4

This chapter shows how to make S-parameter simulations and how to determine matching network values.

Lab 4: S-parameter Simulations

OBJECTIVES

- Measure gain and impedance with S-parameters
- Use a sweep plans, parameter sweeps, and equation based impedances
- Plot and manipulate data in new ways

About this lab: This lab continues the mixer testing by making various S-parameter measurements to determine circuit performance: gain and impedance.

PROCEDURE

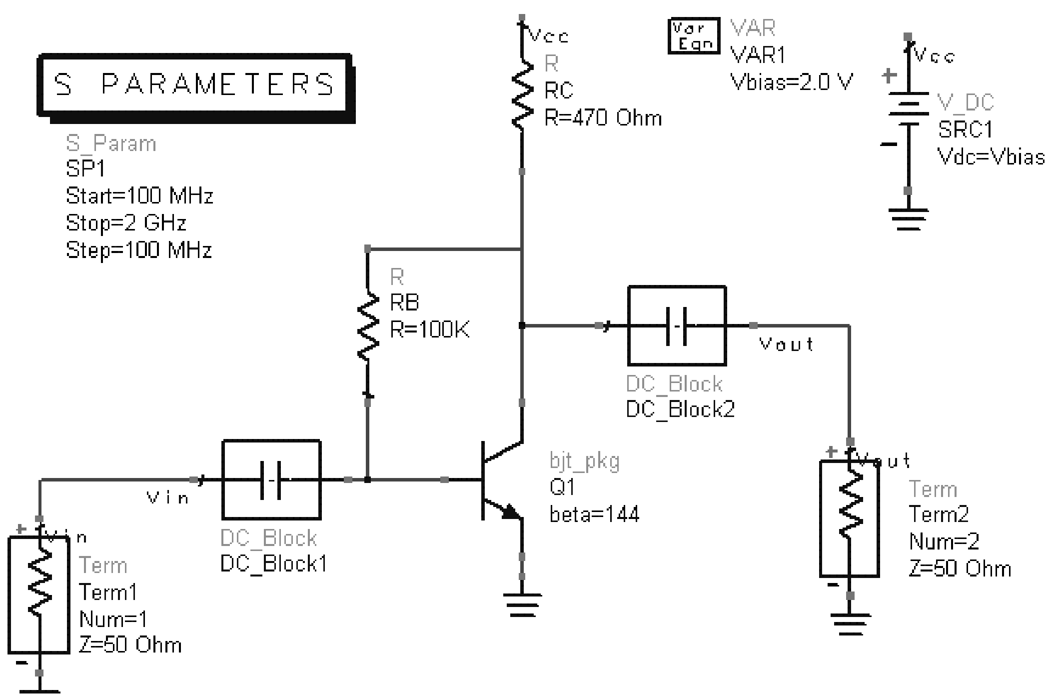
1. Copy the last lab and save it as a new design named: s_params.

After saving the schematic as **s_params**, continue modifying the design to match the schematic shown here, according to the following steps:

a. Delete the AC sources and simulations. Also delete the measurement equations, parameters sweep, etc. – These are components you will not use for S-parameter simulations.

b. Insert an S-Parameter simulation controller (Simulation-S_Param palette) and set: **Start=100 MHz**, **Stop=2 GHz**, and **Step=100 MHz**.

c. Insert S-parameter port terminations (**Term**) instead of a source and load. The Node Names are the same as before: Vin and Vout.



2. Simulate and display results: use the data display options

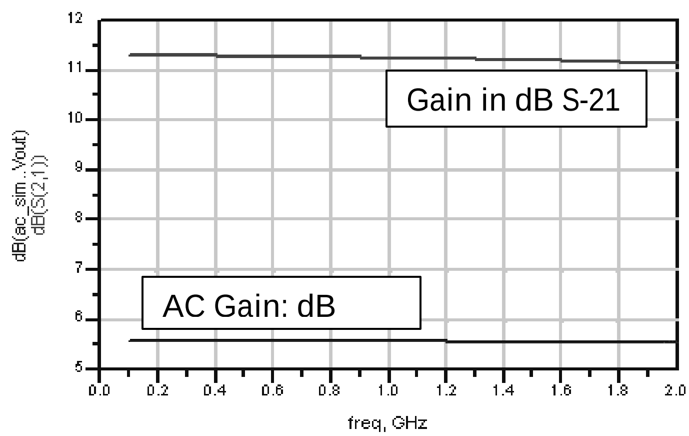
The following steps will compare the S21 measurement to the AC simulation data and show you more about plots, traces, markers and text.

a. Be sure the name of the dataset is: **s_params** and then **simulate**. It is a good idea to always check dataset names before simulating.

b. When the simulation is finished, open a **new Data Display** window and save it as **s_params**. Then insert a rectangular plot of S-21 (dB).

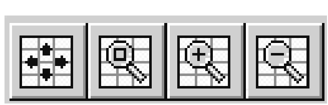
c. On the same plot, insert the trace the db of **Vout** from the **ac_sim** dataset from the last lab.

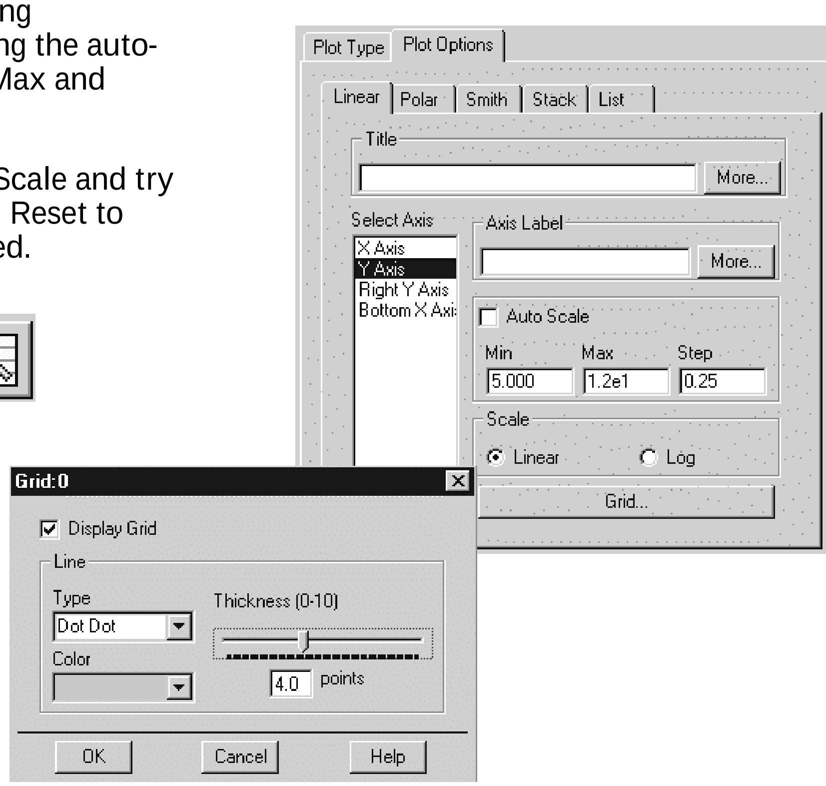
NOTE on S-21 vs dB gain from AC simulation: The dB values of gain are the same because the AC simulation uses the standing wave voltage only. But the S-parameter simulation uses output power divided by incident power (V and I). In addition, the S-parameter source impedance is 50 ohms and the V_AC source 0 ohms.



