ADS Fundamentals - 2009

LAB 5: S-parameter Simulation, Matching and Optimization

Overview - This exercise continues the amp_1900 design. It teaches how to setup, run, optimize and plot the results of various S-parameter simulations. Also, the optimizer is used to create the impedance matching networks.

OBJECTIVES

- Measure gain and impedance.
- Set up and use sweep plans, parameter sweeps, and equation based impedance.
- Calculate values for a matching network.
- Design a matching network.
- Use optimization to meet design goals.
- Use Noise and Gain circles.
- Write a file with the Data File Tool.

Table of Contents

1. Set up the simulation and circuit with ideal components. 3
2. Simulate and plot data with marker readout modifications. 4
3. Write an equation to vary the Term port impedance. 4
4. Calculate L and C values in the data display. 5
5. Replace L and C with calculated values and simulate. 6
6. Use the Smith Chart utility to build a simple matching network. 7
7. Add output matching components. 11
8. Set up an Optimization controller and Goals. 12
9. Enable the components to be optimized. 14
10. Plot the results. 16
11. Update optimized values and disable the *opt* function. 17
12. Simulate the final matched circuit. 18
13. Stability equations with gain and noise circles. 19
14. OPTIONAL – Read and Write S-parameter Data with an S2P file 21
15. OPTIONAL – YIELD analysis 23

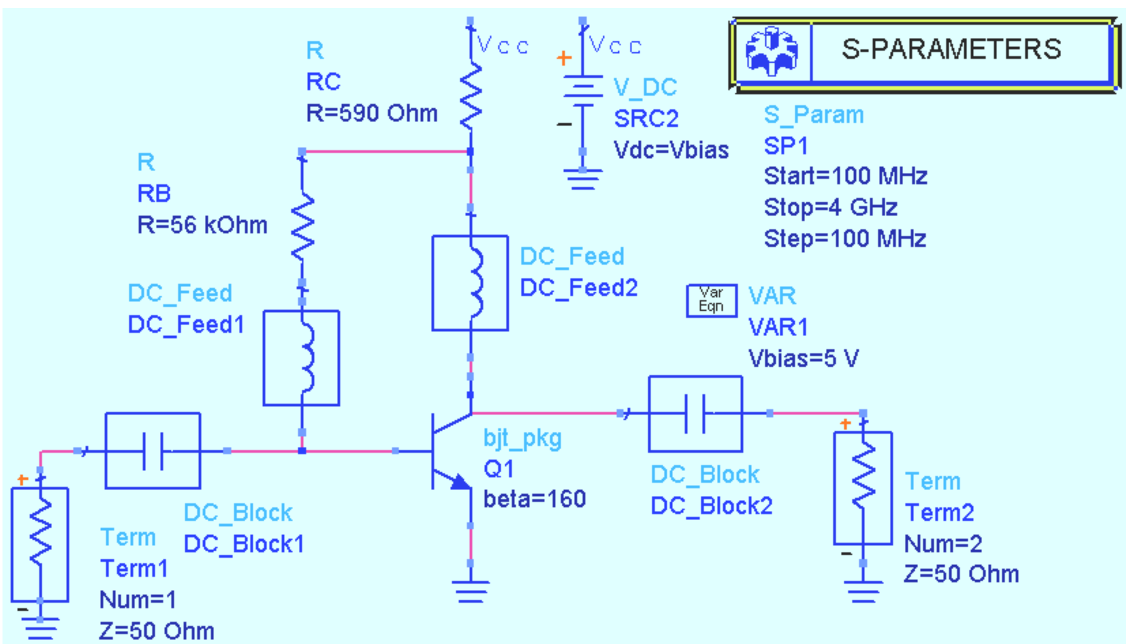
PROCEDURE

1. Set up the simulation and circuit with ideal components.

a. Save the last schematic (ac_sim) design as: **s_params**.

b. Modify the design to match the schematic shown here:

- Delete the AC source and controller. Also delete the measurement equations, parameters sweep, and any unwanted variables, etc.
- Insert terminations (**Term**) from the S-parameter palette.
- From the Lumped Components palette, insert two ideal inductors: **DC_Feed** to keep the RF out of the DC path
- Insert two ideal **DC block** capacitors.
- Delete the node names by click the Name icon and, leaving it blank, clicking on the node names (Vin and Vout). For S-parameter simulation, the port terminations (num1 and num2) provide nodes.



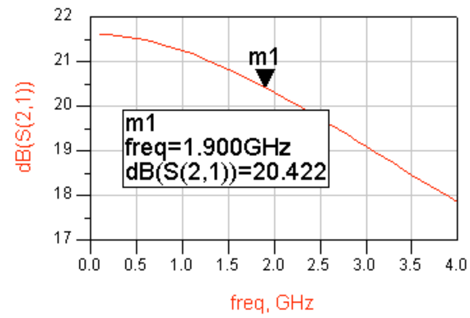
c. Insert an **S-Parameter** simulation controller and set: **Start=100 MHz**, **Stop=4 GHz**, and **Step=100 MHz**.

d. **Save** the design.

2. Simulate and plot data with marker readout modifications.

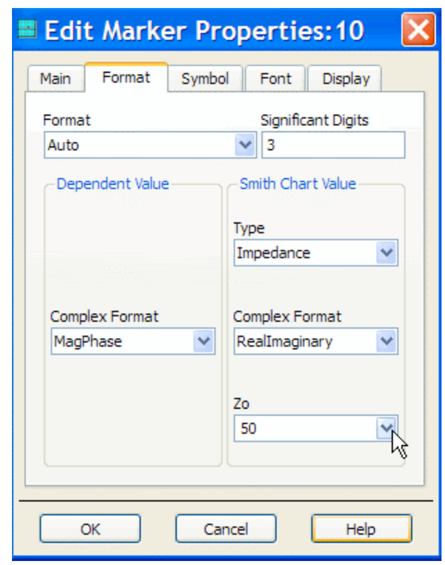
a. Be sure the name of the dataset is: **s_params** and then **simulate**.

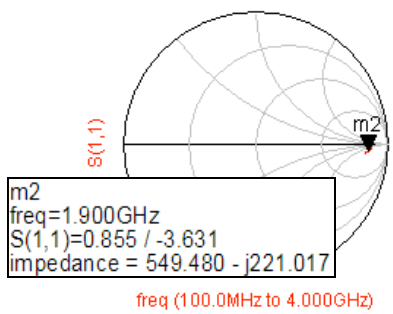
b. When the simulation is finished, insert a rectangular plot of **S21 (dB)**. Insert a marker on 1900 MHz and verify that the gain is about 20 dB.



c. Insert a Smith chart of **S11**and place a marker on 1900 MHz. To move the marker, select the readout and use the arrow keys.

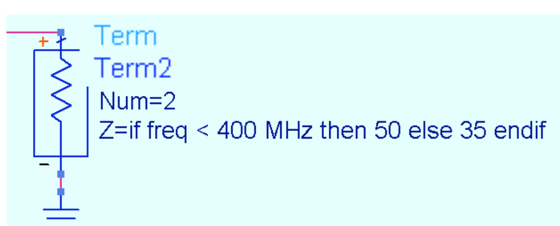
d. Edit the **marker readout** (double click). Go to the the **Format** tab and change **Zo** to **50** as shown. Click Click OK and the marker will now read the value in value in ohms, referenced to 50 ohms.





3. Write an equation to vary the Term port impedance.

a. In schematic, **write an equation** for port 2 Term Z to be 35 ohms above 400 MHz: **Z = if freq < 400 MHz then 50 else 35 endif.**



Type directly on-screen.

b. **Simulate** and then insert a **list** of **PortZ(2)**. Verify that Z is 35 Ohms above 400 MHz.

freq	PortZ(2)
100.0MHz	50.000 / 0.000
200.0MHz	50.000 / 0.000
300.0MHz	50.000 / 0.000
400.0MHz	35.000 / 0.000
500.0MHz	35.000 / 0.000
600.0MHz	35.000 / 0.000
700.0MHz	35.000 / 0.000
800.0MHz	35.000 / 0.000
900.0MHz	35.000 / 0.000
1.000GHz	35.000 / 0.000

c. Reset the value of port 2 Term to 50 ohms: **Z = 50 Ohm**.

4. Calculate L and C values in the data display.

The transmission and reflection characteristics of the biased circuit show about 20 dB of gain but with a mismatch to 50 ohms at the input. Also, the DC feeds and blocks are ideal and need to be replaced with real values.

a. In data display, write an equation, **XC,** for the capacitive reactance of **10 pF** at 1900 MHz. Then **list** equation XC as shown here. If desired, title the list using **Plot Options**. With this low reactance, 10pF will be the blocking capacitor values.

