LAB EXERCISE 2

Matching Networks and Optimization

Topics: Small-signal S-parameter optimization and designing matching networks for amplifiers and other designs. Optional is the Impedance Matching too.

Audience: Engineers who have a basic working knowledge of ADS or have completed the prerequisite course.

Prerequisites: Completion of lab exercise 1 of this course. Also, completion of *Workspaces and Simulation Tools* or equivalent experience, including basic circuit design concepts.

Objectives: Be able to set up and run the optimizer with multiple goals.   Also, simulating to verify amplifier gain and stability using a matching network.

Table of Contents: Lab 2

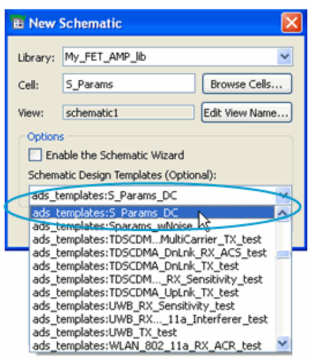
Lab 2: Optimization

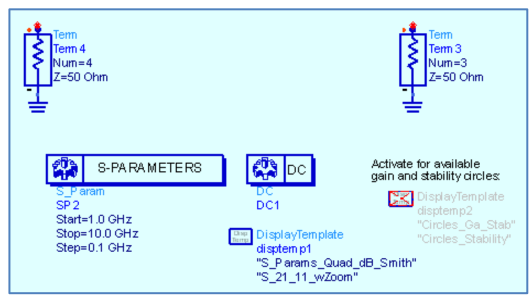
About this lab - This lab uses small signal optimization for matching a FET amp to 50 Ohms. This will teach you how the optimizer and goals are used. Afterward, you will simulate with a matching network based on Load Pull results (lab 1) and created with the ADS Impedance Matching tool (optional step only if you have time). Because many designers use optimization for many purposes, this lab will teach all the basics.

1. Amplifier Design with Template

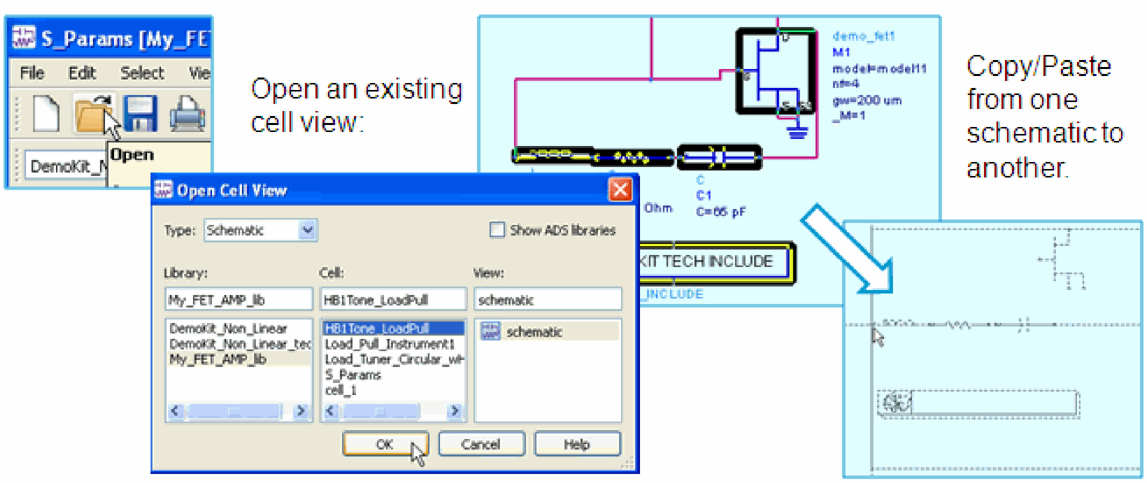
a. In your my_FET_AMP workspace, add a new cell / schematic and select the **S_Params_DC** template. Name the new cell **S_Params** and click OK.

This will be used to build and test the FET amp before optimization. The templates automatically add simulators and components so you don't have to insert them from the palette – this saves time and effort. Your schematic should now have the template items as shown here. Save the design (good practice).