d. Edit the plot (double click). Go to **Plot Options** and try adjusting both axes by deactivating the auto-scale and setting: Min, Max and Step.

e. Reset the axes to Auto Scale and try the plot-zooming icons. Reset to Auto Scale when finished.

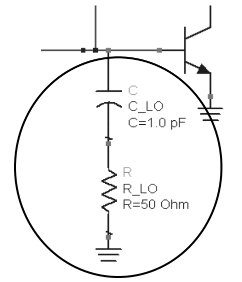




f. Also, click on the Grid button and try changing the grid as desired.

You may want to use these features later.

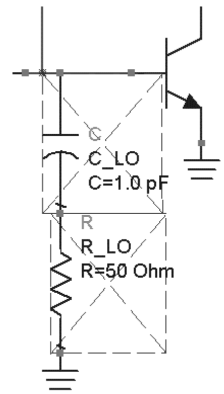
3. Add the LO impedance: series R-C to ground



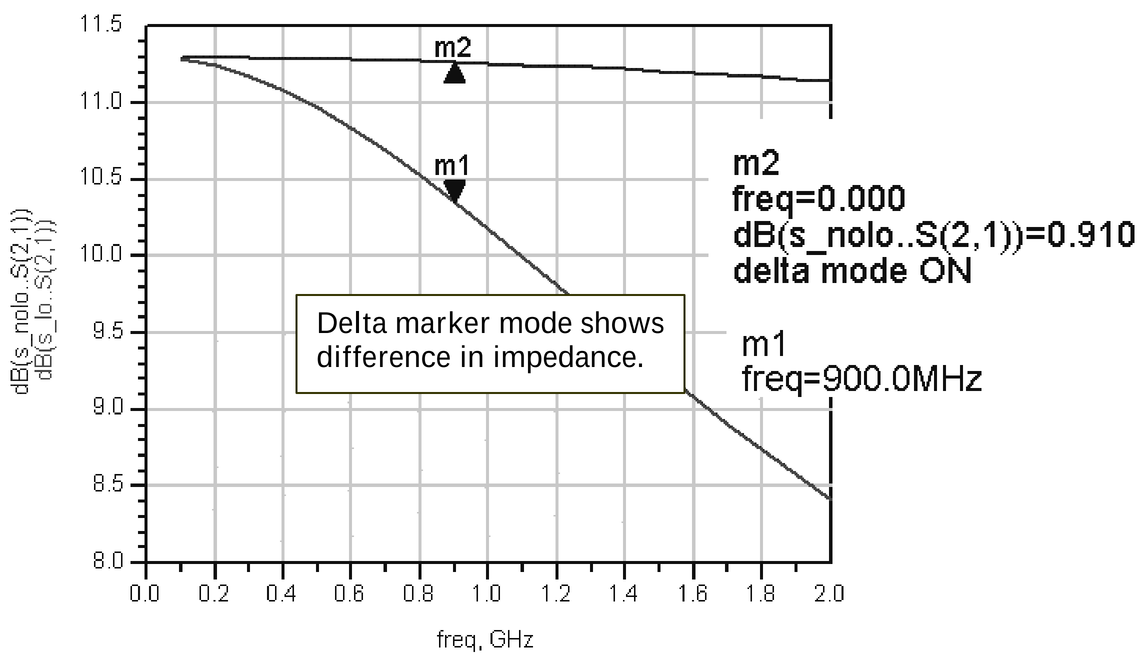
a. In the schematic, insert the following components to represent the impedance of the local oscillator: **C=1.0 pF** and **R=50 Ohms** and ground. Insert onto the transistor base.

b. **Simulate S-21 again** with the frequency range from 100 MHz to 2 GHz (100 MHz steps) and name the dataset: **s_lo**.

c. Deactivate the LO components, and simulate again with the dataset name: **s_nolo** to compare results.



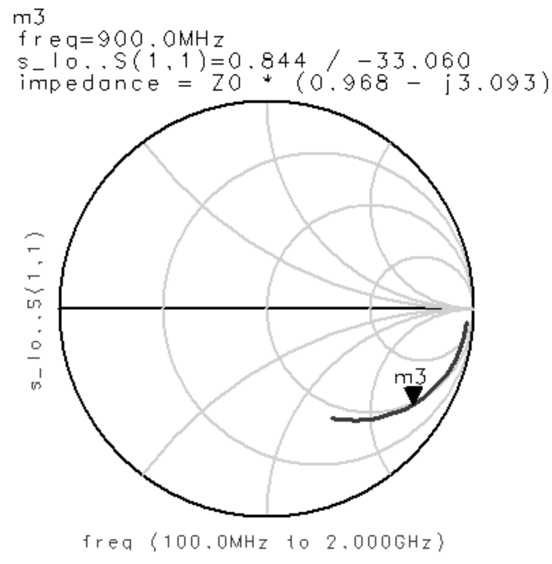
d. In the data display window, insert a new plot with the two S-21 traces: with the LO and without the LO impedance. The display should look similar to the one shown here. Select the two markers and click **Marker > Delta Mode On**. The delta S-21 is about 0.9 dB.