Eqn XC = -1 / (2*pi * 1900M * 10e-12)

Reactance of Cap value:

XC
-8.377

b. Change the value of the capacitor in the equation and verify that XC is automatically updated in the list.

c. Create a table for a range of inductor values and reactances. **L_val** is a range of swept values from 1 nano to 200 nano in 10 nano steps. In ADS, the syntax of two colons is a wild card (all values) and can also be used to indicate a range as shown here. The square brackets are used to generate the sweep. After writing the equations and listing them as shown here, scroll through the list. As the inductor value increases, the reactance at 1.9GHz increases. Therefore, a value of 120 nH should be enough for the DC feed (RF choke).

NOTE: The XL equation will be red (invalid) until L_val is written.

Eqn XL= 2 * pi * 1900M * L_val

Eqn L_val = [1n :: 10n :: 200n]

XL	L_val
11.938	1.000E-9
131.319	1.100E-8
250.699	2.100E-8
370.080	3.100E-8
489.460	4.100E-8
608.841	5.100E-8
728.221	6.100E-8
847.602	7.100E-8

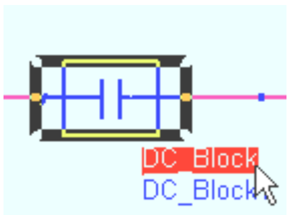
NOTE on equations and tables - You can copy the equations and tables (Ctrl C / Ctrl V) to other data displays. Or use the command *File > Save As Template* to save the data display as a template that can be inserted in other projects.

d. **Save** the current data display and the schematic.

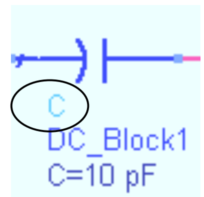
5. Replace L and C with calculated values and simulate.

a. Save the schematic with a new name: **s_match**.

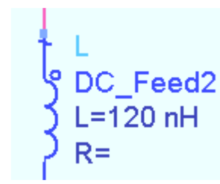
b. Change the component name (DC_Block) of both blocking capacitors to **C** and they will automatically become lumped capacitors as shown here. Assign the value for each **C = 10 pF**.



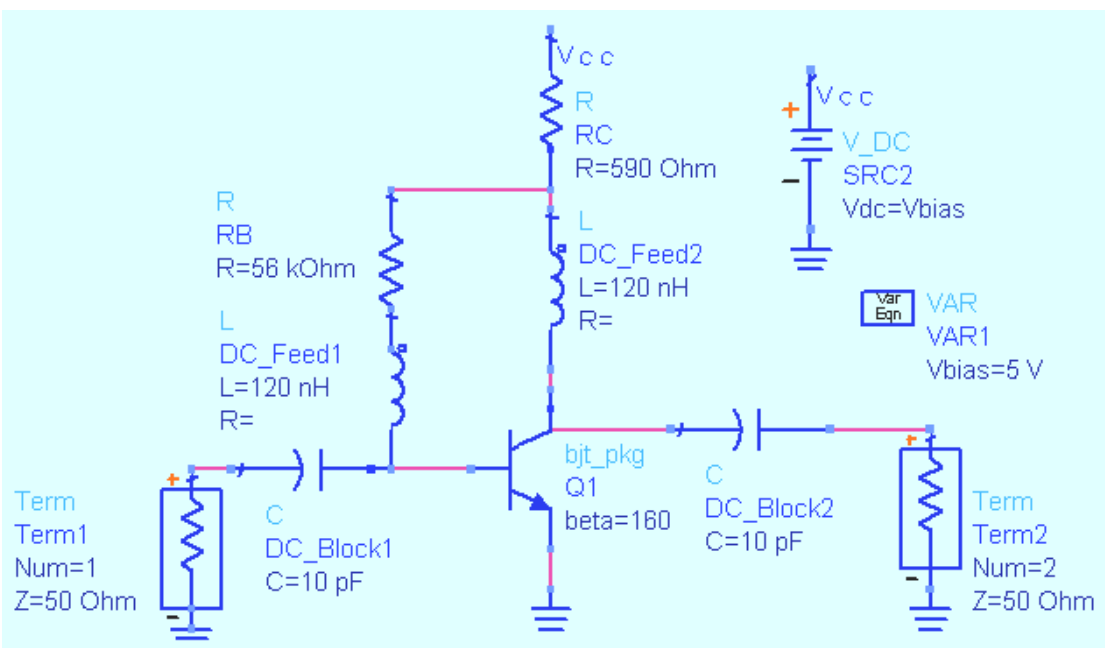
Highlight the component name, type in C, and press Enter: DC_Block will become C. Then change to 10 pF.



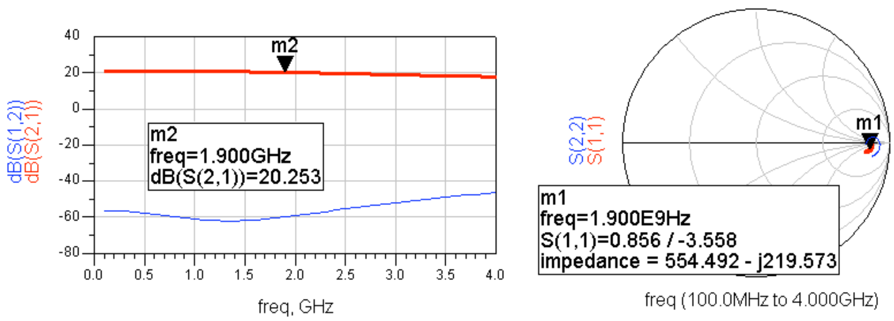
c. Change the ideal inductors (DC_Feed) in the same manner and set **L = 120 nH** for each. According to the XL and L_val table, the reactance at 1900 MHz is about 1.5K, which is reasonable at this point in the design.



d. The schematic should now look like the one shown here. Check your values and then **Simulate**.



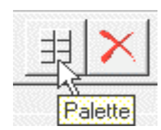
e. In the data display, plot the transmission (S12 and S21) and reflection (S11 and S22) data with markers as shown here. Notice the gain stays relatively flat, the leakage is reasonable, but the impedance is not near 50 ohms. The next step is to create an input matching network.



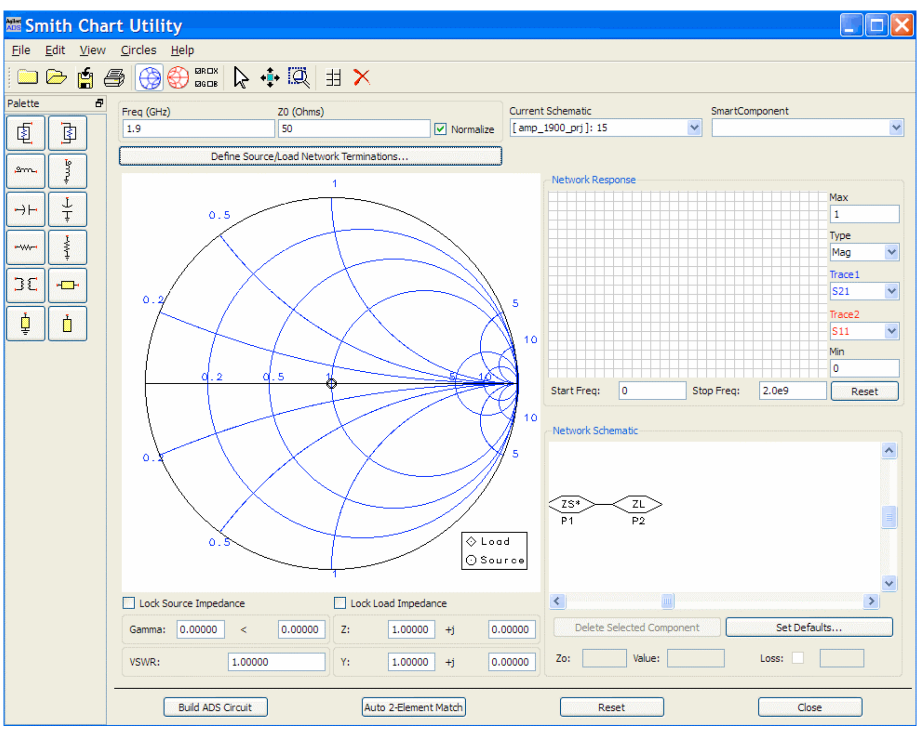
6. Use the Smith Chart utility to build a simple matching network.

a. In the current schematic, click on the commands: **Tools > Smith Chart** (this is the same as DesignGuide > Filter and then selecting the Smith Chart Control window).

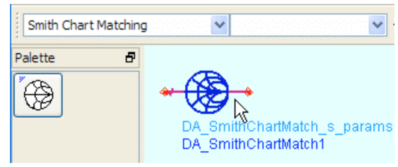
b. Click the **Palette** icon shown here - this adds the Smith Chart palette with the Smith Chart icon to your schematic.