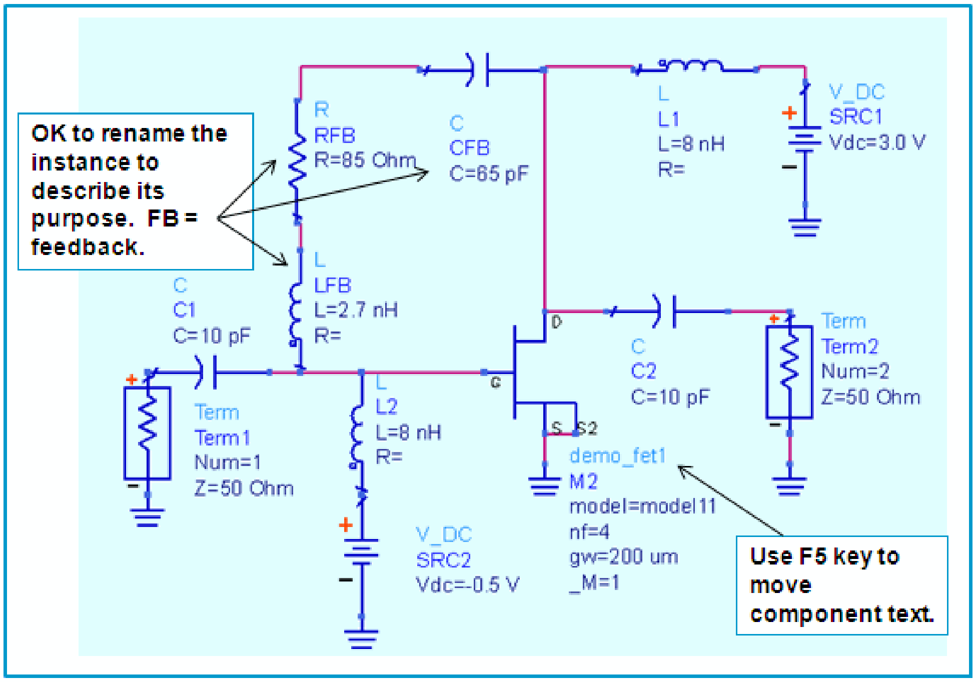




b. In the corner of the schematic, click the **Open** icon as shown here to open the **HB1Tone_LoadPull** schematic. Now, **copy** (Ctrl-C) the **feedback network** (LRC), the **FET**, and **INCLUDE** component as shown here. Then **paste** (Ctrl-V) it into the new **S_Params** schematic. Close the Load Pull schematic.



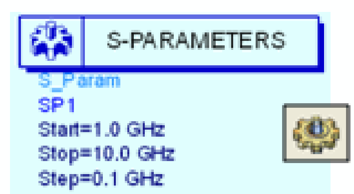
c. **Complete the design as shown here**. You will need to insert or copy the remaining lumped components and DC sources - you should know how to do this already. At the least, rename the feedback components as shown here - this makes it easy to identify them for optimization or not) later. Note that the values used for the bias L and C are approximations.



d. **Activate** the Display Template for the gain and stability circles as shown here. This will create the pages in the Data Display and the plots as you will see.

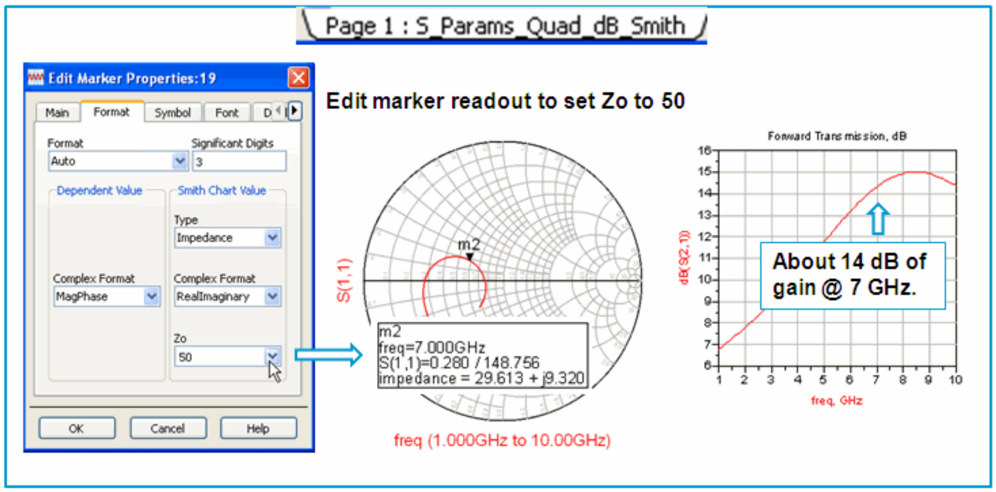
2. Simulate and examine Gain and Stability

a. If your design is correct and you have activated the template, go ahead and **run the simulation** using the default simulation shown here.

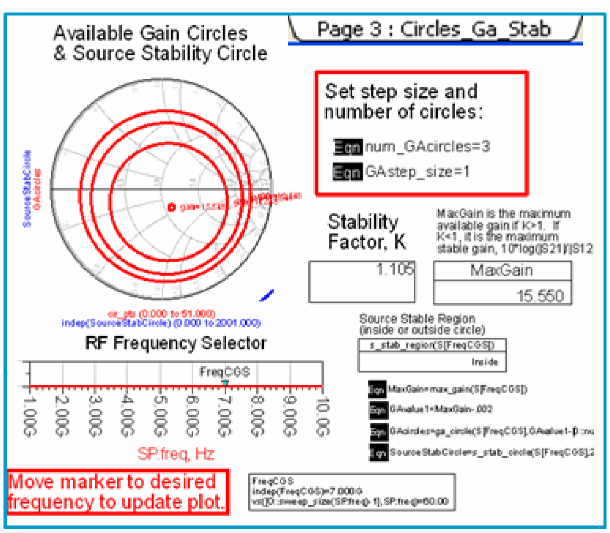


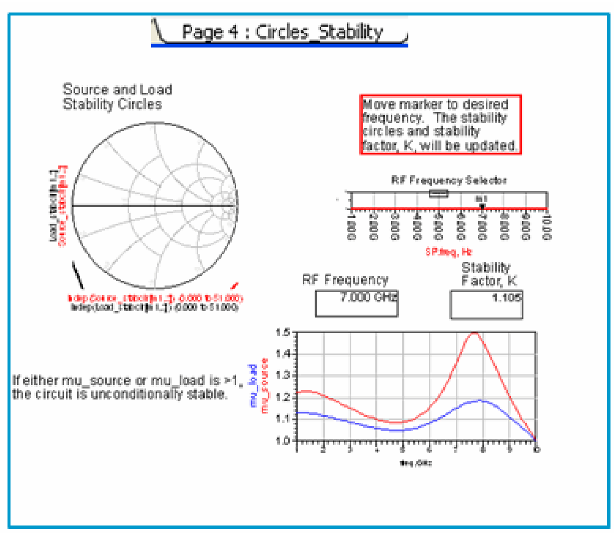
Within a few seconds, the simulation should be complete and the data display window will open with all the template results. This saves a lot of time…