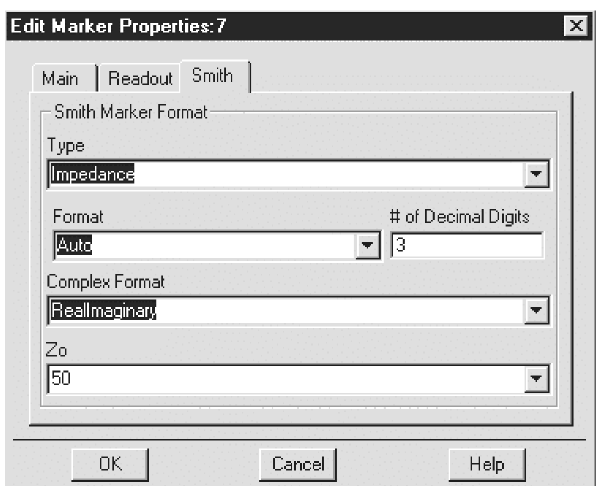
e. Save the data display window as **s_params**. Remember the data display windows are .dds files and the datasets are .ds files in the data directory.

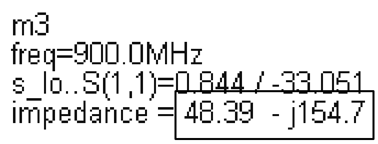
4. Plot the S11 impedance

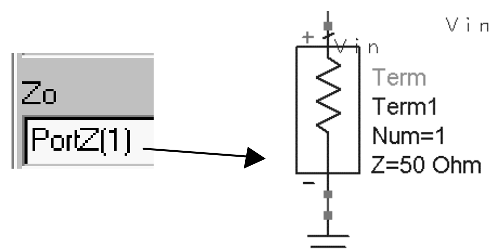
a. In the same data display window, insert a Smith chart with the S-11 trace from the **s_lo** dataset. Insert a marker at 900 MHz and notice that the marker shows the reflection coefficient (magnitude/ angle) and also the impedance (real and imaginary). For the design, there is an obvious mismatch here.



b. Edit the marker readout (double click on the marker readout). When the dialog appears, change Zo to 50 and click OK. The marker will now give you the real and imaginary value of S-11 in ohms:



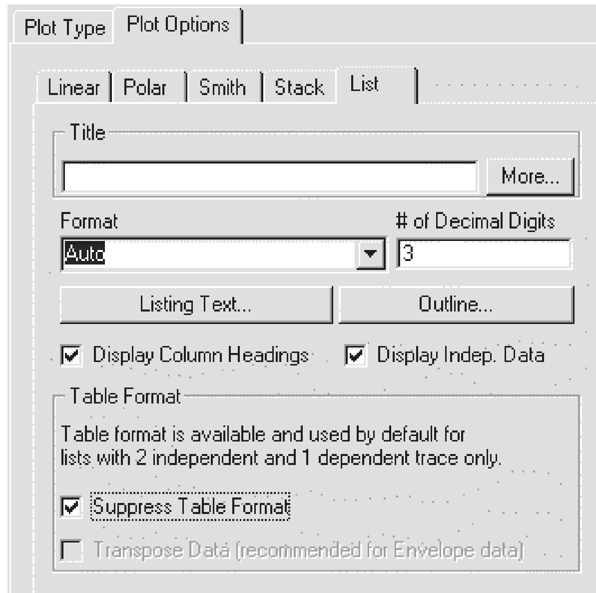


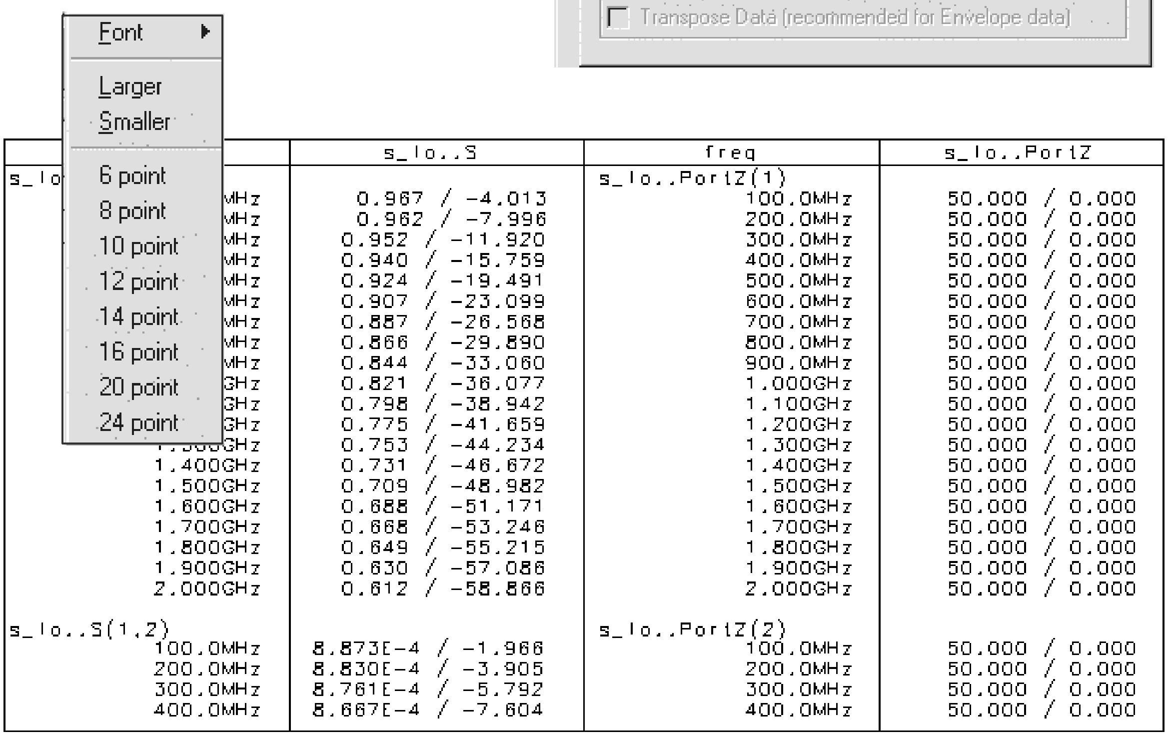


c. Change the Zo setting to PortZ(1) as shown and you will get the same answer as setting Zo to 50. However, if you did have a different port impedance (for example Z=75 Ohm) then the PortZ setting would calculate the readout using 75.

d. Insert a **list**. Then select **S** data and **PortZ** and add them. Then use **Plot Options** to **Suppress Table Format**. This will display all four S-parameters in a tabular format as opposed to selecting only one of the S values. In addition, the port impedance of the termination is given too.

e. In the Data Display window, use the scroll buttons or zoom icons to view the data. Also, try changing the font type and size in the list (you may need this for a presentation) by clicking the command: **Text > Font**.





