5

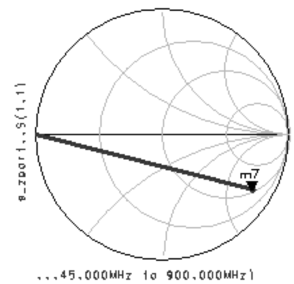
This chapter shows various ways of creating matching networks by sweeping values and using optimization.

Lab 5: Matching & Optimization

OBJECTIVES

- Create an input match to the RF and an output match to the IF
- Tune and Optimize to achieve matching goals

Mixer Design Note: From the Smith Chart S-11 results in the last lab, it appears that a series inductor can be added to the input as a first step in moving toward the center of the Smith chart for the RF match at 900 MHz. However, this does not take into consideration the other L and C components. But as a first step, it is reasonable to add the series inductor and see the effects of tuning as ideal components are replaced with real values.

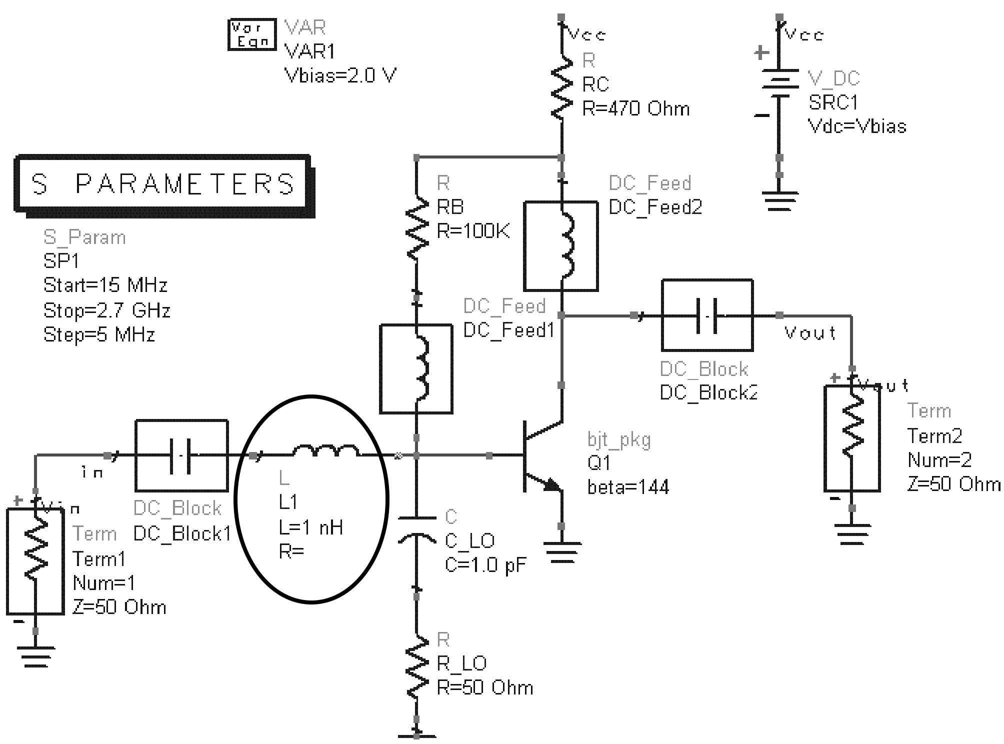


PROCEDURE

1. Create a new schematic design for the input match.

a. Use the s_params design (last lab) and save it as: **s_match**.

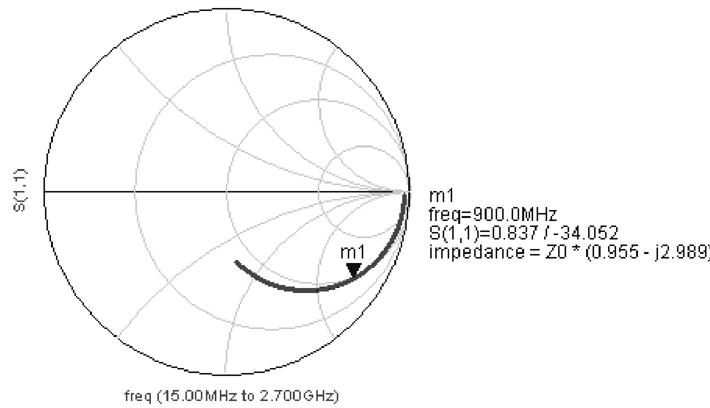
b. Insert an **inductor L in series** to the input, as shown. Your circuit should look like the one here where the Sweep Plan and Z-ports are removed and set the S-parameter controller to sweep 15 MHz to 2.7 GHz – this will simulate most of the frequencies that will result when the LO is added.



c. Check the sub-circuit to be sure there is no capacitor across the base-collector (from the last lab).

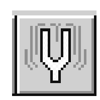
d. **Simulate** and **display S-11** in a new data display window. Position the dds window next to the schematic so you can see both at the same time. The default dataset should be the same name as the schematic: **s_match**. The results of the swept analysis should look like the plot here where a marker is added to show the value of S-11 at 900 MHz:

Use the keyboard arrow keys and the mouse to position the marker



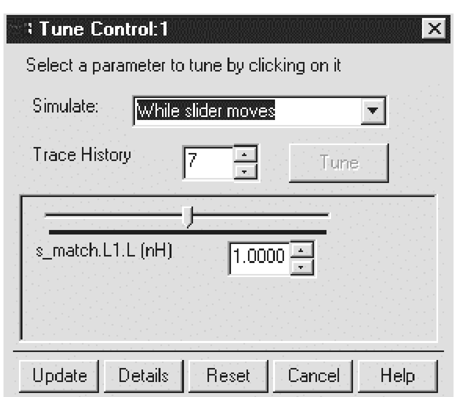
2. Start tuning the inductor

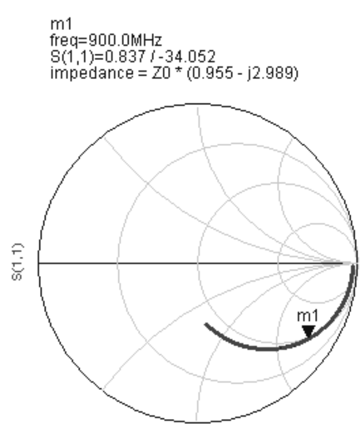
a. Select the **inductor** and **start the tuning mode**.