b. Look at **Page 1** of the Data Display. Put a marker on S11 and set the Format to 50 as shown here. These results are expected – Zin is about 29 +j*8 and the gain is about 14 dB at 7 GHz. Also, look at page 2 for zoomed-in views.



c. Look at **Pages 3** and **4**: these are the activated template pages and they show both gain and stability circles. Move the markers to **7 GHz** as shown here. Try changing the step size of the gain circles. As expected, the greatest gain is at the center of the Smith chart. On page 4, the marker at 7 GHz shows that the amplifier is stable. With the source stability circle outside the Smith chart, it simply means that anywhere inside the chart is a stable region – this is good. Next, it's time to set up the Optimization.

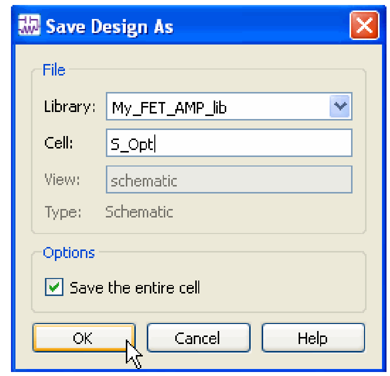




d. **Save** the schematic (keep it open) - **Save** and **close** the **data display** window.

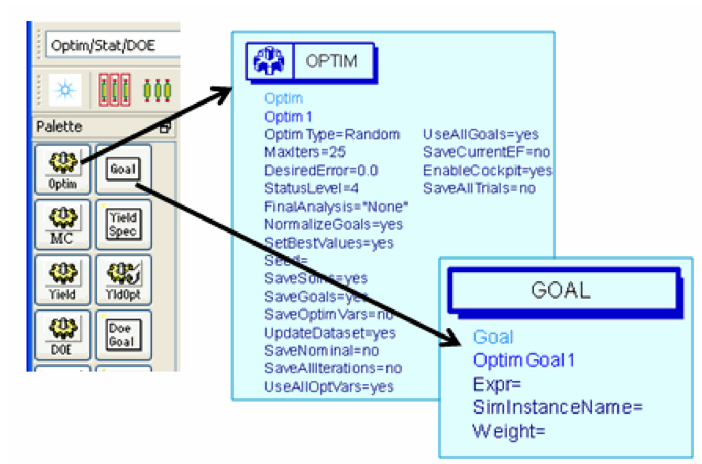
At this point, you have seen some load pull results plus the Gain and Stability results. Next, you will set up an optimization to match the input and output to 50 Ohms.

3. Set up an Optimization controller and Goals



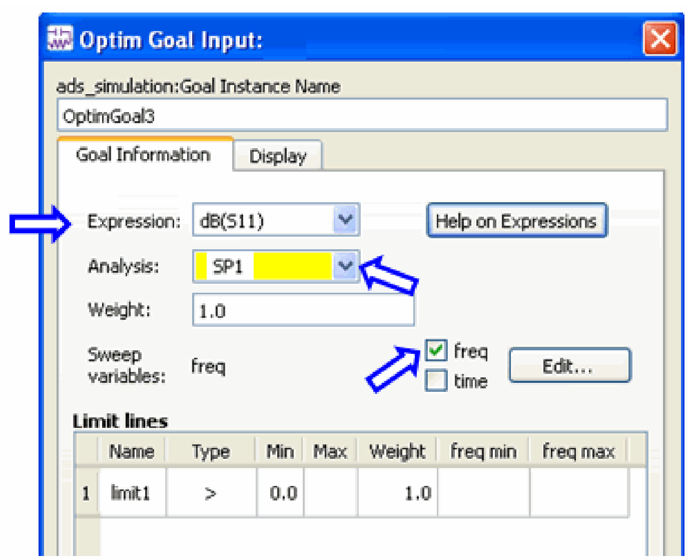
a. In your schematic, use the command: **File > Save As** and save it as **S_Opt**. Now you only have to swap out a few components for the optimization. This also keeps the workspace organized with cells that describe the contents or simulations.

b. Go to the **Optim/Stat/ DOE** palette and insert an **Optimization controller** and one **Goal** shown here.



c. Edit the **Goal** by double clicking. In the dialog box, type in or use the buttons to make the following settings and click **Apply** each time:

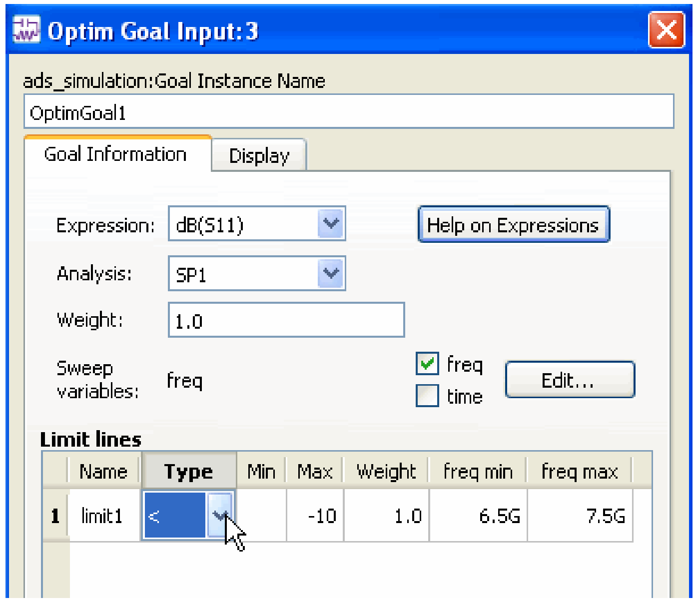
- **Expression: dB (S11)**.
- **Analysis: SP1** (or the S_param controller name that appears) with the arrow button.
- **Freq:** Check the box and type in the frequency range columns will be added to the limit lines – you will set the frequency in the next steps.



- **Type:** Use your cursor to set the limit type to: **<** (less than).
- **Max**: Type in the value of **-10**.

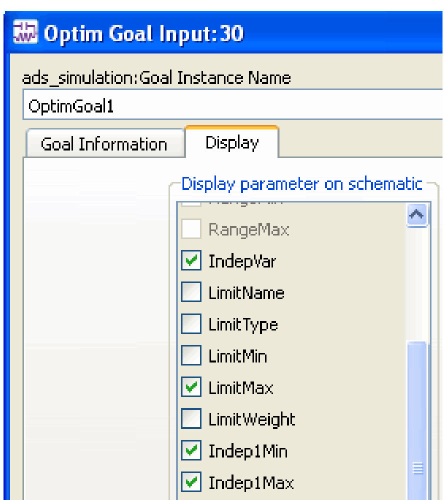
  This means your goal must be a value of S11 that is less than -10 dB. In other words, it cannot exceed the maximum limit of -10 dB.

- **Freq min** & **freq max**: **T**ype in the values for the frequency range: **6.5G** and **7.5G**. This means that the optimization will only operate from 6.5 GHz to 7.5 GHz. Also, G is used (same as e^9) because *freq* is a reserved variable and uses Hertz by default. Click **Apply**.

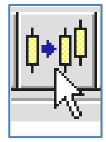


Verify that your Goal Information and limit lines are set as shown here:

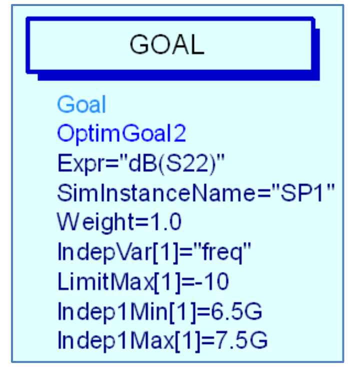
d. Go to the **Display** tab and check the boxes shown here. Click **Apply** and then OK. Your Goal should look like this one - if not, check your work.



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



f. On screen, change the goal expression to **"dB(S22)"** as shown here. Now, you have two goals: one for the input and one for the output match.



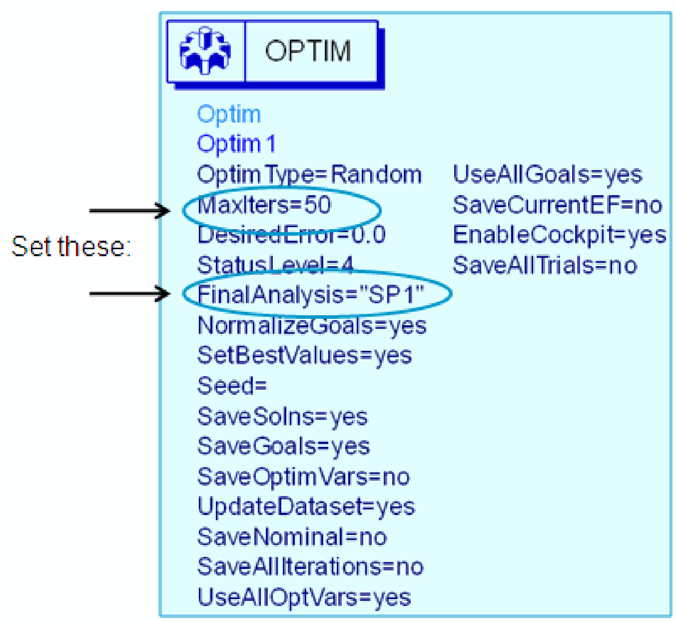
g. Set up the **Optimization controller**. For this lab exercise, most of the default settings can remain, including the Random type. However, use your cursor on-screen to change the **MaxIter = 50** and the **FinalAnalysis = "SP1**". These settings mean that the optimizer will run for up to 50 iterations to achieve the goals. The Normalize goals setting means that all goals will have equal weighting. Also, the Final Analysis set to **yes** means it will automatically run one final simulation with the last optimization values so that you can plot the results without running another simulation.