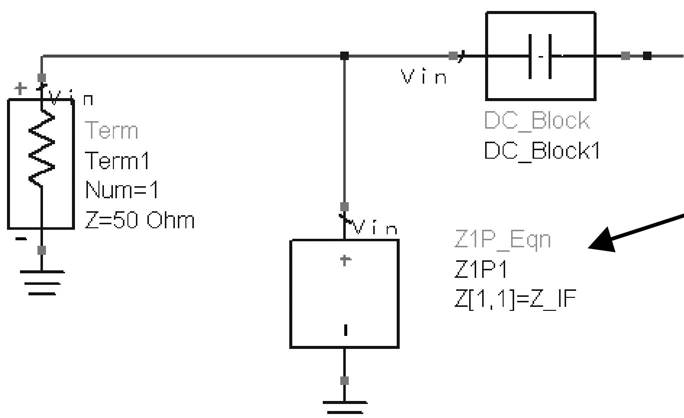
[illegible]	s_lo..S	freq	s_lo..PortZ
s_lo[illegible]		s_lo..PortZ(1)	
[illegible]	0.967 / -4.013	100.0MHz	50.000 / 0.000
[illegible]	0.962 / -7.996	200.0MHz	50.000 / 0.000
[illegible]	0.952 / -11.920	300.0MHz	50.000 / 0.000
[illegible]	0.940 / -15.759	400.0MHz	50.000 / 0.000
[illegible]	0.924 / -19.491	500.0MHz	50.000 / 0.000
[illegible]	0.907 / -23.099	600.0MHz	50.000 / 0.000
[illegible]	0.887 / -26.568	700.0MHz	50.000 / 0.000
[illegible]	0.866 / -29.890	800.0MHz	50.000 / 0.000
[illegible]	0.844 / -33.060	900.0MHz	50.000 / 0.000
[illegible]	0.821 / -36.077	1.000GHz	50.000 / 0.000
[illegible]	0.798 / -38.942	1.100GHz	50.000 / 0.000
[illegible]	0.775 / -41.659	1.200GHz	50.000 / 0.000
[illegible]	0.753 / -44.234	1.300GHz	50.000 / 0.000
1.400GHz	0.731 / -46.672	1.400GHz	50.000 / 0.000
1.500GHz	0.709 / -48.982	1.500GHz	50.000 / 0.000
1.600GHz	0.688 / -51.171	1.600GHz	50.000 / 0.000
1.700GHz	0.668 / -53.246	1.700GHz	50.000 / 0.000
1.800GHz	0.649 / -55.215	1.800GHz	50.000 / 0.000
1.900GHz	0.630 / -57.086	1.900GHz	50.000 / 0.000
2.000GHz	0.612 / -58.866	2.000GHz	50.000 / 0.000
s_lo..S(1,2)		s_lo..PortZ(2)	
100.0MHz	8.873E-4 / -1.966	100.0MHz	50.000 / 0.000
200.0MHz	8.830E-4 / -3.905	200.0MHz	50.000 / 0.000
300.0MHz	8.761E-4 / -5.792	300.0MHz	50.000 / 0.000
400.0MHz	8.667E-4 / -7.604	400.0MHz	50.000 / 0.000

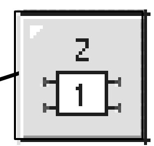
5. Add frequency sensitive terminations for the RF & IF

Frequency sensitive Z ports from the Equation-based Linear component library allow you to describe how a termination responds to changes in frequency. For example, for S11 matching, the Z port on the output is programmed to be an RF short to ground like a filter that you would use. Conversely, the Z port at the input is programmed to be an IF short to ground. This provides a better approximation of the final S-parameters after creating the matching networks.

NOTE: You will move components around and rewire several items to complete the steps. Take your time and create an organized schematic.

a. From the palette, select the **Eqn Based-linear** components and insert a one port **Z Eqn** in parallel with the input.



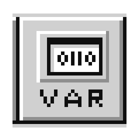


Eqn Based-Linear Z port is programmed to be a short at the IF frequency – used when creating the output matching network (S22).

b. Assign the value of Z[1,1] to be a variable: **Z_IF.**

c. Insert another **Z 1-port Eqn** in parallel with the output impedance and assign the value of Z [1,1] to be the variable: **Z_RF**.

d. Insert a **VAR** (variable equation) and edit it to declare the values of Z_IF and Z_RF as shown here:



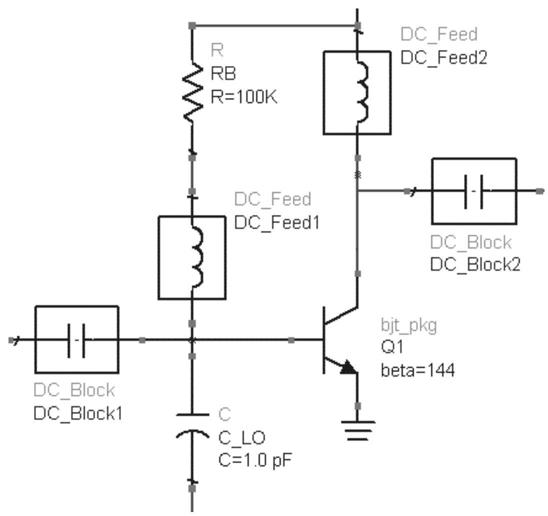
VAR
VAR_Z
Z_IF=if freq <100 MHz then 0.001 else 1e99 endif
Z_RF=if freq >100 MHz then 0.001else 1e99 endif

6. Insert ideal DC feed inductors

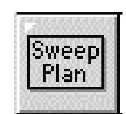
a. Go to the Lumped Components palette and insert a **DC feed** inductor between the transistor base and the resistor RB.

b. Insert another **DC feed** between the collector and the resistor RC.