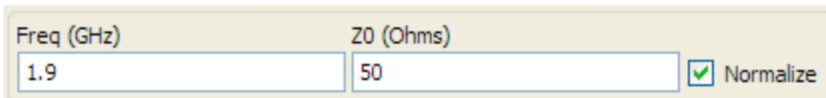
Smith Chart Control Window



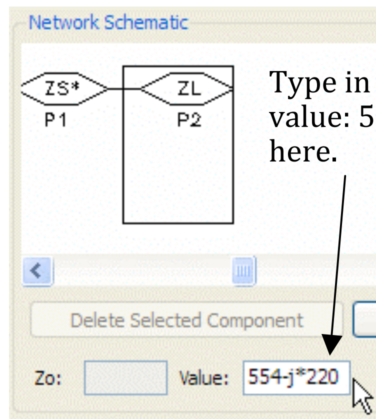
c. In the schematic (s_match), insert the Smith Chart Matching Network component (also known as a Smart Smart Component) near the input of the amplifier – no – no need to connect it – but it is required. Also, click OK click OK when a message dialog appears.



d. Go back to the Smith Chart control window and type in the Freq (GHz) to 1.9 as shown here.

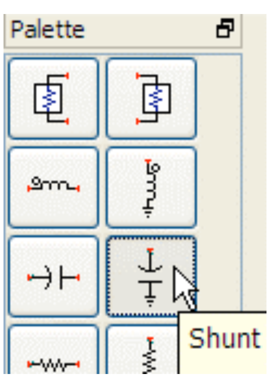


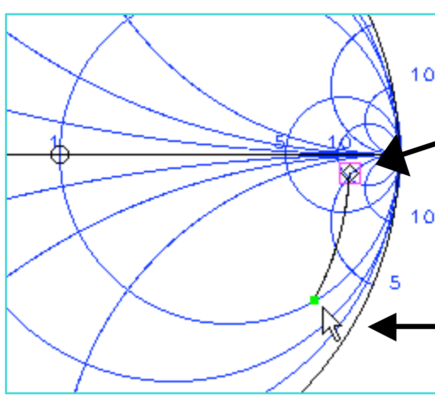
e. In the lower right corner of the Smith Chart Chart utility window, select the ZL component and type in the impedance impedance looking into the amplifier from from the last simulation: 554-j*220 as as shown here and click Enter.



Type in the Z value: 554-j*220 here.

f. Notice that the load symbol on the Smith chart has relocated as shown here. Next, select the **shunt capacitor** from the palette and move the cursor on the Smith chart: when you get to the 50 Ohm circle of constant resistance, click to stop, as shown here (it does not have to be exact for this exercise).



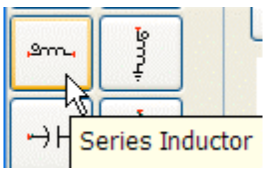


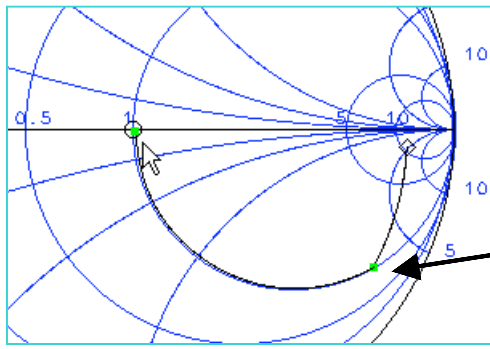
Load symbol is at 554-j220. Source symbol is at 50+j0.

Shunt cap allows you to move to the 50 ohm circle.

g. Next, select the **series inductor** and move the cursor along the circle until you reach the center of the Smith chart and then click.

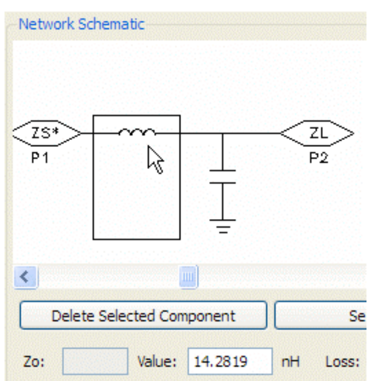
Now you have a 50 Ohms match between the load and source.

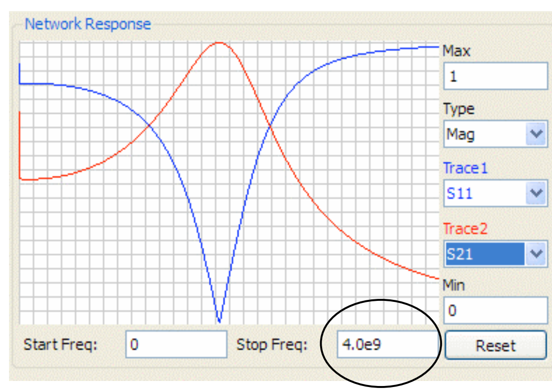




Series L allows you to move along the 50 ohm circle to the center of the Smith Chart.

h. Move the cursor into the lower right corner of the window and click on each of the components in the Schematic as shown here. You will see the values for the inductor and capacitor: approximately L= 14 nH and C = 400 fF or 0.4 pF.

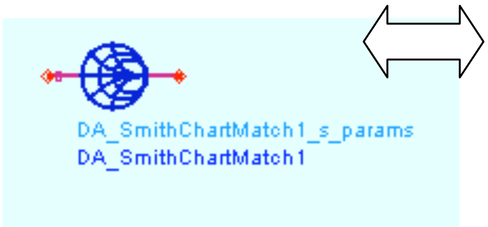


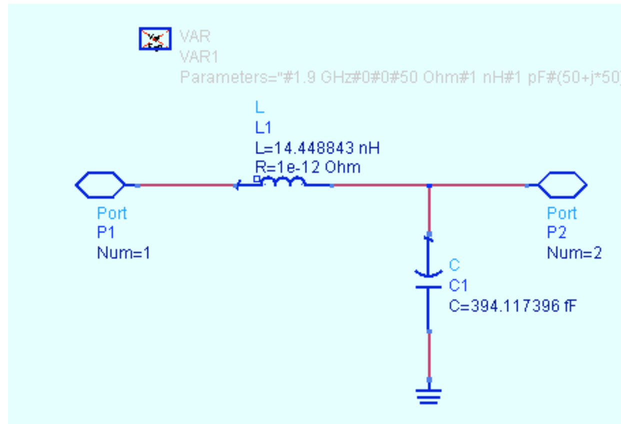


i. To clearly see the response of this network, change the **Stop Freq** to **4 GHz** (4.0e9) and you will see the null (S11) at 1900 MHz. Also, set Trace 2 to S21 to see both reflection and transmission.

j. To have the DesignGuide build the circuit, click the button button on the bottom of the window: Build ADS Circuit. Circuit. Click OK to any messages that appear.

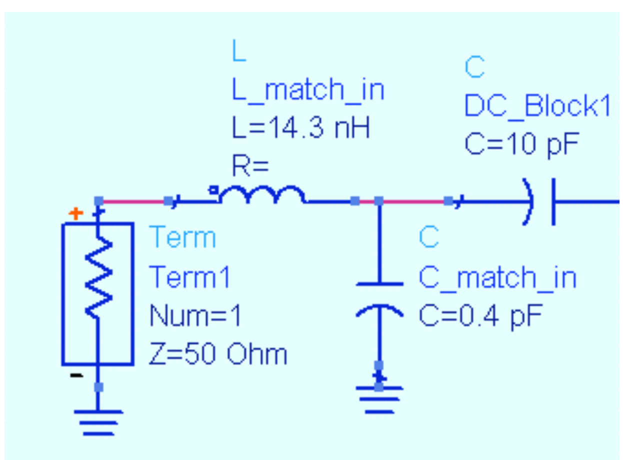
k. On the schematic, push into the Smith Chart component and you should see the network similar to the one as shown here. You values may be slightly different which is OK. Pop out when finished.





Now it is time to use the matching network with the amplifier. You could use the component by connecting it to the amplifier input. However, because you will be using the optimizer, it is better to have the L-C components on the schematic.

l. Either copy/paste the L-C and ground components onto the amplifier or simply insert an L and C on your schematic. Then set the values to **L=14.3 nH** and **C=0.4 pF**. The amplifier input should now be as shown here.

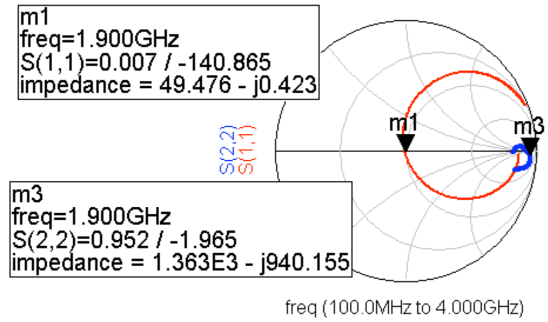


m. **Delete** the Smith Chart component from the schematic and **close** the Smith Chart utility window.

n. Simulate with the input network.

o. Set the simulation step size to **10 MHz**, and **simulate**.