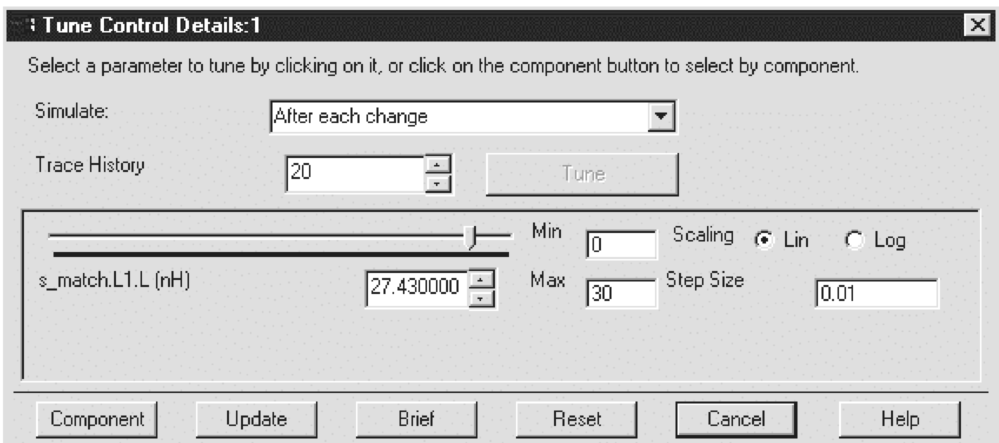
b. After the tuning dialog and status appear, open and position a **new data display window** near the tune control so you can see them both – move the schematic aside if necessary. Notice that the default dataset name **s_match** will appear (same as the schematic). Insert a Smith chart with S11 data and put a marker at 900 MHz. Notice that the S-11 trace is now changed with the real values of C and L.

c. Now, set the tune control to **slider** mode and move the slider back and forth between the ends. Notice that the value of S-11 changes very little because the range of inductance is too narrow.

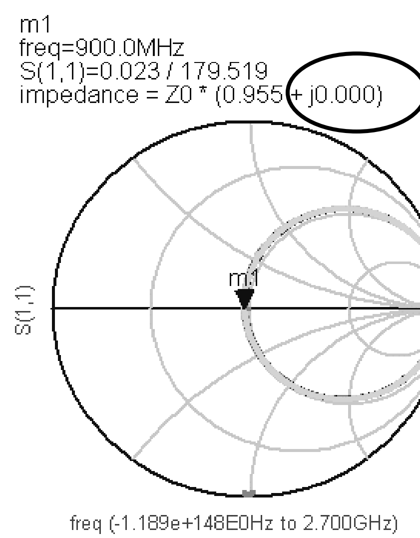




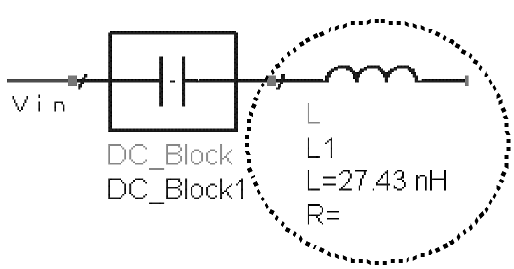
d. Increase the tuning range: click the **Details** button and the more detailed tune control appears. Increase the range from 0 to **30** by typing over the existing value. Based on the imaginary part of the impedance (-j3.1), the conjugate value of inductance of **30 nH** is close enough. Also, set the resolution Step Size to step to something small such as **0.1** or **0.01** and increase Trace History to **20**.



e. You should now be able to carefully move the slider and click the step buttons until you reach the impedance of **j0.000** as shown by the marker on the last trace. You can use this technique for determining the sensitivity of any component.



f. Click the **Update** button on the tune control and the value of L will appear on the component:



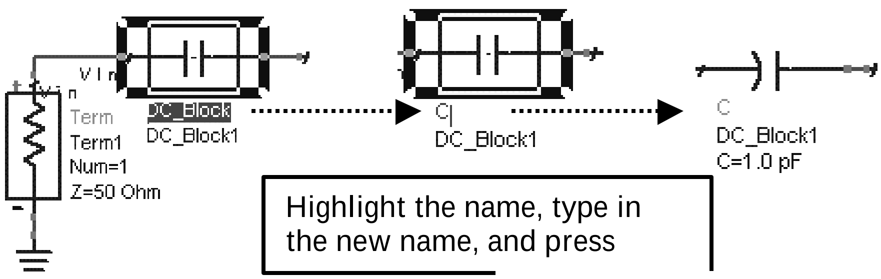
g. Save the data display as **s_match**.

3. Build a new input matching network (new configuration)

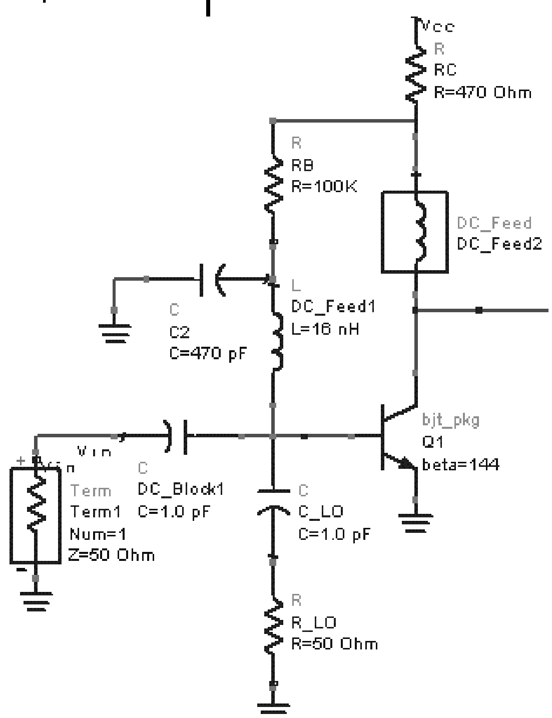
CIRCUIT DESIGN NOTE: At this point, the addition of the series inductor is only a first approximation. The remaining ideal components (DC feeds and blocks) must be replaced by realistic values and this may require a completely different topology other than just adding a series inductance. Also, a shunt capacitor needs to be added to the input to remove the IF signal that may appear there. Therefore, instead of continuing to add components in an attempt to create a match, you will use the following configuration that will solve all the matching problems for the input. This will speed up the lab exercise.

a. On the input, remove the series inductor you just tuned. It will be replaced by a network which will achieve the desired RF match and also provide the filtering.

b. Change the DC_Blocker to a real capacitor by **highlighting the component name** (see drawing - DC_Block) and typing in the new component name **C** and pressing Enter on the keyboard. The DC Block will automatically become a lumped capacitor:

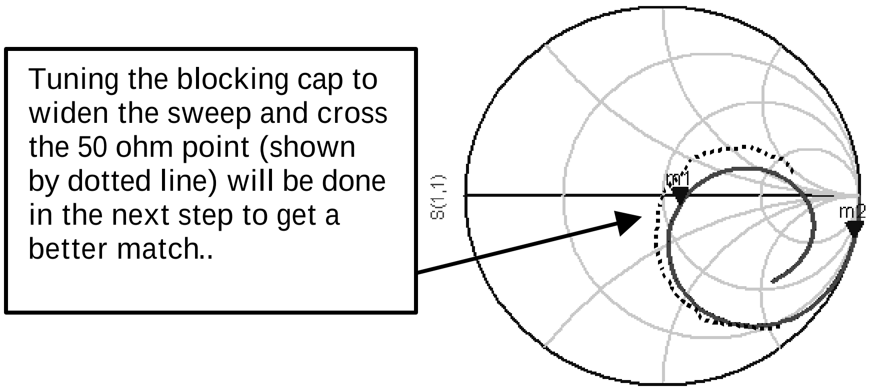


c. Continue modifying the input topology: Insert **C=470 pF** to shunt the IF (470 pF is a short to 45 MHz). Also, change the DC_Feed1 to **L=16 nH** to allow the dc to flow but it will block (choke) the RF. Lastly, be sure the **Z-ports** have been removed**.**

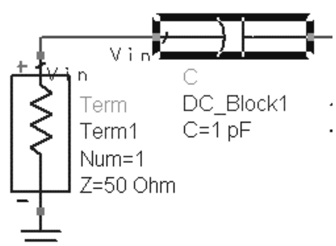


d. **Simulate** the new input network with a new dataset name: **s_match_in**.

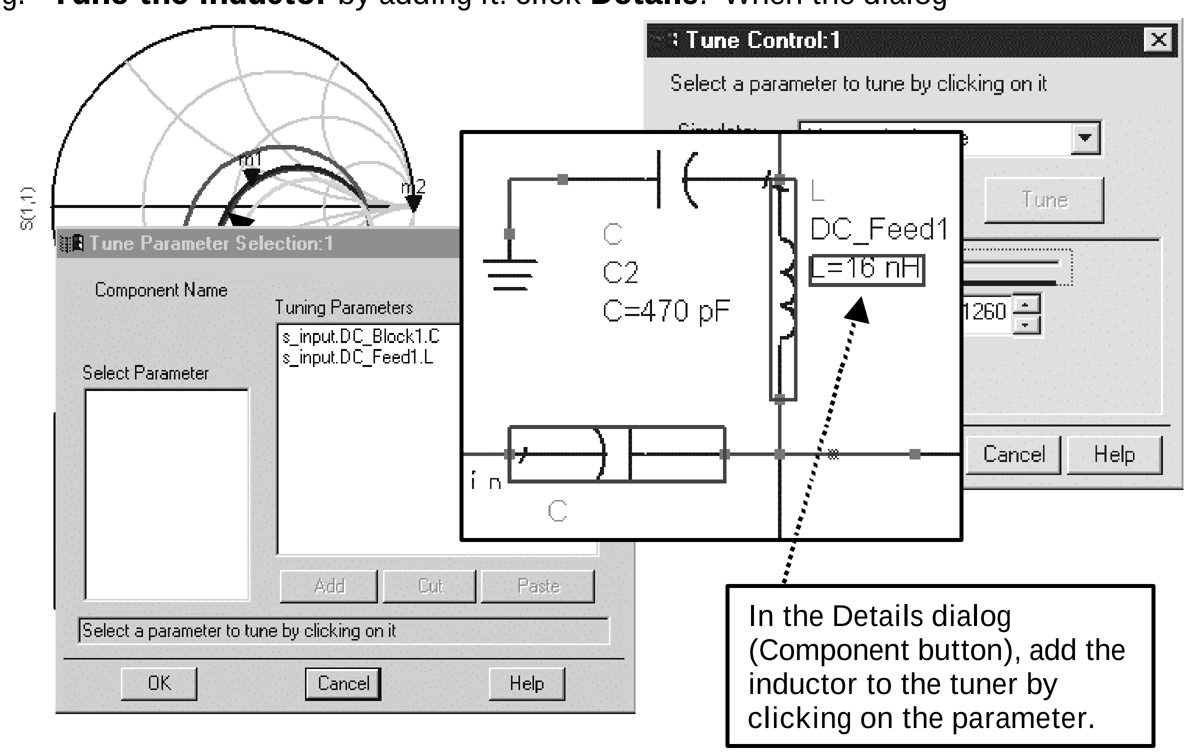
e. Plot the results and you should see a response like the one shown here where marker 1 is at the RF and marker 2 is the IF (almost an open). However, the response can be more finely tuned (next steps) so that the trace crosses directly through the 50 ohm point.



f. **Select the blocking capacitor** and start **tune** mode. Adjust the value of capacitance until the trace cuts though the center of the Smith chart. The next step will be done to adjust the inductor so that 900 MHz is directly in the center.

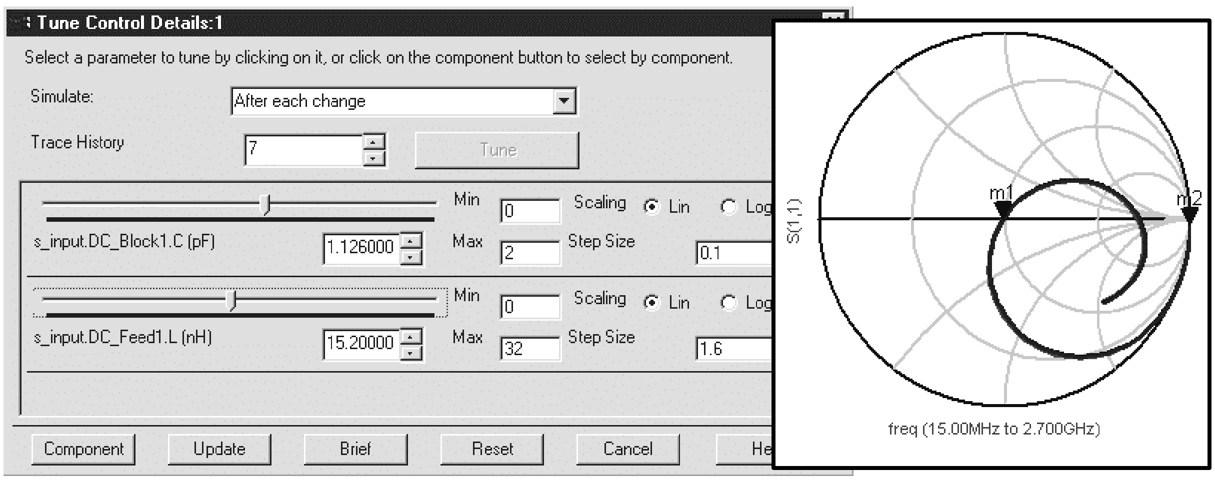


g. **Tune the inductor** by adding it: click **Details**. When the dialog



appears, select the **Component Button** and add the inductor by clicking on the parameter value (not the component) L=16 nH.

h. Adjust the inductance and you should get an almost perfect match at 900 MHz. In addition, the matching network is very efficient because it uses a minimum of components to block the dc, choke the RF, and shunt the unwanted IF frequency to ground. Click the **Update** button and the values will be updated on the schematic.