Partial view of schematic shows DC_Feed components used as ideal inductors: short to DC and open to AC



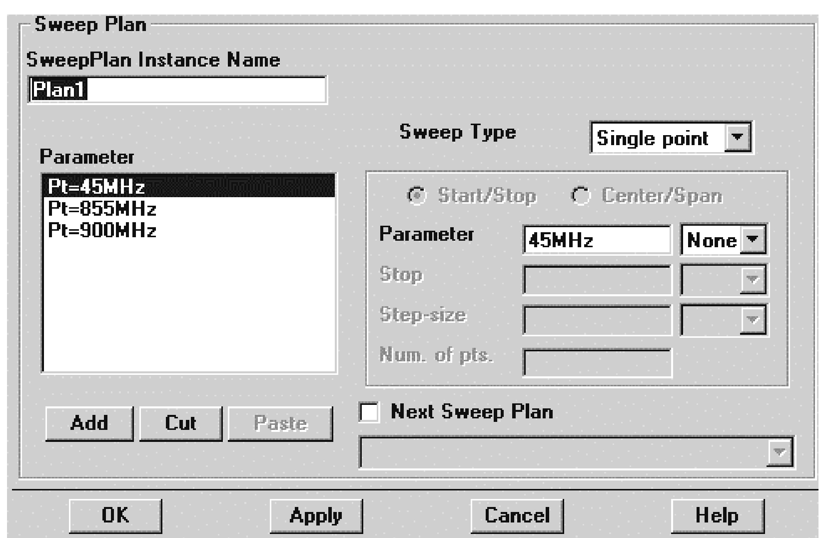
7. Set up a Sweep Plan (SweepPlan)



a. Sweep Plans are in simulation palettes - insert a **Sweep Plan.**

b. Set the sweep for three single points: RF, IF and LO frequencies. First, click Sweep Type and set it to **Single point**. Then type in the frequency points and click the **Add** button after every entry. Use **Add** or **Cut** to remove or change the position of any unwanted parameters.

c. Click **Apply** and **OK** when the dialog has these entries only.



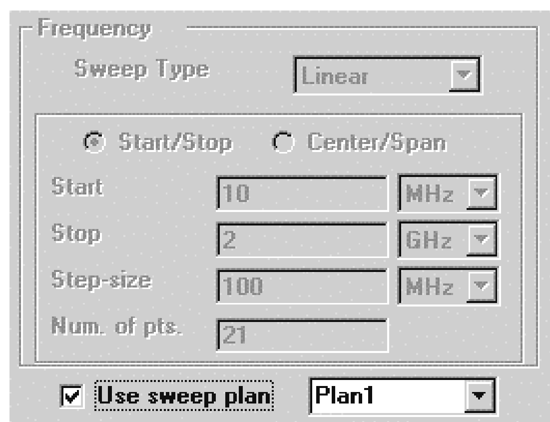
8. Edit the S-Parameter Simulation Controller

a. Select the S-parameter simulation controller and click the **Edit icon** – this is the same as double clicking on the component.

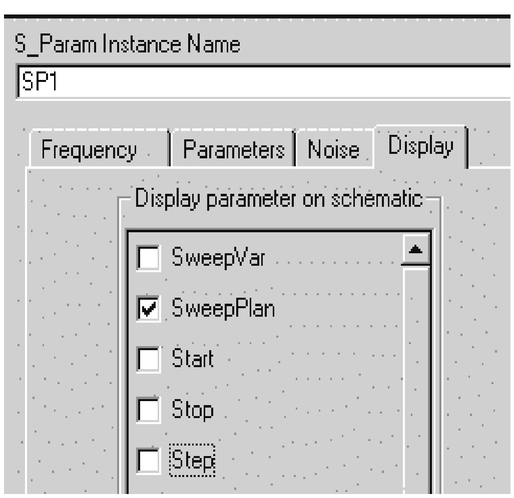
Edit Parameters icon

b. In the **Frequency tab**, click the box **Use sweep plan** and select the sweep plan **Plan 1** which you have already set up – the controller recognizes inserted sweep plans. Click Apply and note that the start and stop settings should now be grayed out.

NOTE: Please be sure that the Sweep Type is **NOT** Single Point. If it is, only one Frequency would be used for simulation.