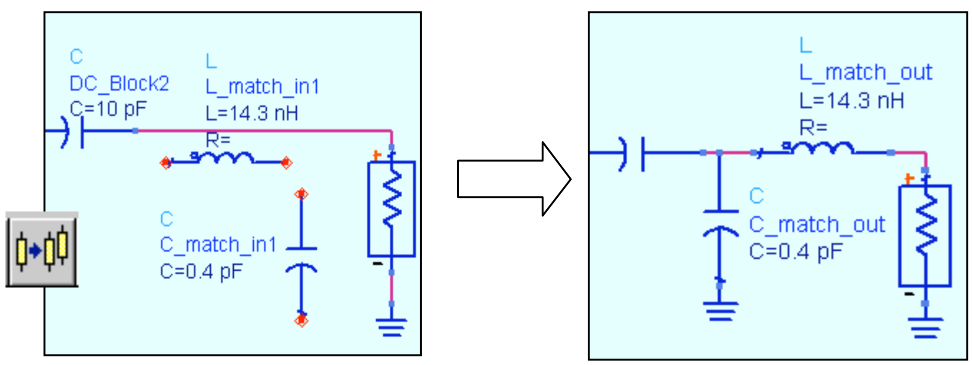
p. When the simulation is complete, add S-22 to the Smith chart. Place a marker on S-22 at 1.9 GHz. The results show S-11 is good but S-22 is not matched as shown here.



7. Add output matching components.

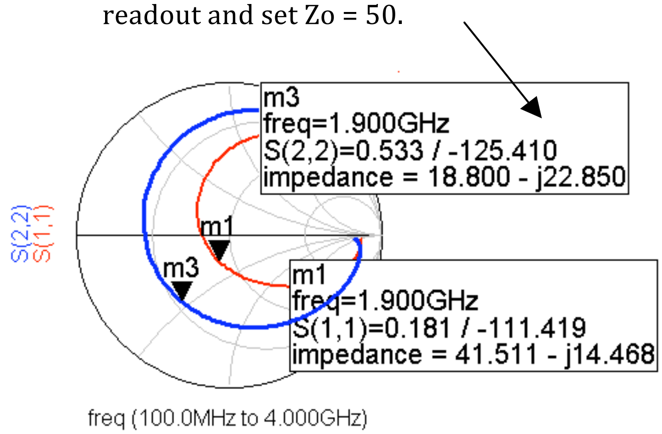
Because S-22 is similar to the unmatched impedance of the input, it is reasonable to put a similar topology on the output and simulate the response.

a. Select the input L-C network and use the **copy icon** (shown here) to make a copy to the components. Then place them near the output, with a ground on the capacitor. Delete the wires and insert them as shown at the output.



b. **Simulate** and check the response. Your data should be similar to the results shown here where S22 is now closer to 50 ohms. However, S11 has shifted, as you should expect. Set the S-22 marker readout to Zo = 50.

Double click to edit the marker readout and set Zo = 50.



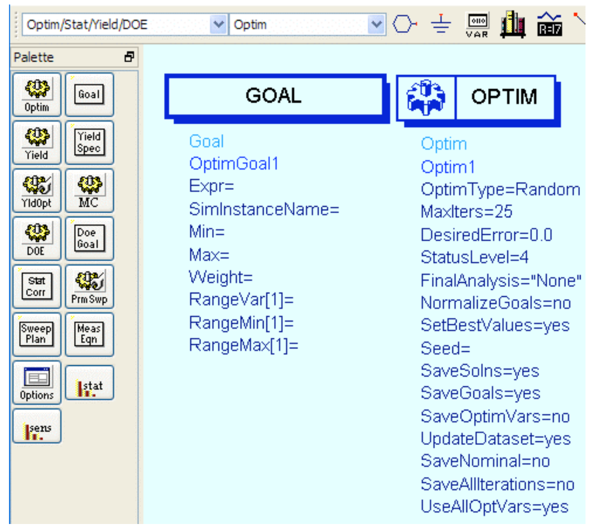
c. **Save** the design but do not close the window.

8. Set up an Optimization controller and Goals.

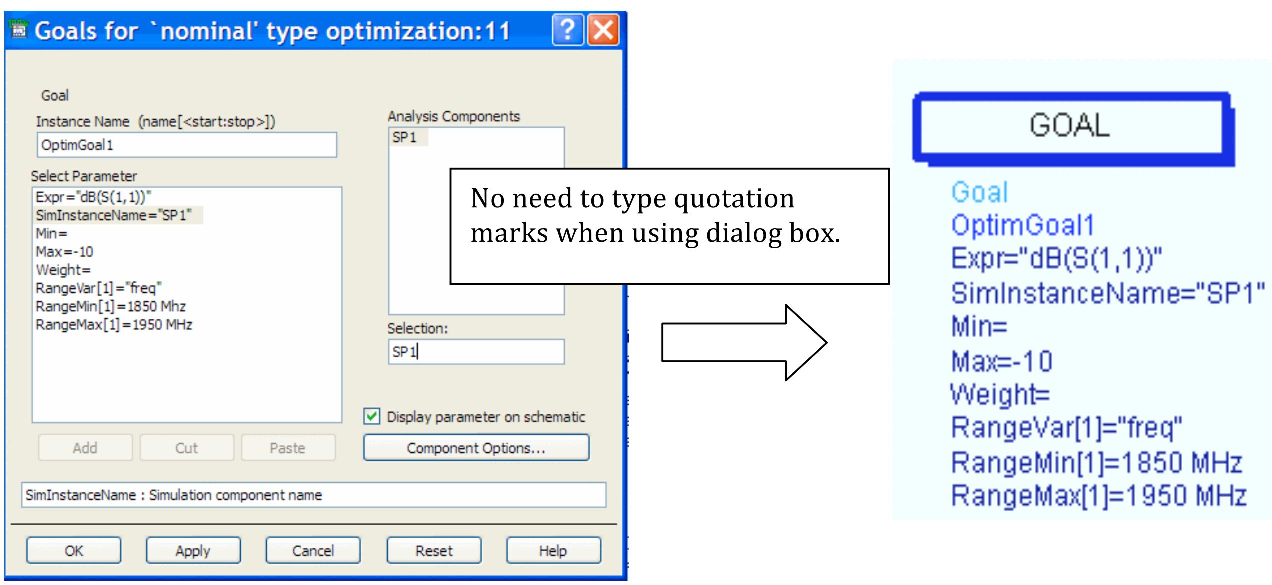
a. Save the s_match schematic design with a with a new name: **s_opt**.

b. Go to the **Optim/Stat/Yield/DOE** palette palette and insert an **optimization controller** and one **goal** as shown here. here.

c. Edit the **goal** by double clicking. In the the dialog box, type in the following following settings and click **Apply** after after each one and OK when done.

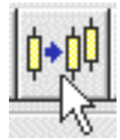


- **Expr: dB (S(1,1))**
  SimInstanceName: SP1.
- **Max= -10** (S11 must be at least –10 dB to achieve the goal)
- **RangeVar=freq RangeMin=1850 MHz RangeMax:1950 MHz**

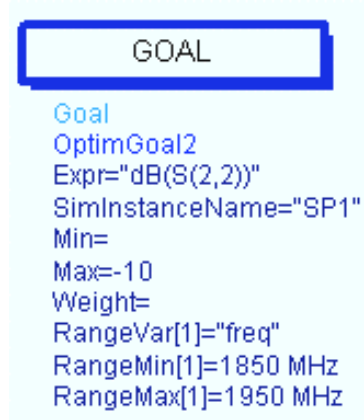


Note on quotation marks - The range values do not need quotes because they are values and not strings (variables).

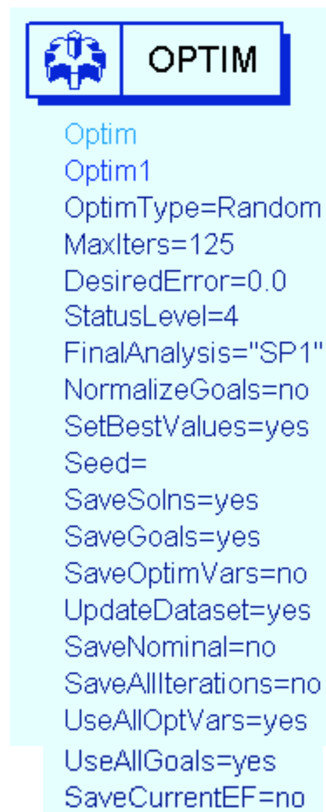
d. **Copy the S11 goal** - select it and use the copy icon.



e. On screen, change the goal expression to **"dB(S(2,2))"** as shown here. Now, you have two goals for the input and output match.



f. **Set up the OPTIM controller**. For this lab exercise, most of the default settings can remain, including the Random type. However, edit the controller and set the **MaxIter = 125** and set the **FinalAnalysis = "SP1**". These settings mean that the optimizer will run for up to 125 iterations to achieve the goals. The Normalize goals setting means that all goals will have equal weighting. Also, a final analysis is automatically run with the last values so that you can plot the results without running another simulation.