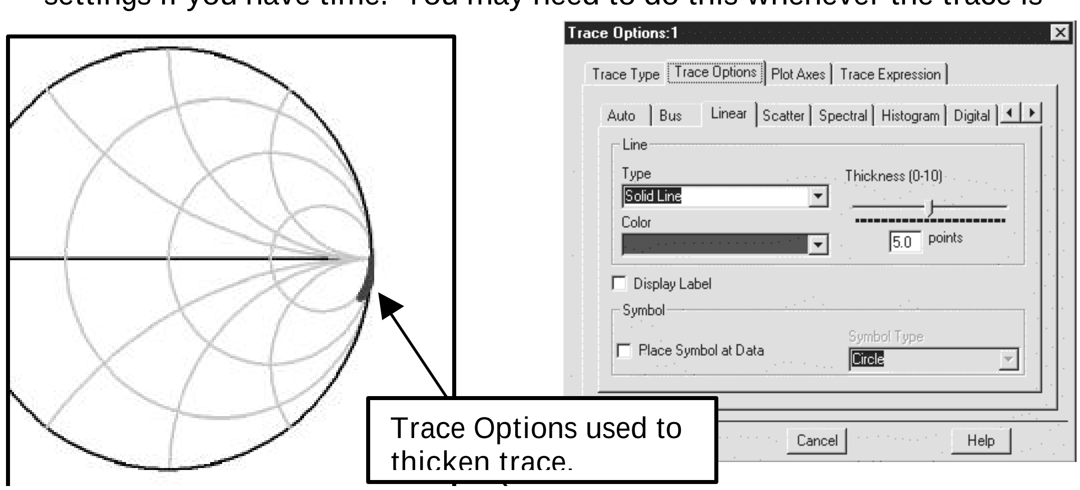


Design Note – L and C values: The tuned values of L and C will vary depending upon how finely you tune. However, C should be just about 1 pF and L should be between 15 and 16 nH for the following steps.

4. Examine the S-22 data

a. In the data display, insert a plot of S-22 from the last tuning simulation. You should see that S-22 is close to an open circuit over the frequency range.

b. Zoom into the trace area and double click on the trace. When **the Trace Options** dialog appears, thicken the trace and try using the other settings if you have time. You may need to do this whenever the trace is



difficult to see or when it is in a very narrow range. Build the output circuit.

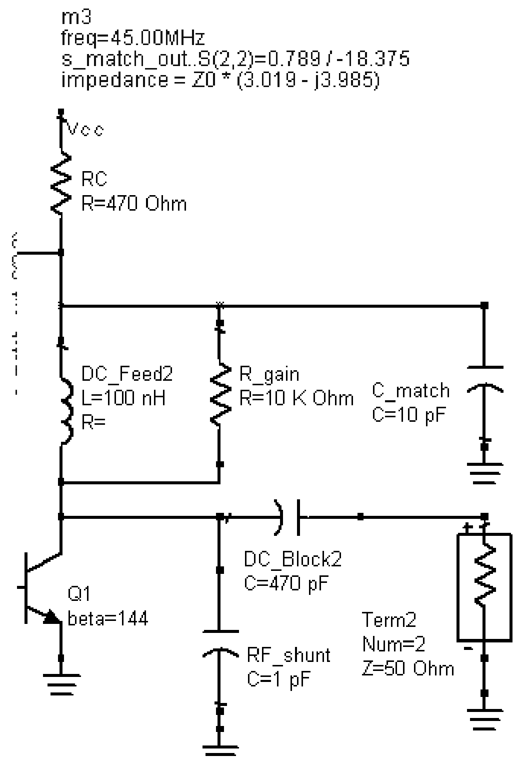
Output Match Design Note: For the next part of the lab exercise, you will use the optimizer to achieve the output match with a given topology.

5. Build the IF output matching network

Build the output to look like the one shown here. The DC feed is a 100 nH inductor in parallel with R_gain resistor (10K) which controls conversion gain. The capacitor (RF_shunt = 1 pF) will help short higher frequencies. Looking into the transistor from the 50 ohm load are two other capacitors for blocking (470 pF is a short to the IF) and C_match for matching.

6. Simulate and plot the S-22 results

Simulate (dataset name= s_match_out) and then note your results. The trace should be similar to the one shown here. S-22 at 45 MHz (shown by marker 3) is not matched to the characteristic impedance of 50 ohms. While you could use the tuner to try and achieve a match, the optimizer can also achieve the same goals.

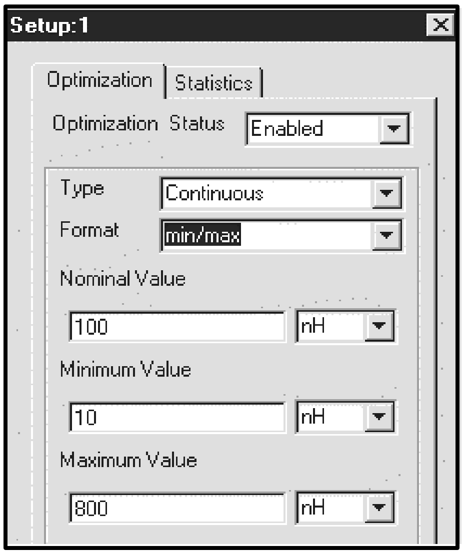


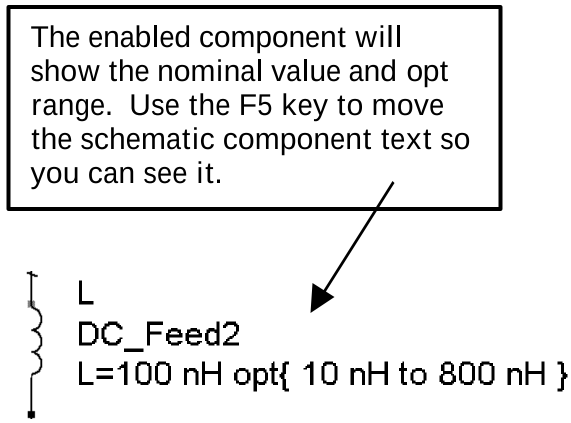
Optimization NOTE: The following steps show how to set up an optimization in three steps: 1) Enabling the components to be optimized, 2) Defining the Goals, and 3) setting up the Optimization control.