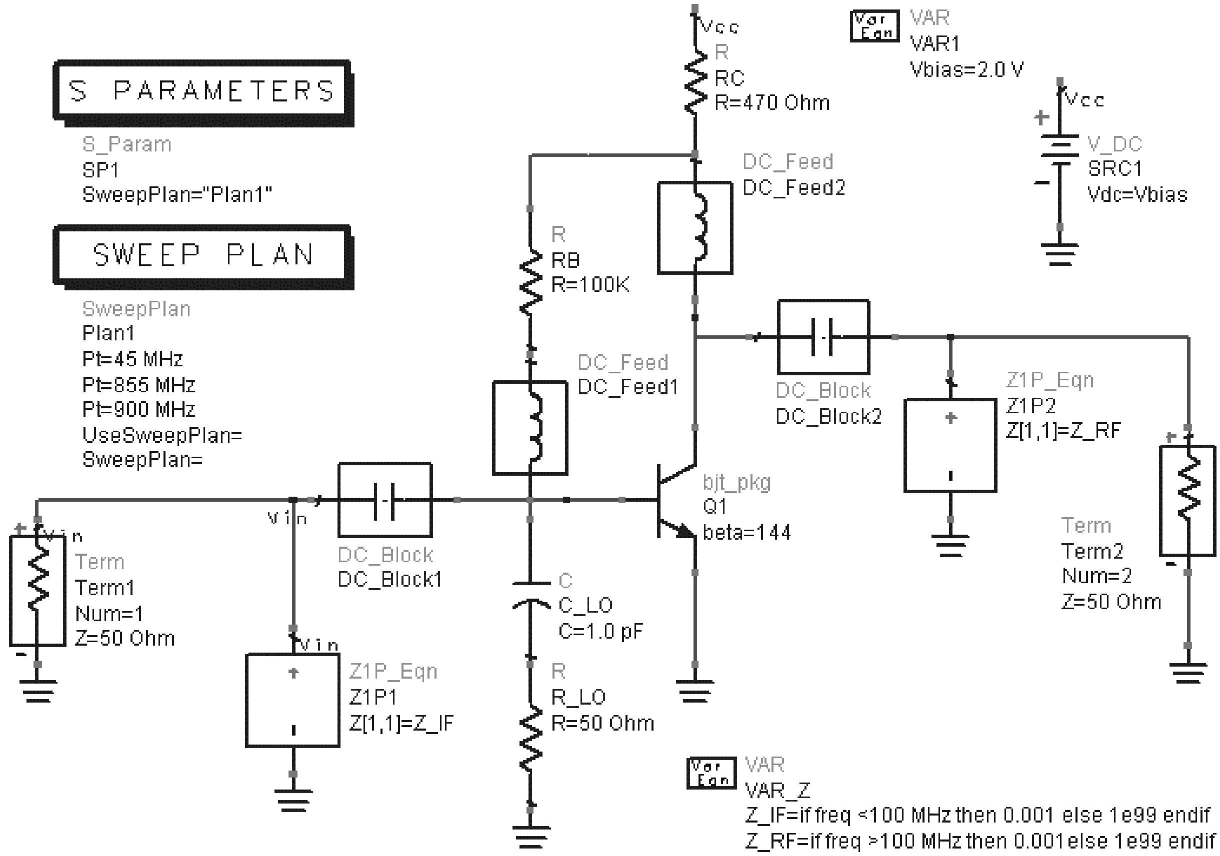


c. In the Display tab, check the SweepPlan and remove the Start, Stop and Step check boxes. Click OK and the simulation controller should now look like the one here.



S-parameter simulation controller set to assign FREQ to be the sweep plan values.

9. Check the circuit - it should look like the one shown here:

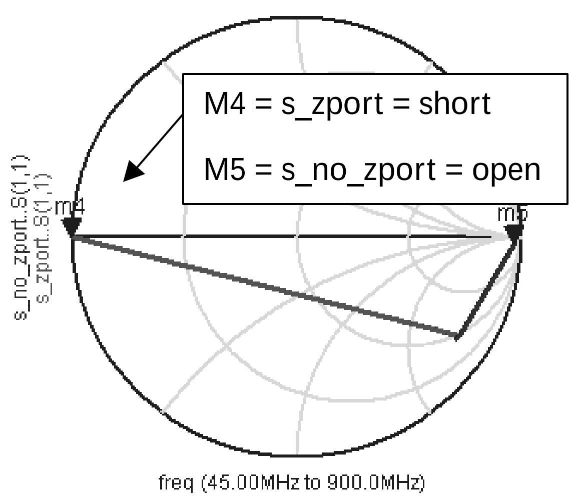


a. Be sure the LO impedance is activated. Set up a new dataset name: **s_zport** and **Simulate**. Then plot the **S-11** data on a Smith Chart.

b. **Deactivate the Z-ports**, and simulate again with the dataset name: **s_no_zport**.

c. Add the **s_no_zport** trace to the Smith chart and notice the difference at 45 MHz between the two traces by putting new markers at 45 MHz for each trace. As you can see, the IF response is a short with the Z port and almost an open without the Z ports (with opposite phase).

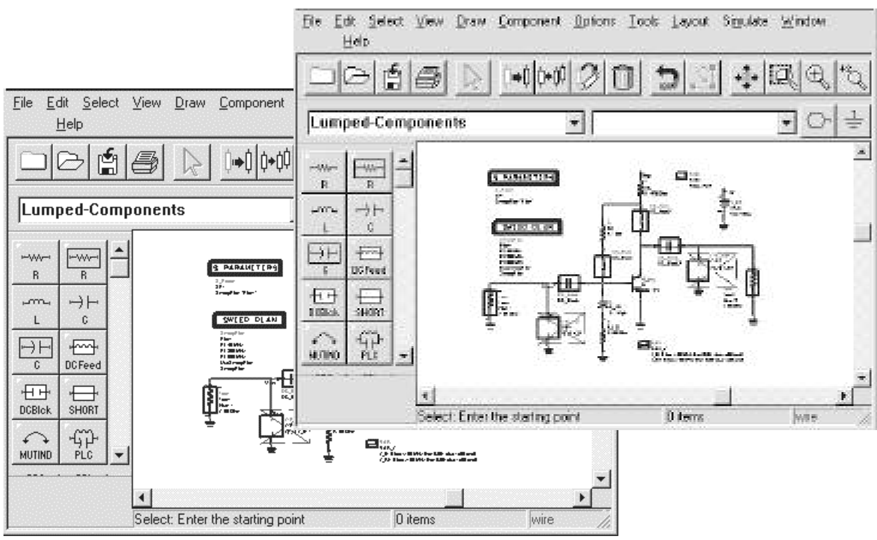
d. **Save** the data and schematic (s_params).



10. Modify the BJT_PKG sub-circuit (B-C capacitance)

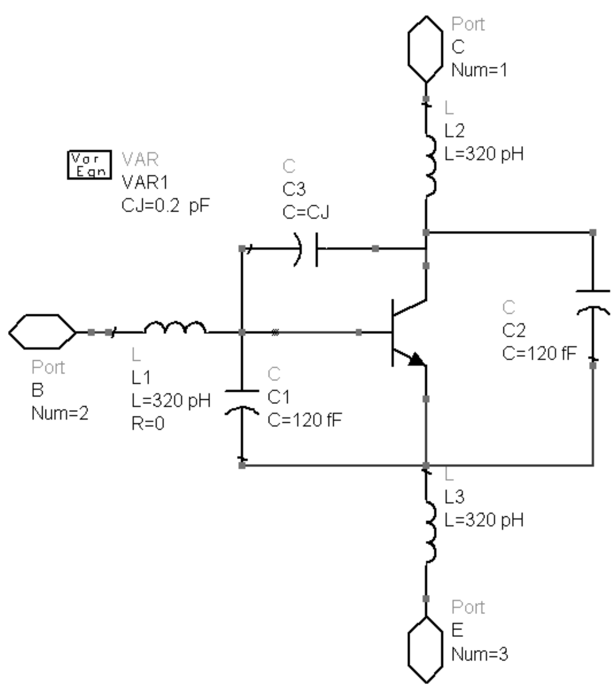
The following steps will further demonstrate the use of Z ports, sub-circuits and simulations in the hierarchy.

a. **Open a second schematic window** as follows: from the current schematic window (s_params) click Window > Schematic or use the **hot keys: Ctrl+Shift S**. Now you have two windows of the same design. This is good for viewing details of a large schematic while keeping the overall design viewable.



b. In the new schematic window, **push into the bjt_pkg**. Now, in this lower level hierarchy, add a capacitor between the collector and base **C=CJ** and add a **VAR = 0.2 pF** being sure to put spaces between the number and the units as shown.