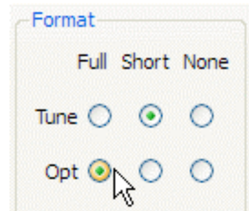
Note on Optim parameter settings –

NormalizeGoals = no means that multiple goals are not equally weighted. To equally weight all goals, set this to yes. For this lab it is not required.

SetBestValues = yes means that the components on schematic can be updated with the best optimized values. The *Save* settings all save data to the dataset. In some cases, this can be a lot of data and use a lot of memory. Also, the default is to use all goals and all enabled components (next steps) on the schematic. However, you can edit the OPTIM controller and select which goals or variables to use. All of the settings are explained in the HELP (manuals).

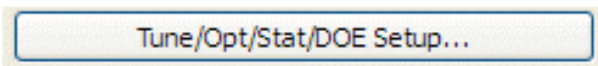
NOTE: The 'Save' parameters that are set to 'no' mean that those values will not be written into the dataset.

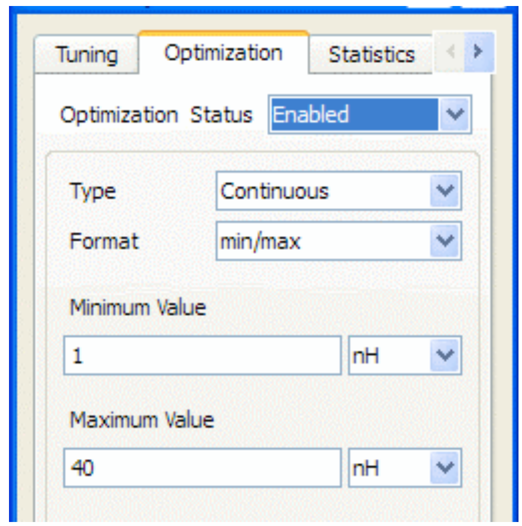
9. Enable the components to be optimized.

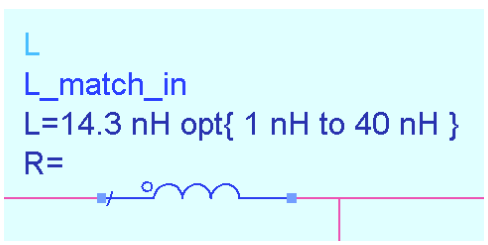


a. In schematic, go to Options > Preferences > and select the tab tab marked: **Component Text/Wire Label**. Turn on the **Full Full** display for Opt as shown here and click **OK**. This will allow allow you to see the range settings.

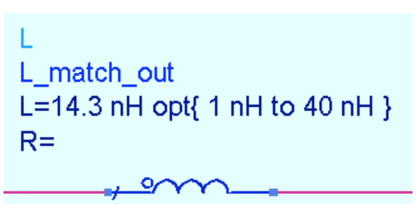
b. Edit (double click) the inductor **L_match_in**. When the dialog dialog appears, click the **Tune/Opt/Stat/DOE Setup** button. In the **Optimization tab**, set the inductor to be **Enabled** as shown and type in the continuous range from **1 nH** to **40 nH** as shown here. Click **OK** and the component text will show the *opt* function and range.

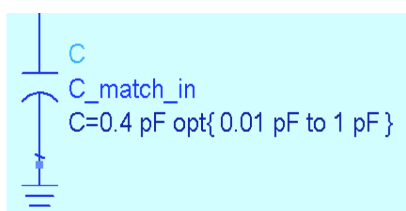


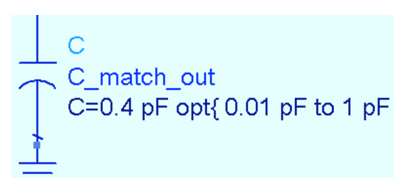




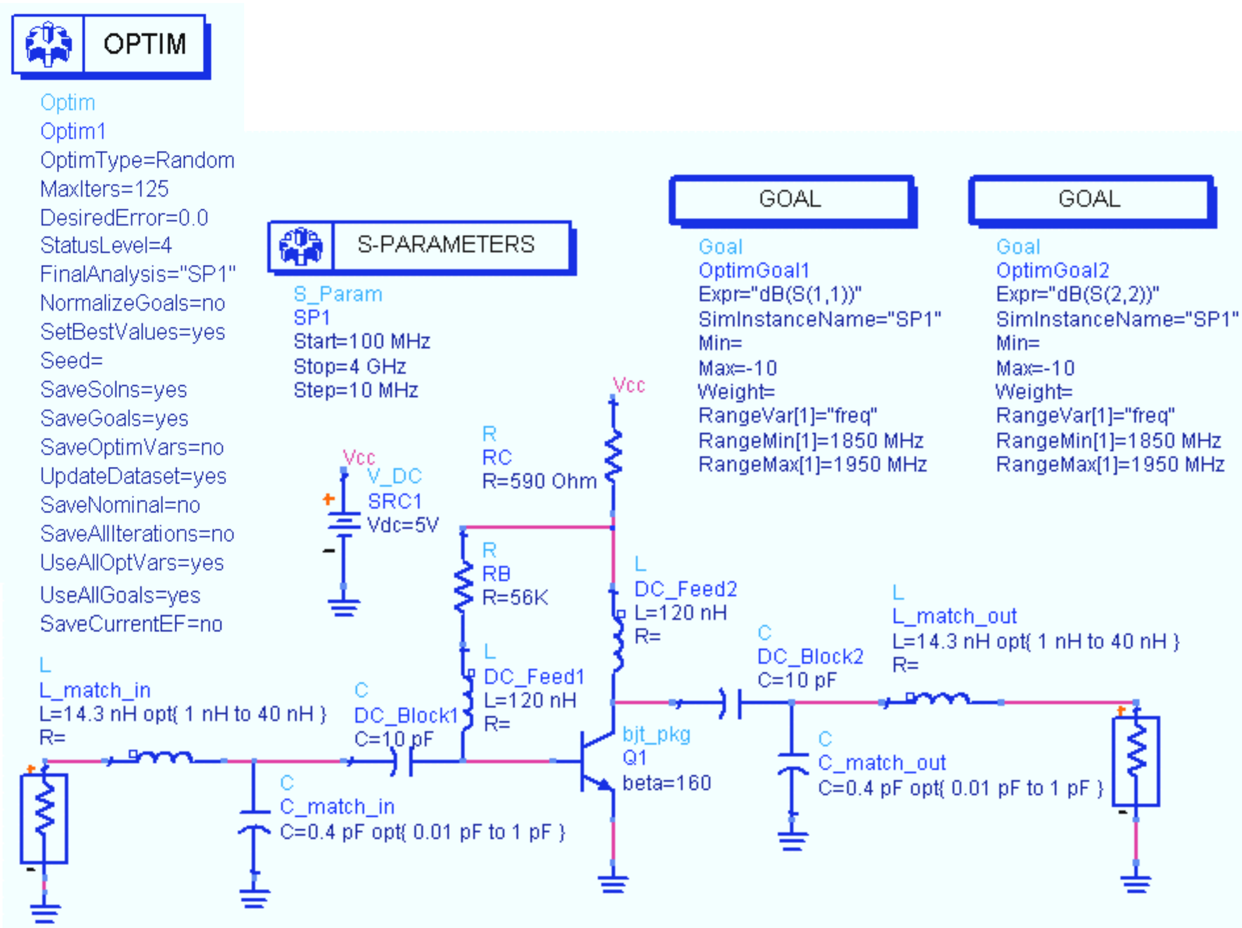
c. Go ahead and **Enable** the other three matching components as shown. Edit each one using the dialog box or you can type directly on-screen using the *opt* function and curly braces for the range. Also, use the F5 key to move component text as needed:



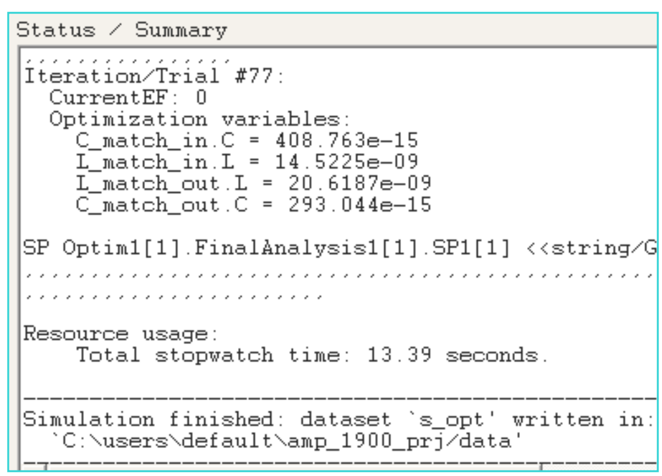




d. Check the circuit as shown here and then **Simulate** and watch the status window.