7. Enable the components to be optimized

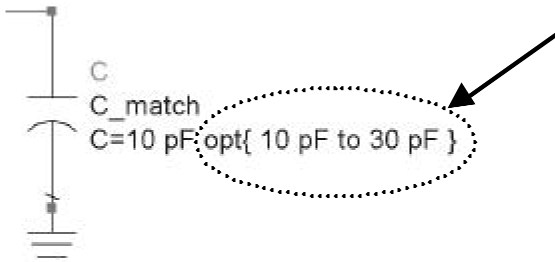
a. Edit (double click) **the DC_Feed2** inductor and click the Optimization/Statistics Setup button.

b. In the dialog, enable the dc feed inductor component for optimization, type, and range as shown. For this step, you will use Continuous optimization with min/max values: 10 to 800 nH. Click OK as needed.





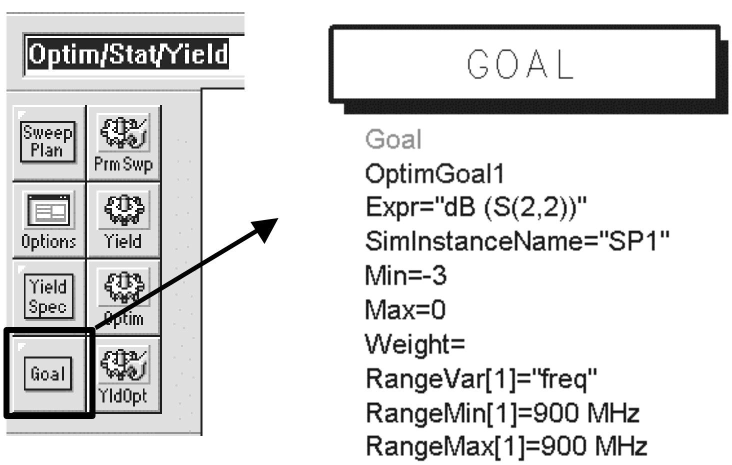
c. **Enable the C_match capacitor** for continuous min/max optimization also over the range of **10 to 30 pF**. Edit the component, using the dialog box to do this - after a component is enabled for optimization, the annotation will appear. Or, you can edit it directly on the screen by typing in the **opt** function and range as shown here.



Components can be enabled for optimization by on-screen editing using the **opt** function and the range in curly braces.

8. Define optimization goals

a. Insert the first optimization goal from the Optim/Stat/Yield palette. Goals are required (named) in the optimization component. Set up the goal as shown using the steps here:



NOTES: You can also edit the goal by double clicking on it.

The 900 MHz **range** is required by the simulator.

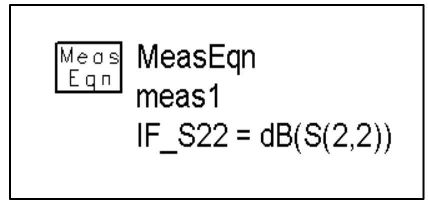
b. Enter the **Expr**, which is return loss: **"dB S(2,2))"**

c. Type in the SimInstanceName - the name of the S-parameter simulation controller: "**SP1"**.

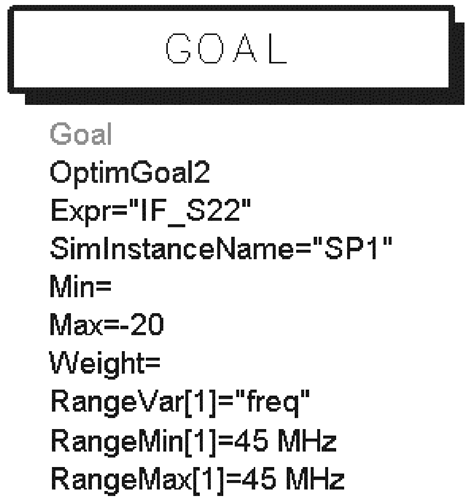
d. Type in the **Expr min/max range**: **–3 dB to 0 dB of return loss**

e. Type in the Range Variable: use the global variable **"freq"** and set the range which will be at one frequency: **900 MHz**.

f. **Insert a measurement equation to be used in the second goal.** Measurement equations are found in all simulation palettes. This goal will be available in the dataset. Type in the equation as shown where IF_S22 (or some name of your choice) will be the expression for achieving the IF return loss goal:



g. Insert the second **optimization goal** for the IF and type in the expression name as shown here. Enter the max goal value of –20. There is no need to set min or you can set it to –1000).