Note on other Optim parameter settings:

NormalizeGoals = yes means that multiple goals are equally weighted.

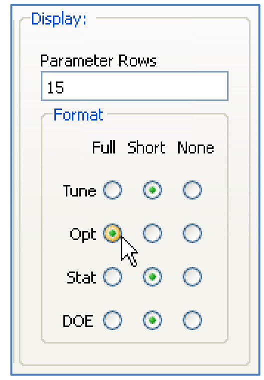
SetBestValues = yes means that the components on schematic can be updated with the best optimized values.

All the *Save* settings save data to the dataset. In some cases, this can be a lot of data and 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.



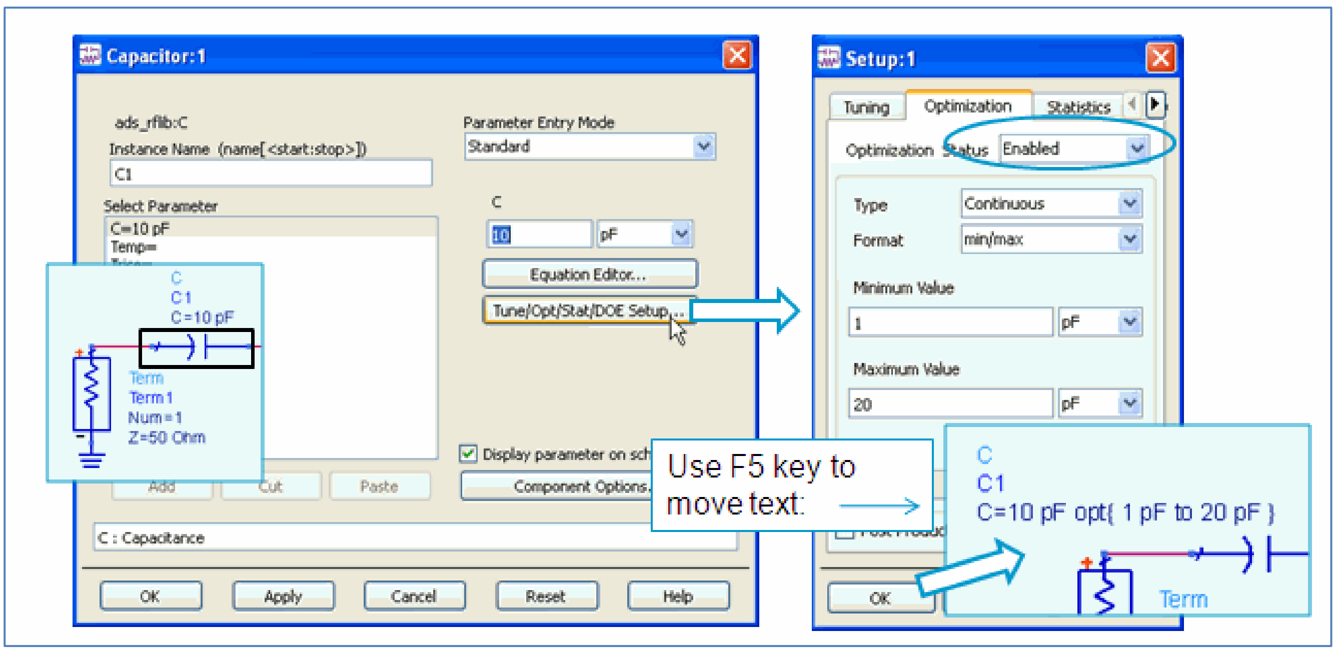
NOTE: The 'Save' parameters that are set to 'no' mean that those values will not be written into the dataset.

4. Enabling Variables or Parameters to be Optimized

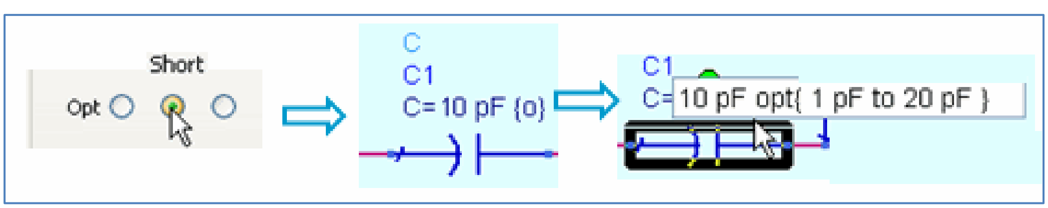


a. In schematic, use the command **Options > Preferences** and select the tab: **Component Text/Wire Label**. Turn on the **Full** display for **Opt** as shown here and click **OK**. This will allow you to see the range settings.

b. **Edit** (double click) the input capacitor (C1). When the dialog appears, click the **Tune/Opt/Stat/DOE Setup** button. In the **Optimization tab**, set the Optimization Status to **Enabled**. Then type in the continuous range: **1 pF** to **20 pF** as shown here. Click **OK** twice and the component text will show the ***opt*** function and range in the full format. Use the F5 key to move the text if it overlaps the component or wires.