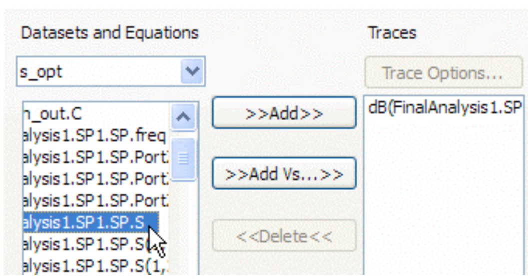
e. The status window reports progress. If the goals are met, the EF (error function) = 0. A successful iteration occurs if the EF moves closer to zero. With EF = 0 (or close in some cases), the next step is to update component values and plot the results. If your EF is not zero, check the schematic and try it again.

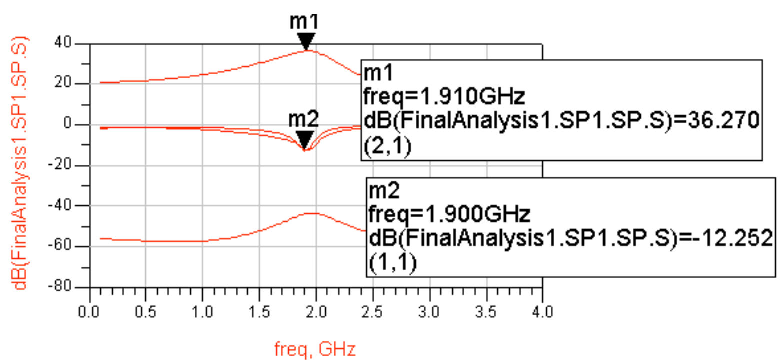


NOTE on optimization EF that does not reach zero - If an optimization does not meet the goal, you can loosen the goals or Desired Error. Also, look for components that are being driven to the ends of their *opt* range and widen them. Also, try another optimization method, increase the number of iterations, or try another topology.

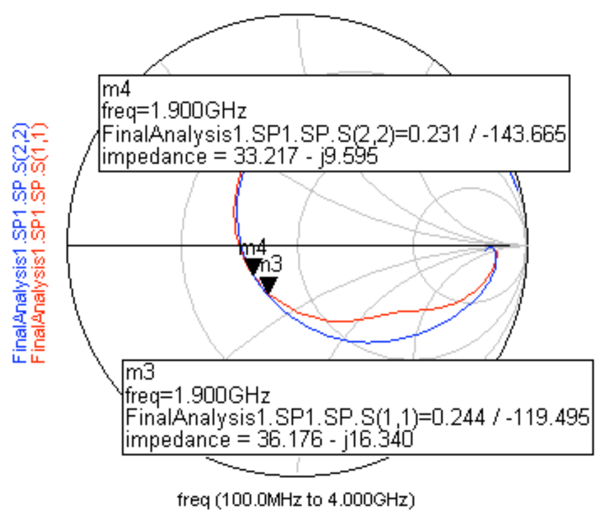
10. Plot the results.

a. In the data display, insert a rectangular plot. Then, as shown here, add the complete **S** matrix of the final analysis in **dB** to see all four S parameters. This way you can quickly verify the results. Your values may differ slightly but the goals from the optimization should be met.



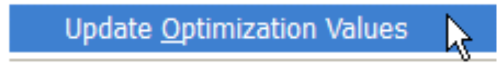


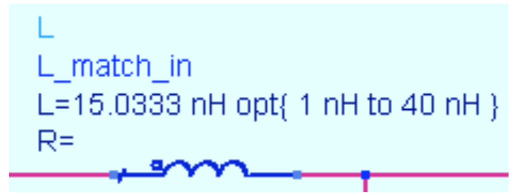
b. Plot the impedance **S11** and **S22** on a Smith chart. Change the marker readout to Zo = 50. As you will see, the impedance is not close enough to 50 ohms, even though the goals were met. Therefore, some modifications will be made in the next steps. But first, you will update the schematic with the values from the optimization.



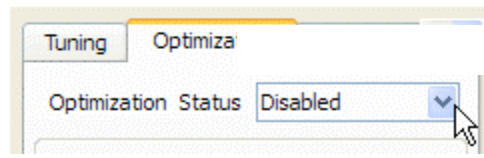
11. Update optimized values and disable the *opt* function.

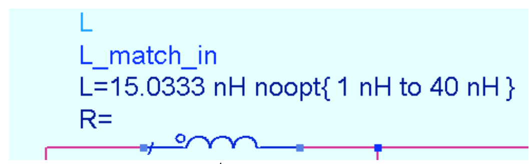
a. Click the command: **Simulate > Update Update Optimization Values.** The enabled enabled components should now have the the final (best) values as the nominal values. values. For example, the input inductor may inductor may look like the one shown here - here - your values may vary a little because of because of the random mode and no seeding. seeding.

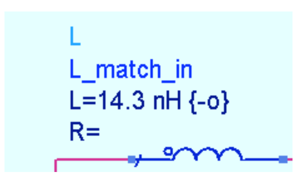




b. Disable a component. Edit (double click) **L_match_in** inductor. Then click the **Tune/Opt/Stat/DOE** button. Select **Disabled Disabled** as shown here and click OK. Notice Notice that the component function changes changes from *opt* to *noopt*. This means the component will not be used in an optimization. You can also disable a component by inserting the cursor on-screen and typing no infront of the *opt* function to make it *noopt* – try it.



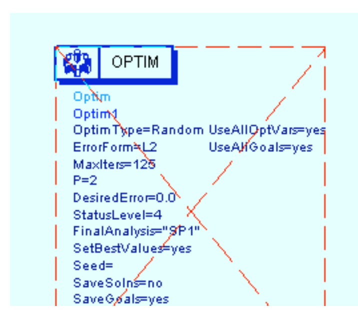




Disabled opt function = noopt, or {-o} if the Options>Preferences (Display Format) is set to Short.

c. **Save** the s_opt schematic. In the next set of steps, you will set up a final matched circuit.

NOTE on deactivating the optimization controller - If you want to simulate without running an optimization, you must deactivate the optimization controller.



12. Simulate the final matched circuit.

a. Save the s_opt schematic as: **s_final**.

b. **Deactivate** (use the icon) the optimization controller and goals.

c. Modify the four L and C matching component values, adding resistance to the inductors as shown here. This will result in a good match and will be used for the remainder of the lab exercises so that all students have the same circuit. Go ahead and change the values by typing directly on-screen as shown here:

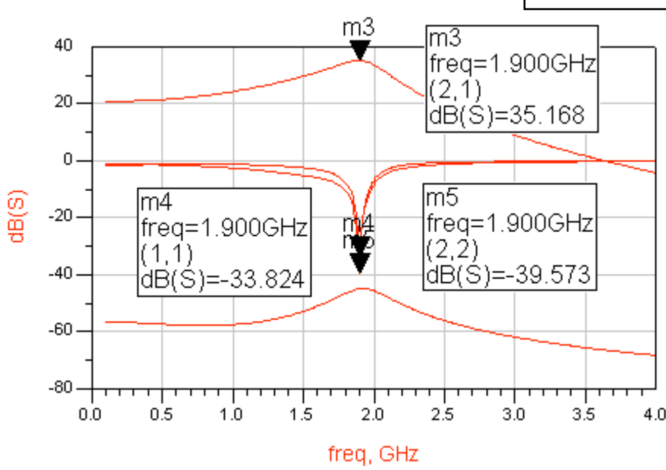
L_match_in = 18.3 nH & R=12 Ohm **L_match_out=27.1 nH & R=6 Ohm**

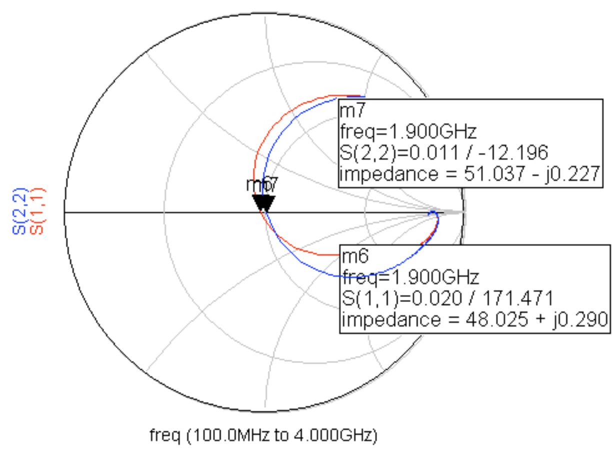
C_match_in = 0.35 pF **C_match_out = 0.22 pF**

d. With the new final component values and **Simulate**.

e. When the data display opens, plot the entire S matrix by selecting **S** in the dataset. Also plot the S11 and S22 on the Smith chart to verify the match is close to 50 ohms at 1900 MHz. With these results, the next steps will be to simulate stability, gain and noise circles.

Improved gain with improved match.





f. **Save** the final design and data display. Close the data display but keep the schematic window opened.

