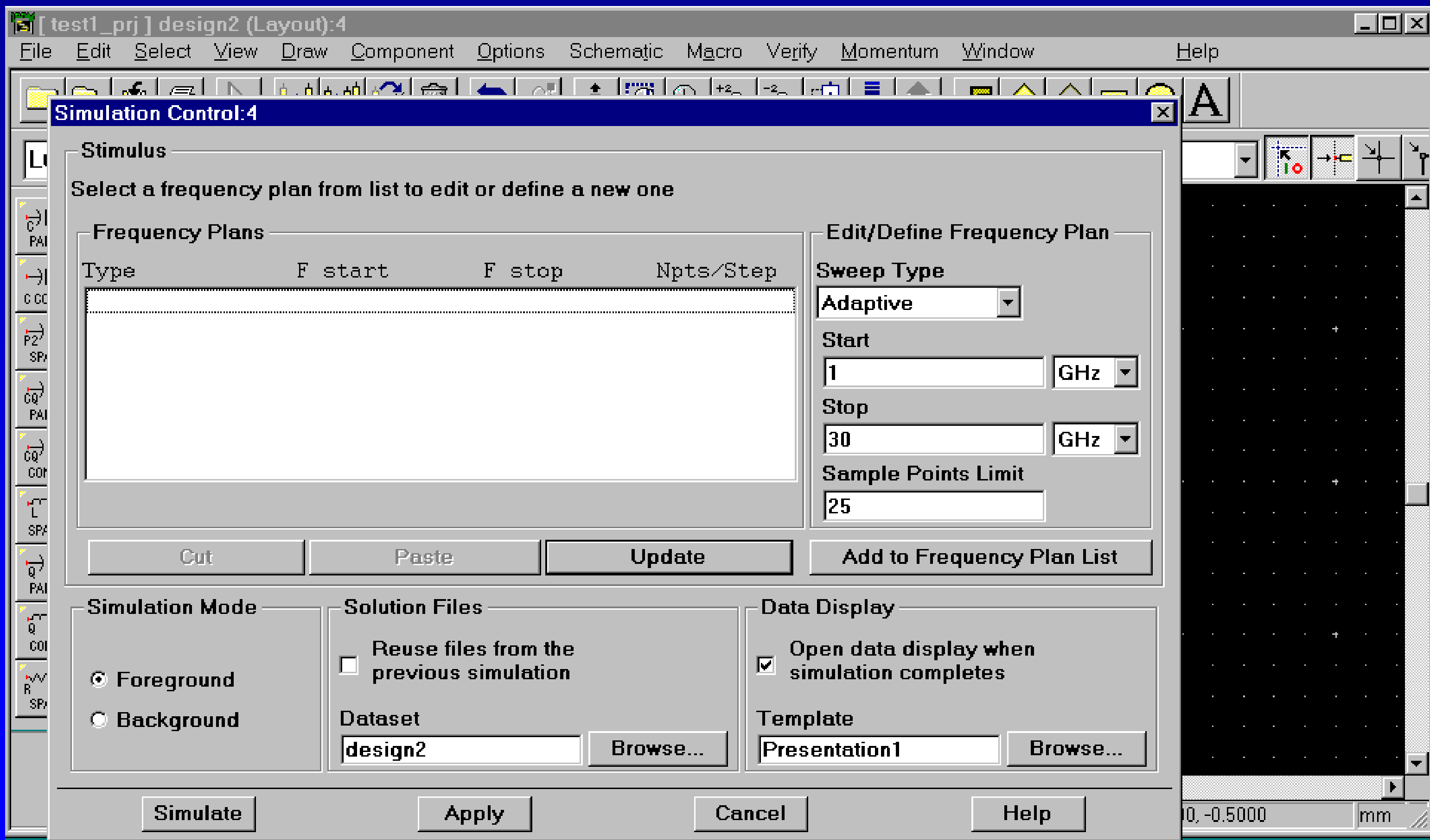
Momentum Simulation (1)

- In the layout view, select Simulation->Simulate Circuit in the Momentum menu.



Momentum Simulation (2)

- In the Simulation Control menu, enter appropriate values in the Start, Stop and Sample Points Limit boxes. Then, click "Add to Frequency Plan List".



Momentum Simulation (3)

- After the frequency sweep plan has been entered. Click the Simulate button to perform a simulation on the layout. By the end of the simulation, the results will pop up in a Data Display window.