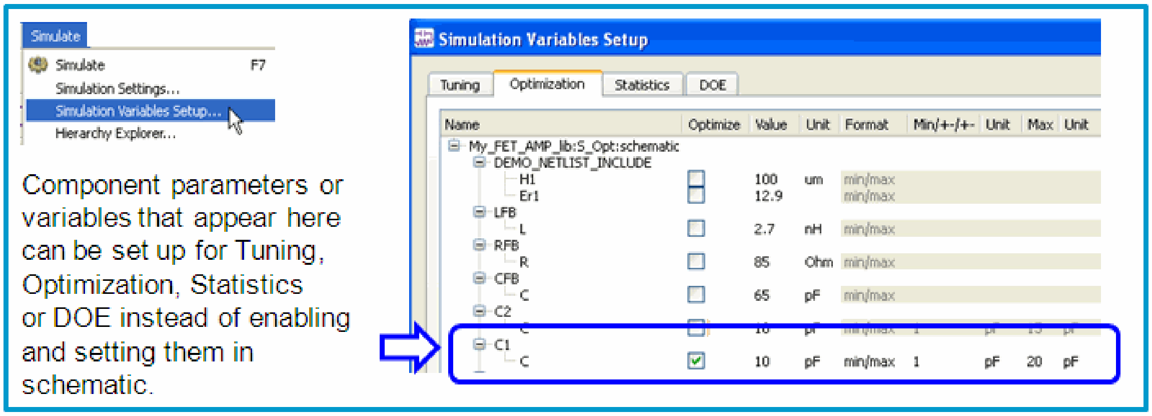


c. Go back to **Options > Preferences**, select the **Component Text/Wire Label** tab and reset the Opt Display Format to **Short** and click **OK**. Then insert your cursor on-screen into the value area and notice that the full details appear. This is how you control displayed component values for Tuning, Optimization or Statistics.

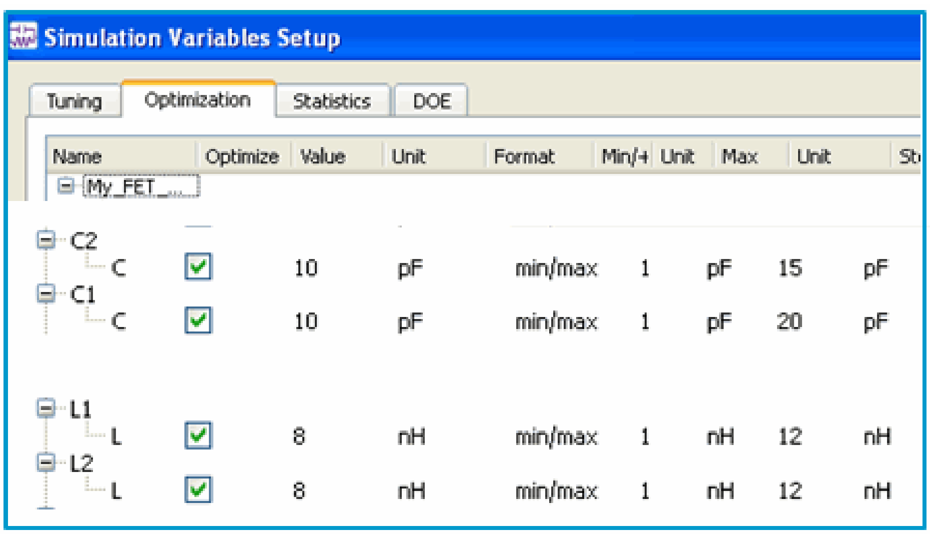


Now you know how to enable and display the values for a single component. However, there is another way to control variables for optimization, especially if multiple variables are to be tuned, optimized, or use for statistical analysis as you will now see.

d. Click the command **Simulate > Simulation Variables Setup**. Go to the **Optimization** tab and notice that your enabled capacitor and values appear. This setup makes it easy to set multiple parameters or variables (VARs).