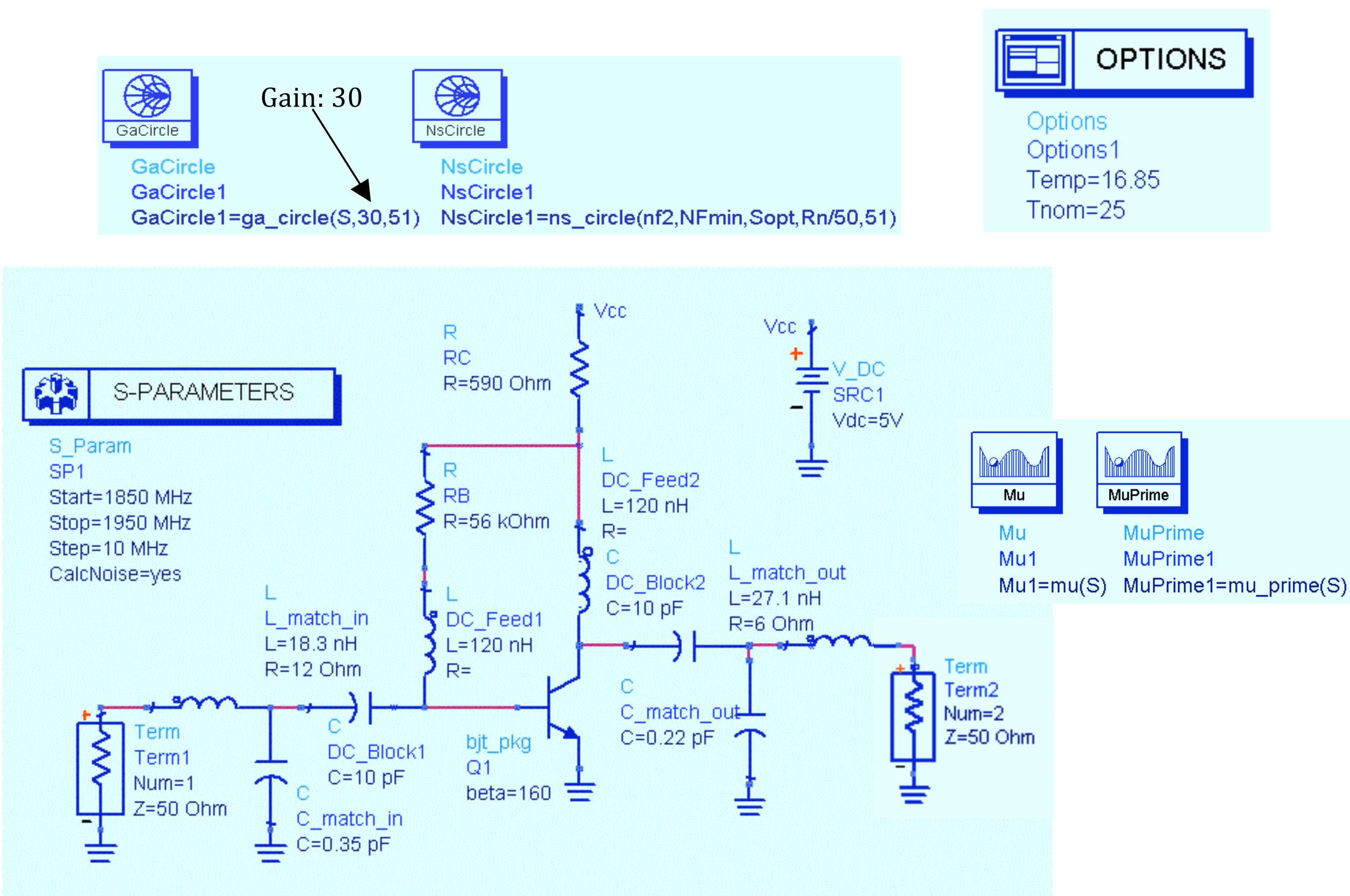
13. Stability equations with gain and noise circles.

a. Save the s_final design as: **s_circles**.

b. Go to the S-parameter simulation palette and insert two stability measurement equations **Mu** and **MuPrime** (icons shown here). These can be used with their default settings. .

c. Scroll down in the palette and insert two measurement equations: **GaCircle** and **NsCircle** as shown. Also, insert the **Options** controller and set **Temp = 16.85** to avoid the warning message for noise. Tnom is the temperature at which a device model is extracted any should not apply here. All other Options default settings are OK.

d. Change the dB gain in the GaCircle to **30** as shown. No setting is required for the NsCircle – it will use NFmin (calculated minimum noise figure) from the simulation data.

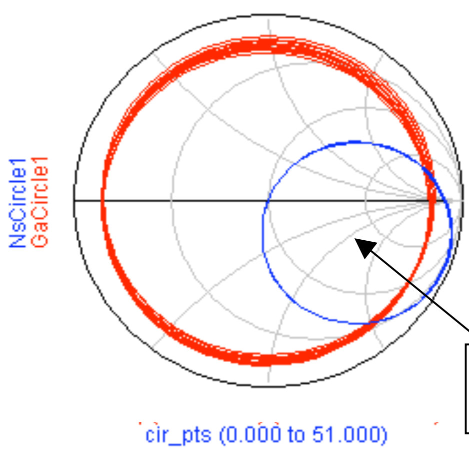


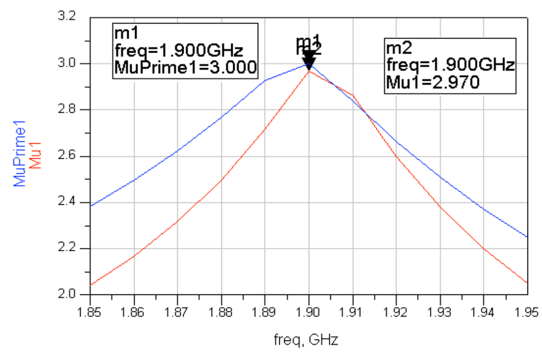
e. **Change the simulation frequency** = **1850 MHz to 1950 MHz** so that fewer data points (circles) will be created. Check the schematic, be sure the noise calculation is turned ON in the controller, and **Simulate**.

f. When the data display opens, plot the measurement equations: **NsCircle1** and **GaCircle1** on a Smith chart as shown.

g. In a rectangular plot, add Mu1 and MuPrime1 as shown.

h. Also, insert a list of **nf(2)**, **NFmin**, and **Sopt**.





If source reflection coefficient is equal to the center of the noise circle, you get minimum noise figure.

freq	NFmin	Sopt	nf(2)
1.850GHz	1.055	0.788 / -20.779	3.153
1.860GHz	1.055	0.789 / -20.735	3.171
1.870GHz	1.056	0.791 / -20.691	3.189
1.880GHz	1.056	0.792 / -20.647	3.207
1.890GHz	1.056	0.794 / -20.602	3.225
1.900GHz	1.057	0.795 / -20.557	3.243
1.910GHz	1.057	0.797 / -20.512	3.261
1.920GHz	1.057	0.798 / -20.467	3.279
1.930GHz	1.057	0.800 / -20.422	3.298
1.940GHz	1.058	0.801 / -20.376	3.316
1.950GHz	1.058	0.803 / -20.330	3.334

NOTE on results - On the Smith chart, the area inside the Gain Circle indicates the load impedance that will result in 30 dB of gain. The Noise Circle is different because its center indicates the optimum value of source reflection coefficient that will result in the minimum noise figure (NF min). With the center of the Noise Circle is within the Gain Circle, both Gain and NFmin can be achieved. The two stability traces, Mu (load) and MuPrime (source), are greater than one. This means the circuit is stable (it will not oscillate) within the 100 MHz bandwidth. Finally, the listed value of nf (2) is the noise figure when port 2 is the output port. This value would improve if the source reflection coefficient were equal to Sopt (optimum source match).

i. **Save** and **close** all the designs and data displays in the project. At this point, the amplifier is ready to be tested with the non-linear simulator, Harmonic Balance. However, before doing so, you will return to the system project in the next lab and build the two filters for the RF system.

14. OPTIONAL - Read and Write 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. Open a new schematic and save it as: **s2p_data**.

b. Click on the **Data File Tool** icon.

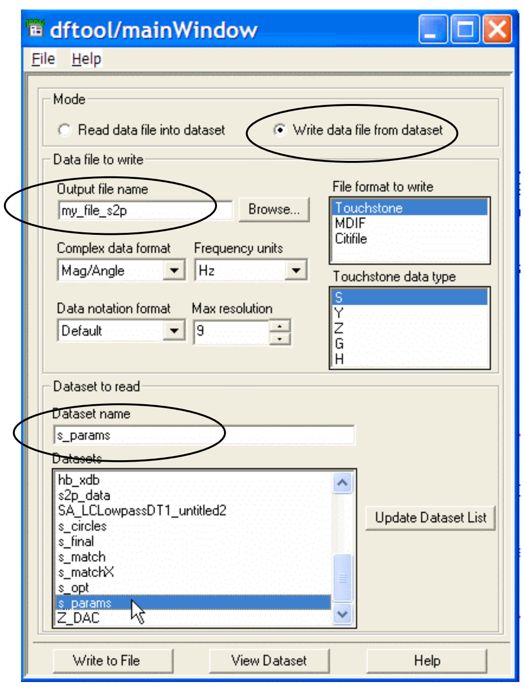
c. When the dialog box opens, click the box to **Write data file from dataset**, then select the **Touchstone** format. You are going to write (convert) your existing ADS dataset (s_params) into a Touchstone file. It will represent measurement data from a network analyzer.