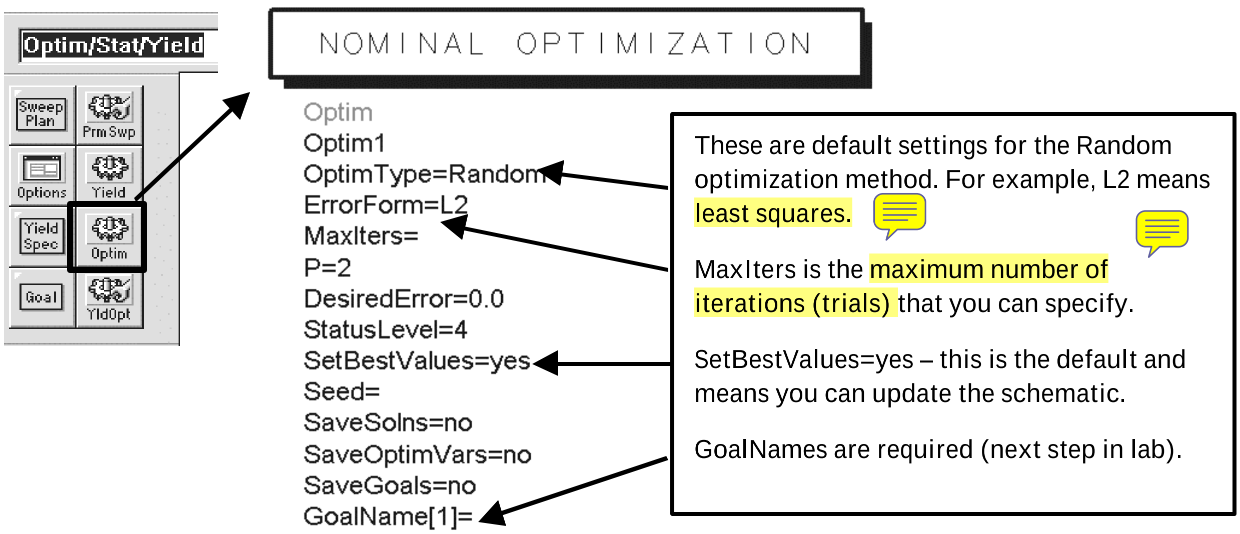


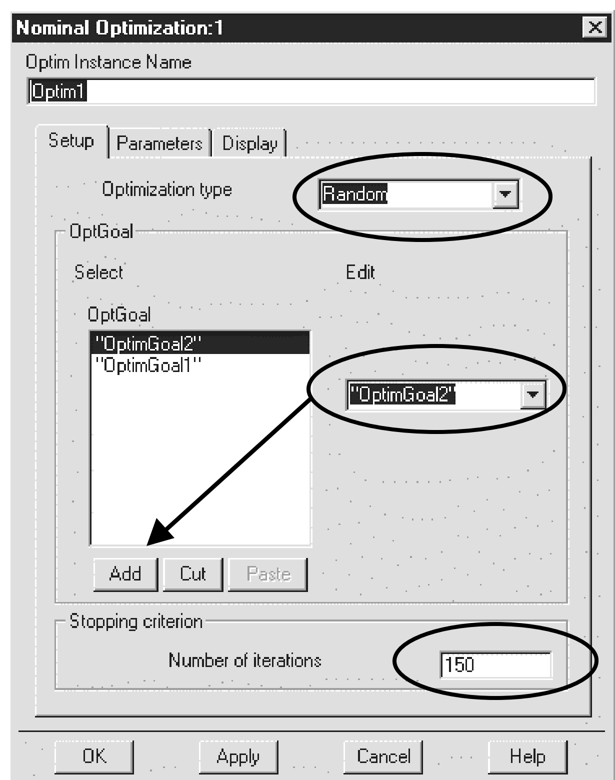
Review of Opt Goals: Goals must refer to the simulation controller name: "SP1" (similar to a parameter sweep). The ***expression*** usually refers to the measurement (data in array form). By specifying a min and max range for the expression, you are specifying what goal you want to achieve. Here, the goal is to have an IF match of at least -20 dB (no min is required) and an RF match between 0 and -3 db. In simple terms, you want a good match at 45 MHz at the output and a bad match on the output at 900 MHz.

9. Set up the Optimization control

The optimization component controls the simulation by receiving data and testing the data until the goals are reached or the maximum number of iterations has expired.

a. Select **Optim/Stat/Yield** in the schematic window palette and insert the Nominal Optimization controller (Optim).



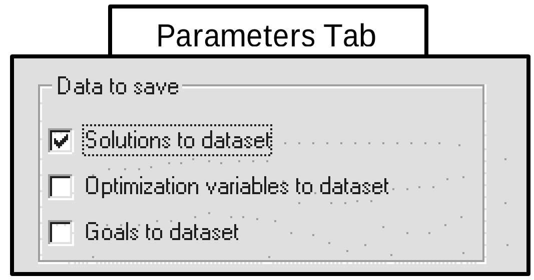


b. Edit (double click) the Optimizer control cmponent and add the two goals (**OptGoal**) by clicking their names. If you do not select specific goals, the default is to run all the goals.

c. Be sure to select and use **Random** optimization (most common).

d. Use **150** iterations. For Random optimization, one iteration is a successful simulation and may or may not get closer to the goal.

e. In the Parameters tab, check the box for Solutions to dataset. This will put the S parameters in the dataset. Also, always be sure the **Set best values...** box is checked (yes on display). This allows the optimized component values to be updated on the schematic.



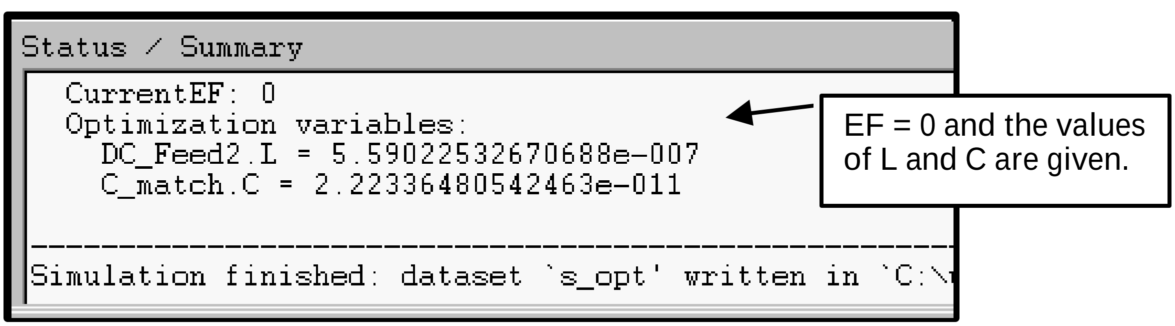
Parameters Tab Note: The ***Data to save*** selections can create large datasets that you may not need. To avoid this, do not check any boxes and, if you achieve the goal (EF=0), update the component values, deactivate the optimizer and do a regular simulation. However, for this lab, you will use the Solutions to dataset.

f. In the **Display** tab, set only the things you want to be displayed – this is a good practice for keeping organized schematics and simulations.

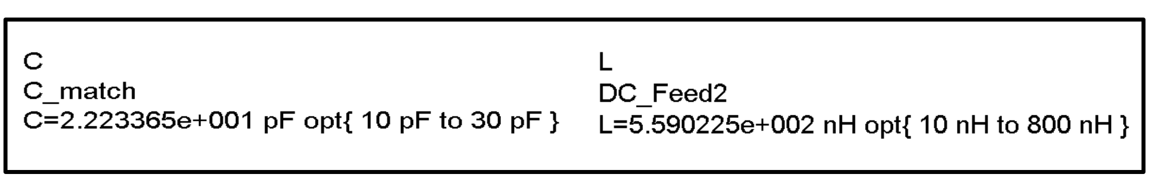
10. Optimize

a. Use a new dataset name (such as **s_opt**) and **Simulate (F7)** with the simulation set 15 MHz to 2 GHz with 5 MHz steps to land on RF and IF.

b. Watch the **Status Window** for the results of the optimization. Use the scroll bar if necessary to read it. If the optimization is successful, you should see a message that the EF (error function) = 0. If not, check your work, or try another type such as Gradient, or adjust the ranges.

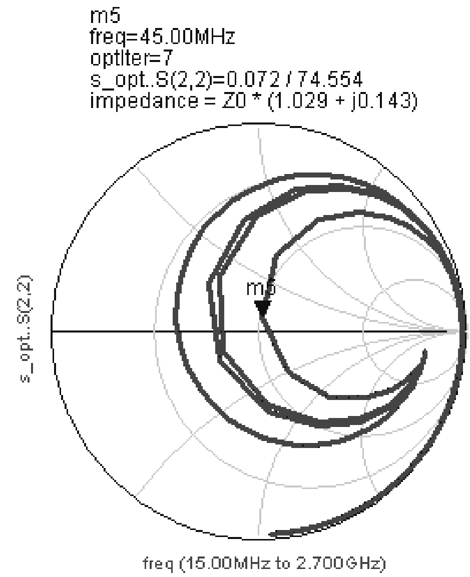


c. If the EF is 0, go to the schematic and click **Simulate > Update Optimization Values**. The optimized values of L and C will appear as exact values but you can round them off. Here, C is about 22 pF and L is about 560 nH (your answer may vary slightly).