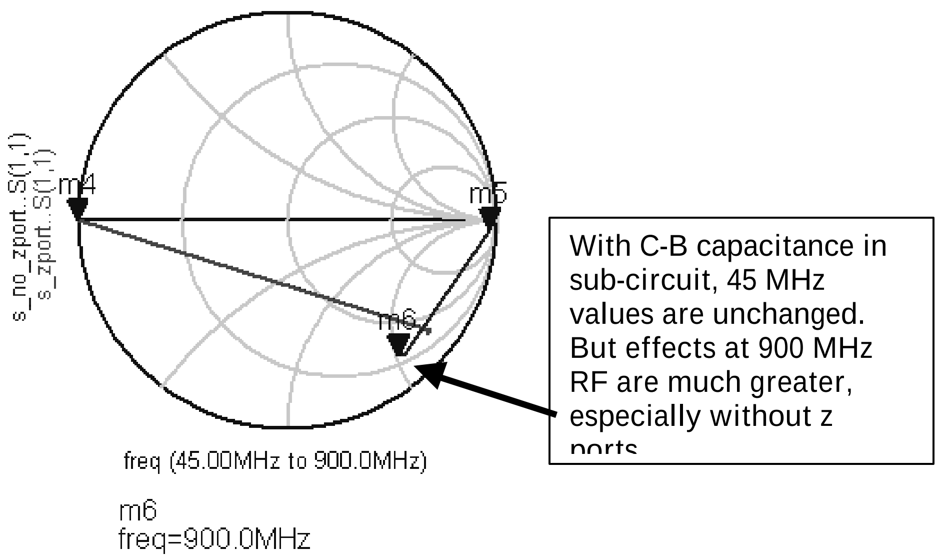
NOTE: This will have a similar effect as modifying the Cjc parameter in the model card.



c. At this lower level, press the **F7 simulate** key. You should get an error message in the status window because there is no simulation controller.

d. Move the cursor back to the other schematic window (higher level hierarchy with the simulation controller). Keeping the same dataset name from the previous simulation (**s_no_zport**), and **the z ports deactivated**, **simulate** and note the change to the S11 data.

DESIGN NOTE: With the added capacitance (base to collector), at 45 MHz there is little or no change. But at 900 MHz, the input impedance is more capacitive as it would be with a real device. Also, the z port (Z_RF) now shows that it can be used to create a mathematical representation of the behavior of a matching network. In this case, it will allow you to start designing the input match with the output already represented, thus eliminating iterations.



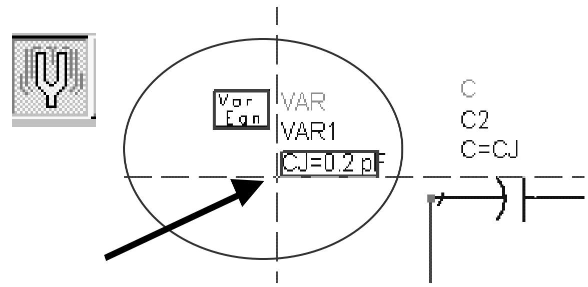
11. Tune the Sub-circuit Variable

The following step shows how to tune a variable (VAR) in a sub-circuit and keep the data separate from the existing datasets. This will require moving windows and dialogs around on the screen and using both schematic windows. This step is not critical to designing the mixer but it does demonstrate a procedure you may need in your own designs later on.

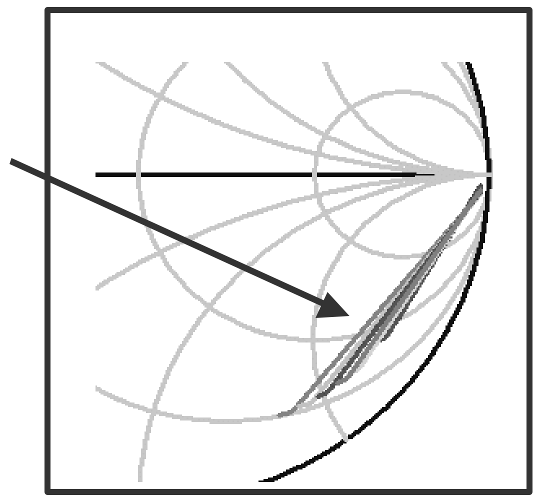
a. Setup a new **dataset name: s_cj_tune** and **Simulate** again. This will create a dataset.

b. **Close the existing data display** window and **open a new data display** window. The default dataset name s_cj_tune should appear. Insert a Smith chart of the S-11 data.

c. In the upper level schematic, start the **Tune mode** (Simulate> Tuning). You will see the status window and the Tune control dialog appear. Next, move the cursor back to the **bjt_pkg sub-circuit** window and select the **CJ parameter value.**



d. Position the data display window so you can see the new trace values resulting from the tuning. Here is a case where small changes occur and so you can zoom into the Smith chart.



e. To easily end tuning, press the keyboard **Esc** key or use the right mouse button End Command.

f. **Delete the Smith chart** so the data display is empty.

g. In the subcircuit: **remove the Capacitor and VarEqn**.

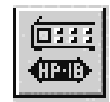
NOTE: Save the s_params design. It will be used for the next lab to develop the final input and output matching networks for the mixer.

NOTE on the next 2 steps: Do these only if you have time. They are not required for the mixer design. They demonstrate how to write or read data into ADS.

12. Reading and Writing S-parameter Data with an S2P file

You can read or write data in Touchstone, MDIF, or Citifile formats. ADS can convert supported data into the ADS dataset format. Typically, these data files are put in the project directory but they can also be sent to the data directory. You can control where they reside.