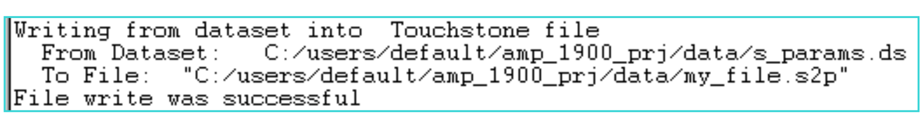
d. In the **Output File Name** field, type: **my_file.s2p**. This will be the name of the the Touchstone format file that will be converted from ADS data.

e. Select the Output Data Format as **Mag/Angle**. **Mag/Angle**.

f. In the Datasets field, select the dataset: **s_params**. This was the dataset from the the simulation using the ideal components. components.

g. Click **Write to File**. Check the Status Window. Window. If successful, you will see a message. message. This means *my_file.s2p* is now a a Touchstone file in the data directory of the the amp_1900 project. You can check this if you this if you want and you can use a text editor editor (ADS Main window: Tools > Text Editor) Editor) to look at or to modify the file.

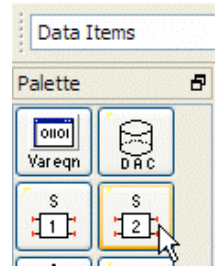




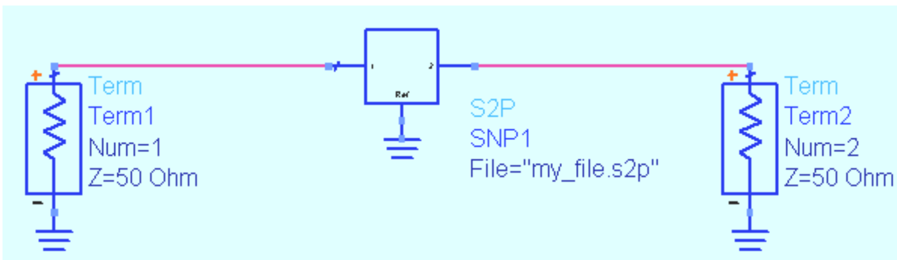
```
Writing from dataset into  Touchstone file
  From Dataset:   C:/users/default/amp_1900_prj/data/s_params.ds
  To File:  "C:/users/default/amp_1900_prj/data/my_file.s2p"
File write was successful
```

h. Close the Data File tool window.

i. In the empty schematic, insert an **S2P** component from the **Data Data Items** palette. You will notice that the component variable variable (*File=*) is not yet assigned



j. To assign the data, edit the S2P component and another dialog dialog box will appear. Next, **browse** for the file name. When the When the next dialog appears, select **my_file.s2p** and click the the **Open** button and the file name will be assigned (shown here).



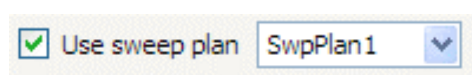
(shown here).

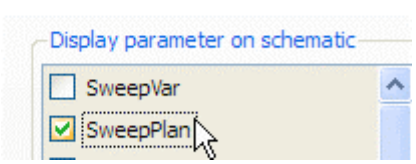
k. In the schematic, insert an **S_Params** template (Insert > Template) and wire the S2P component to the Terms with grounds as shown here.

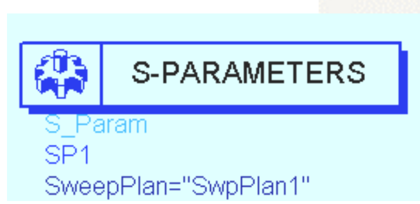
l. From the Simulation- S_Param palette, insert a **Sweep Plan** and set it to **Start= 100 MHz, Stop = 3 GHz** and **Step=100 MHz** as shown. Sweep plans are normally used for frequency sweeps within sweeps but you can use it here to see how it replaces the Frequency settings in the S-parameter simulation controller.

m. To use the Sweep Plan**, edit the simulation controller**. In the Frequency tab, select the **SwpPlan1** as shown here. Go to the Display tab, tab, select SweepPlan and remove the start, stop and stop and step as shown here.

Simulation Controller:
Frequency tab -

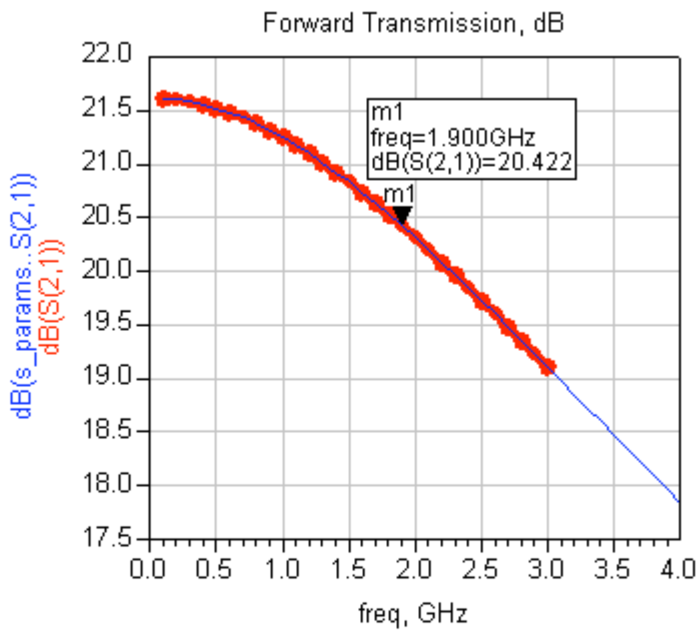






n. **Simulate** and the results will automatically appear in the Data Display window because the template has a DDS display template also.

o. Zoom in on the S21 measurement and add the S21simulation data from your original s_params dataset to verify that the Touchstone file correctly represented the data. As you will see, the two traces are identical except that the S2P simulation only goes to 3 GHz. Here, the trace thickness and types have been adjusted (using Trace Options) to show both traces more clearly. Markers have also been added.

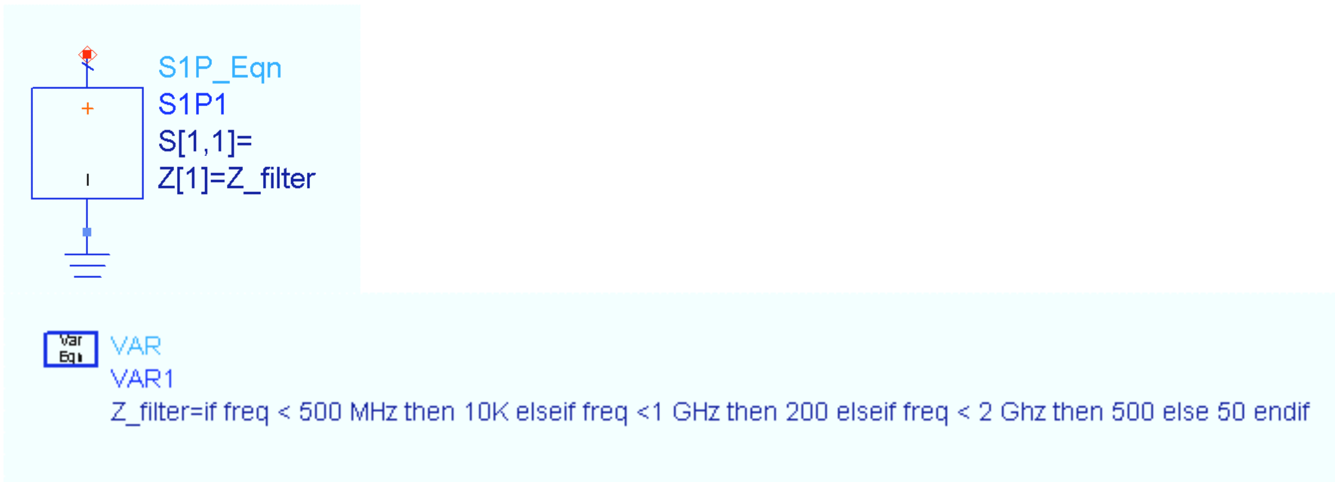


15. OPTIONAL – YIELD analysis

Refer to the theory slides and run the yield analysis example with a different yield spec (15dB for example) using a different frequency range. Examine the results in the data display.

***EXTRA EXERCISES*:**

1. Z_PORTS - In a separate schematic, set up an S parameter simulation of impedance that is described by an equation as shown here. Plot the response and try adjusting the values.



2. Set up the simulation using a sweep within a sweep using two or more Sweep Plans.

3. Go back to the optional exercise and use the text editor to edit the s2p file, change some values, and simulate to verify that you can do this.