11. Plot the S22 data.

It will be similar to the plot shown here where all the successful iterations are traced. Notice that one of the traces is near the center of the Smith Chart (marker). That trace represents the last optimization iteration where the goals were met.

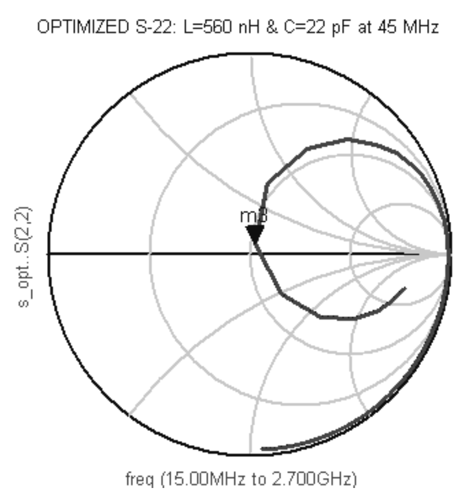


12. List the *meas eqn* data

a. Insert a list of your equation: **IF_S22** that was used in the goal. The equation will be in the same dataset as the S-parameters (s_opt). You should see the value of the equation at 45 MHz which represents the optimized goal.

freq	s_opt..IF_S22
15.00MHz	-2.096
20.00MHz	-2.493
25.00MHz	-3.176
30.00MHz	-4.417
35.00MHz	-6.931
40.00MHz	-13.368
45.00MHz	-20.102
50.00MHz	-7.794
55.00MHz	-3.972
60.00MHz	-2.274
65.00MHz	-1.429
70.00MHz	-0.970

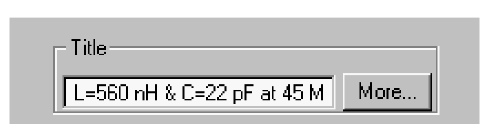
Your measurement equation: IF_S22 = dB (S(2,2)), from the schematic is shown for the 45 MHz IF as reaching –20 dB of return loss using the optimized values of L and C.



b. **Deactivate** the Optimizer and edit the component values on screen by highlighting and deleting the unwanted values and typing in the values of L and C as: **L = 560 nH and C = 22 pF**.

C=22 pF ~~opt{ 10 pF to 30 pF }~~

c. **Simulate** and your plot of S-22 will now have only one trace similar to the one shown here. Also, edit the plot and use the **Plot Options** to title the plot.



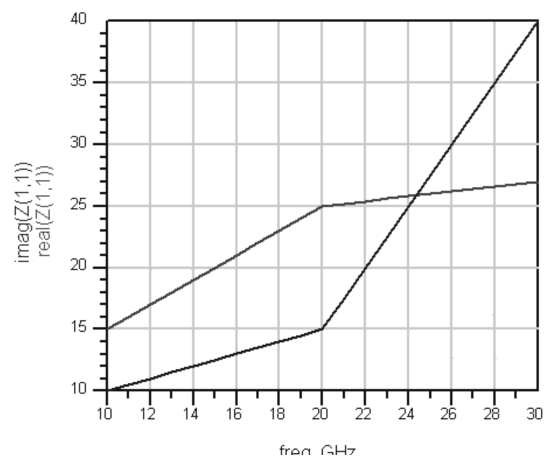
At this point the mixer has good input and output matching networks. Of course, you could refine the output match with the tuner but it is not necessary.

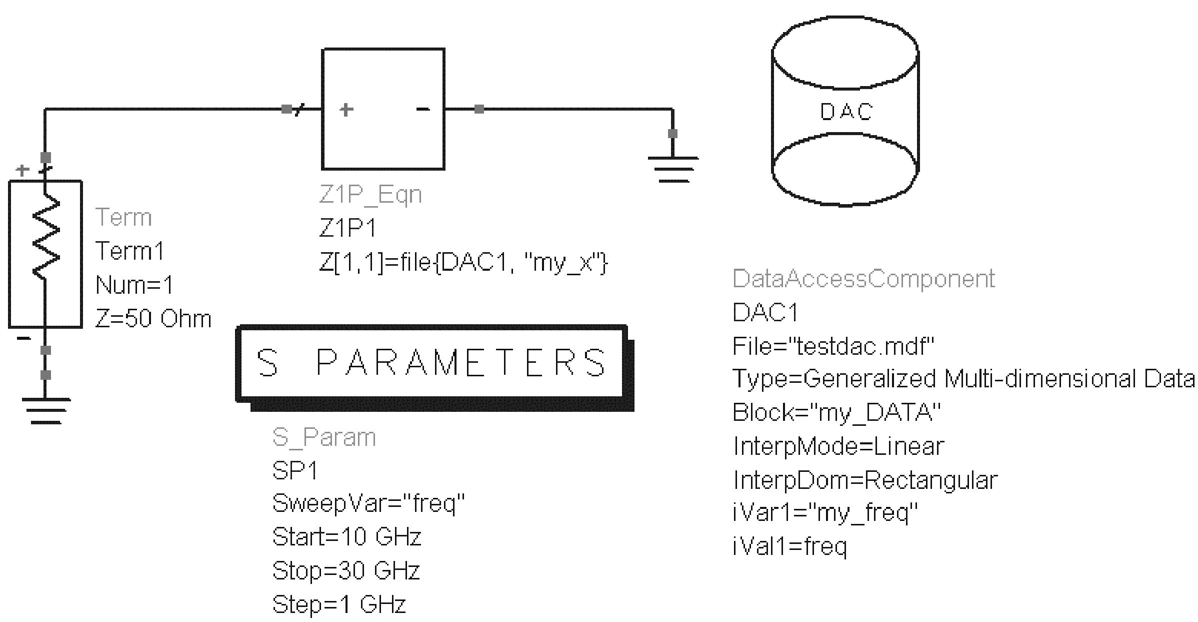
NOTE on the *opt* and *noopt* function: Refer to the schematic where the optimized component value had annotation such as: C=7.95462189+001 pF opt{ range]. If you type noopt instead of opt, that component (noopt) will not be optimized. This is easier than editing the component in the dialog box.

EXTRA EXERCISES:

1. Optimize again using gradient method instead of random or try to optimize to better goals: S-22 = -25 or better dB at IF. To do this, try using another optimization type such as genetic.