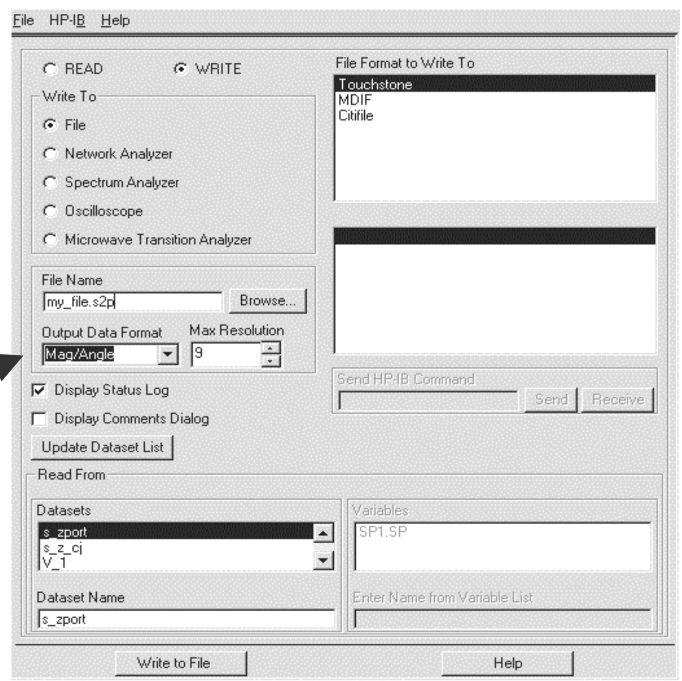
a. In the data display window (s_params.dds), click on the HP-IB icon (Instrument Server).



b. When the dialog box opens, click the box to **WRITE** and Write to **File** then select the **Touchstone** format.

IMPORTANT NOTE:

Be sure to select the type of Output Data Format you want whenever you use this feature. For this lab, select Mag/Angle.



c. In the File Name field, instead of using the name *default*, type in the name: **my_file.s2p**.

d. Select the Output Data Format as **Mag/Angle**. In the Datasets field, select the dataset **s_zport** to be translated and select the variables (shown above).

e. Click the **Write to File** button and then check the Status Window. If everything is correct, you will get a message confirming the dataset write was successful: my_file.s2p is now a Touchstone file.

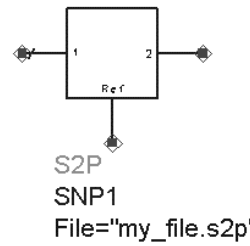
13. Assign the S2P component to the data file and simulate

In this step, you will write and read an s-parameter Touchstone file using the s2p component. This is similar to downloading an s-parameter file from the web for use in a simulation.

a. Open a new schematic window (untitled) using **Ctrl N** where N means new (schematic).

b. Insert an **S2P** component (type it in or get it from the palette **Data Items**. Notice that the component variable (*file*=) is not yet assigned.

c. To assign the data, type in the file name or edit the S2P component. Another dialog box will appear. Now, set the browser to look for **All Files** (***.***).



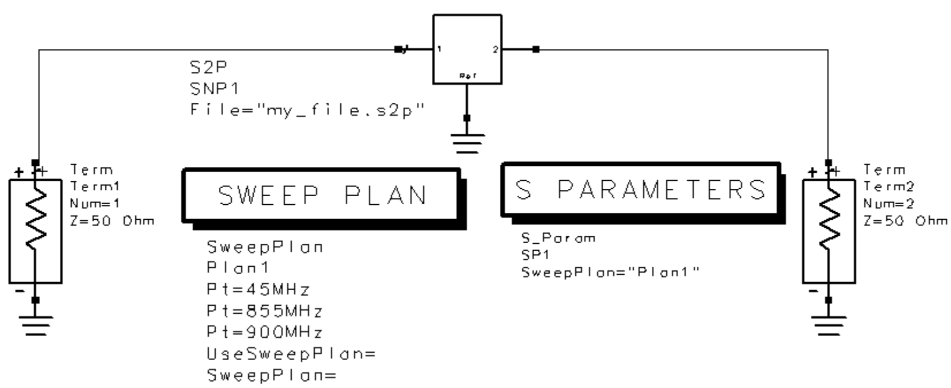
d. Next, **browse** for the file <u>in the directory where the data was written</u>. Then use the **Open** button to select the file: **my_file.s2p**.

NOTE: You can use a text editor to look at or to modify the values.

e. In the untitled schematic insert the template: **S-param** (command: File > Insert Template) and wire the S2P component to the ports and insert a reference ground

f. Go back to the **s_params** schematic and copy the **Sweep Plan** to the buffer and paste it into the untitled schematic. Set the Simulation controller to use the **Sweep Plan**.

g. **Simulate** with the dataset name: **my_s2p**. Plot the results in the Data Display window to verify the simulation.



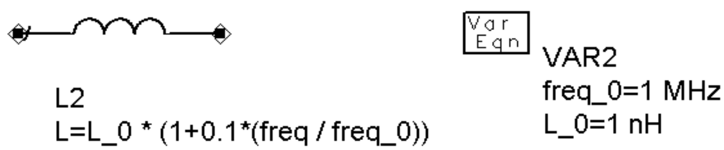
h. Save the design and data with the name **s_2p** and close the windows.

EXTRA EXERCISES:

1. Translate data from a Touchstone file into a dataset. Use the **Instrument Server** window and **READ** the file back into the data directory as an ADS dataset. Then simulate the S2P component with another name and save the file.

2. Use a real library device and simulate both with and without the Z ports to actually see the difference in S11. For example, use a device from one of the BJT libraries.

3. Try writing an equation to vary the value of a package parasitic – for example, a value of L that varies with frequency:



4. Try writing an equation so that the port impedance changes with frequency and then verify that the marker readout calculates the proper value of impedance.

5. Use a Z 2-port and create a conjugate match based on the initial S11 data.

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

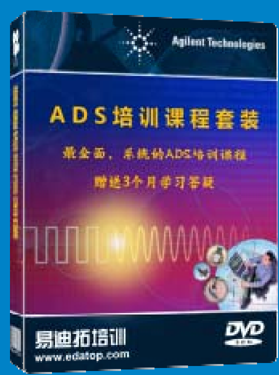
该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

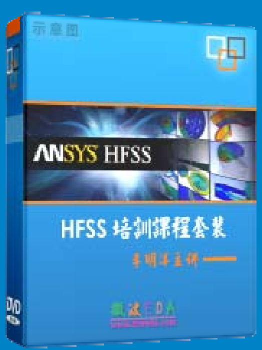
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html



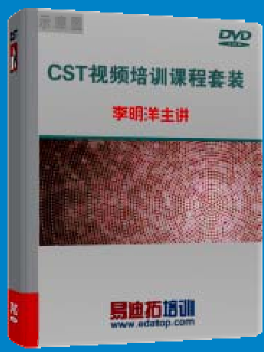


HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装



该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com