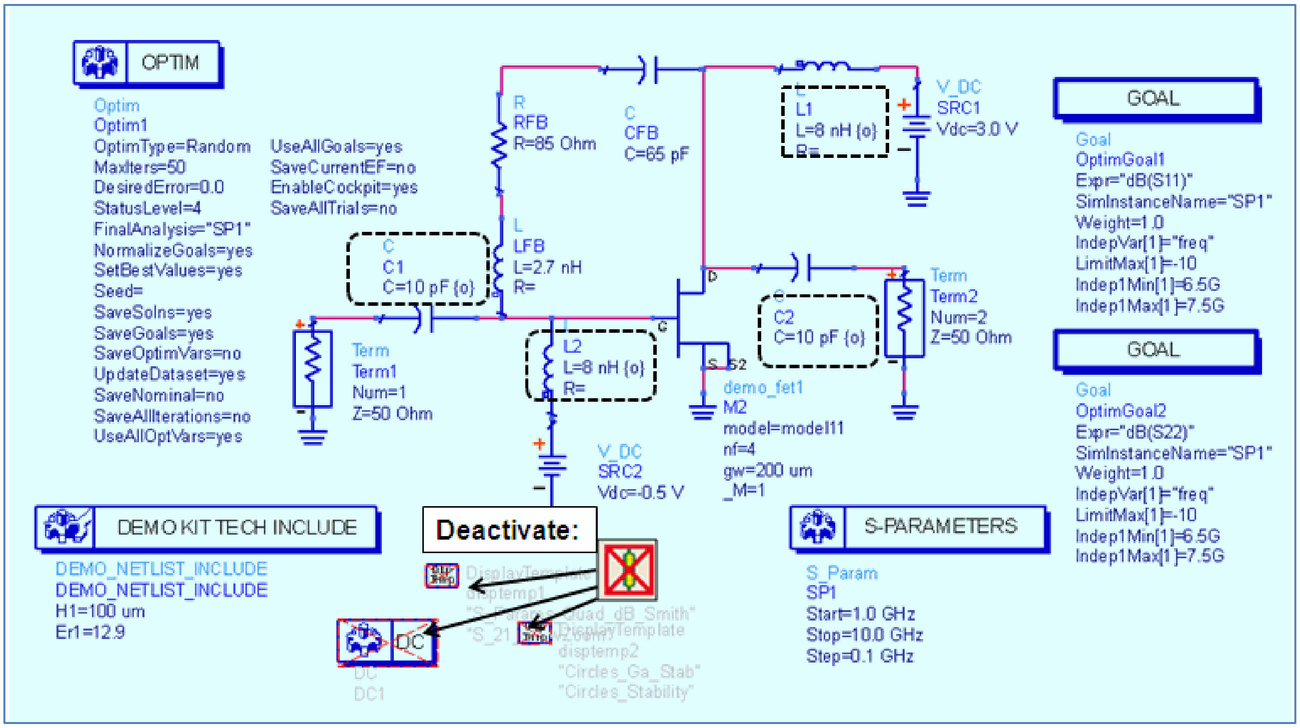


e. Set the other bias **capacitor** and **two inductors** (C2, L1 and L2) to be optimized (check box) with the values shown here – click **OK** when done.



NOTE: Naming the stabilization feedback components (LFB, RFB, CFB) made it easy to identify from the bias L and C components without looking at the schematic.

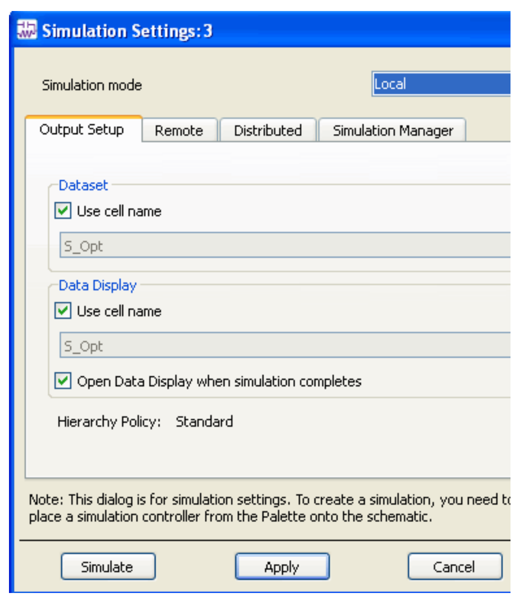
f. **Deactivate** the **DC** controller, and the two **data display templates**. These can be re-activated later if you need them. Verify that your schematic looks like the one shown here with four components enabled for optimization.



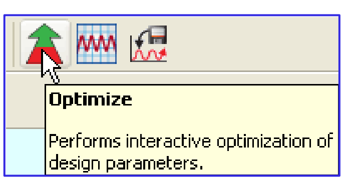
5. Running an Optimization

a. Before running the optimization, check the **Simulate > Simulation Settings** and set the dataset to the cell name as shown here. Click **Apply** and then **Cancel**.

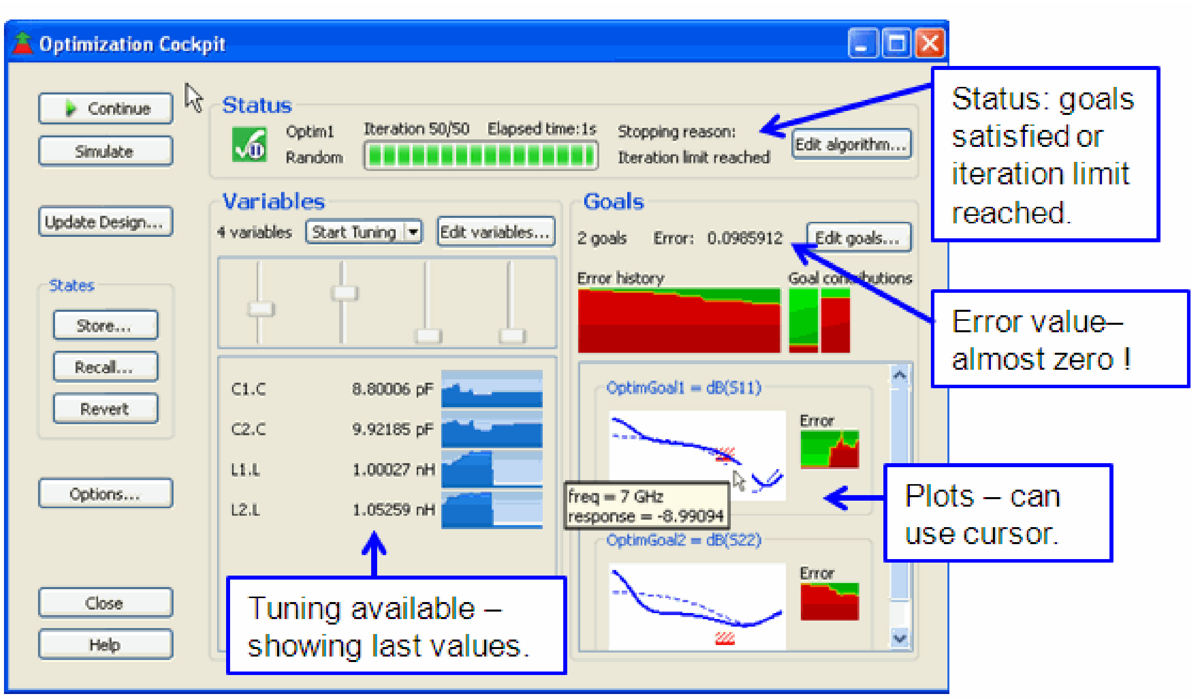
This is always a good idea to ensure you do not overwrite a dataset.