2. Try using a DAC component to create a frequency sensitive inductor. As the plot here shows, the real and imaginary values change with frequency. These curves are described by a file which is read by the DAC. To do this, you need to write a file for the data and build the schematic required schematic. Step by step instructions follow on the next page…



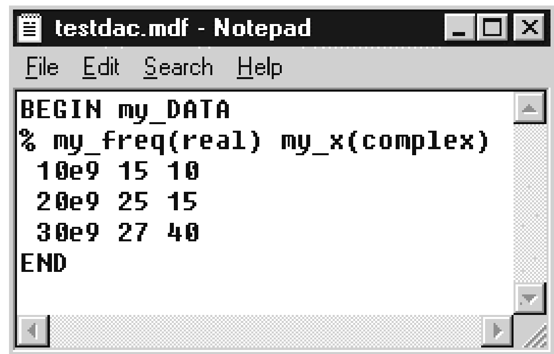


DAC instructions:

a. Open a new schematic saved it as DAC_Z. Refer to the previous circuit and insert the components in their default state:

- S-parameter controller, Termination and ground, Z1P from the equation based linear palette, and a DAC from the Data items palette.

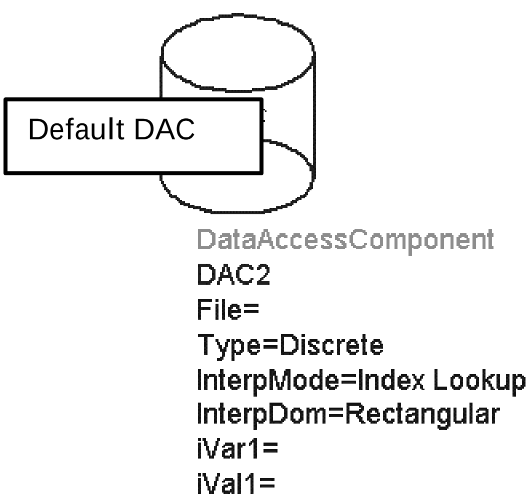
b. Write an **mdf** file using the ADS main window Options > Text Editor (use only Note pad not Word pad which has formatting - this is a must). Write the file shown here and save it in the DATA directory as: **testdac.mdf**. If necessary, you may need to use the windows file explorer to change the name if it is saved as a .txt file. Also, be careful of the syntax in the file - the first column contains 3 frequency points, the second and third columns contains the real and imaginary parts of the reactive component.

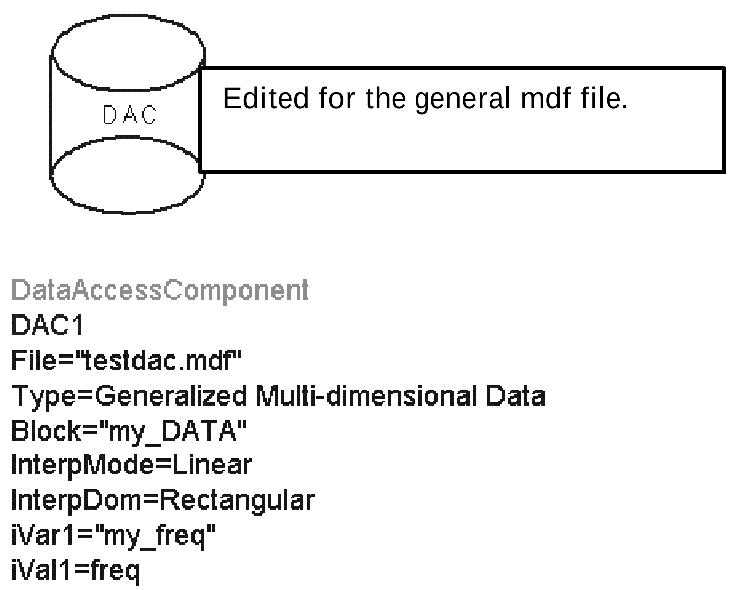


c. On schematic, edit the S-parameter controller. In **Parameters tab**, set to compute **Z parameters** not S. In the **Display ta**b, check the the Sweep Var and start, stop, set and set them as shown to sweep the global variable **"freq"** from 10 to 30 GHz in 1 GHz steps. You will get interpolated data for all the steps.

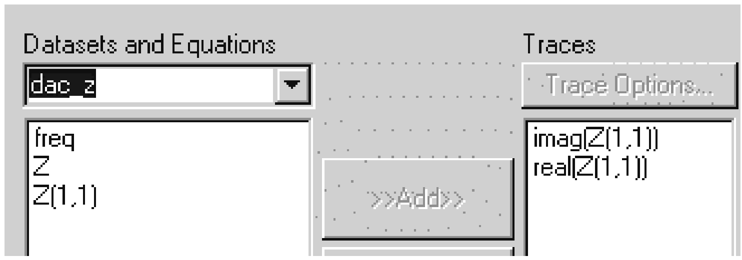
d. On schematic, set the Z1P value of **Z[1,1]= file{DAC1,"my_x"}**. The value of Z11 is the variable "my_x" in the DAC1 file. Of course, the file is testdac.mdf.

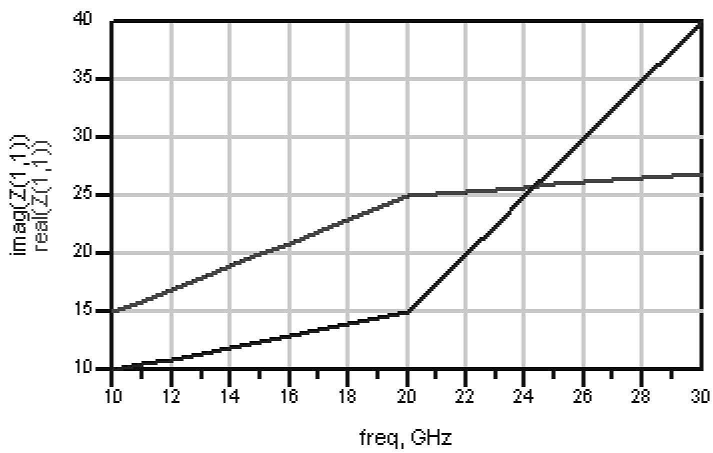
e. On schematic, edit the DAC as shown here. IVar1 is the independent variable and iVal1 is the swept variable. As "freq" is swept, "my_freq" will be indexed and the DAC will return complex values of "my_x" interpolated over the frequency range.





f. Check the circuit and simulate. Then plot two traces, real and imag, of Z(1,1) as shown where X changes with frequency. Now, the Zport can be used wherever a frequency sensitive component is required. For multiple components, simply create different files and access them as required.





THIS PAGE IS INTENTIONALLY BLANK

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

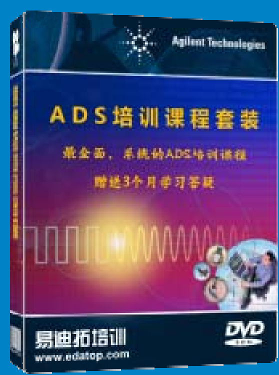
该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

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