b. Click the **Optimize** icon to run the optimization.

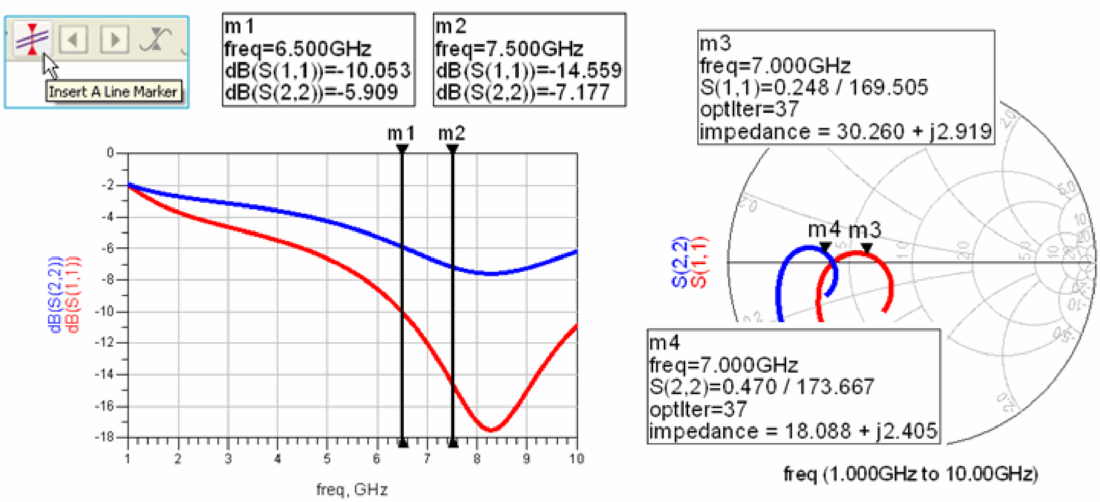


c. Immediately, you will see the **Optimization Cockpit**, Status window and Data Display appear. The cockpit shows all the information you need. Notice that the goals are not achieved after 50 iterations but the error function is close to 0.



NOTE: Your values may differ slightly due to the random seeding and algorithm.

d. In the **Data Display** (S_Opt) insert a rectangular and a Smith chart as shown here. On the rectangular plot, insert **Line Markers** on **6.5 GHz** and **7.5 GHz**. On the Smith chart, set the markers to **7 GHz** in **50 ohm format**. Notice that the S11 goal was achieved but S22 was **not** achieved.

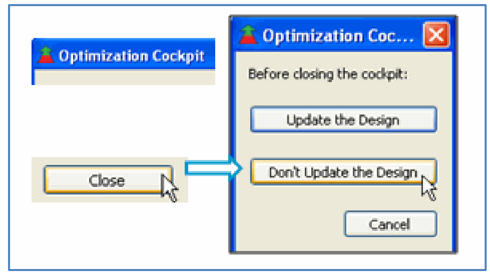


e. **Save** and **close** the **Data Display**

f. Click the **Close** button in the Optimization Cockpit – the click the button: **Don't Update the Design**. This means the L and C values will not be updated – this is OK because you can re-run the optimization to get the results back again. Also, you are going to be using new values for the amplifier as you will see later.

g. **Save** the schematic but keep it open.

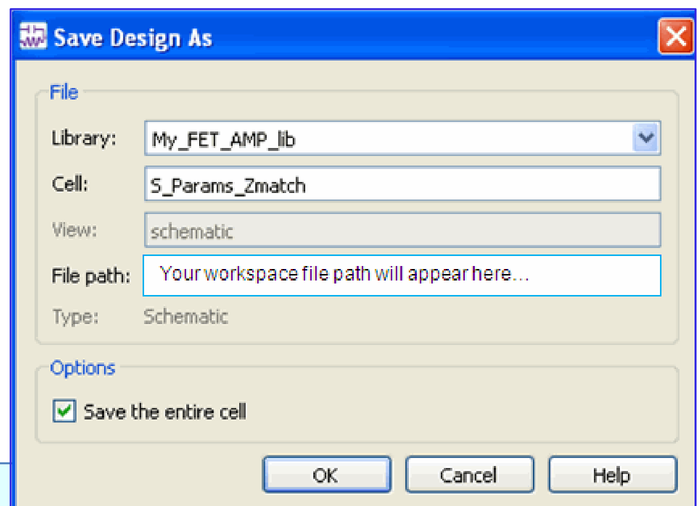
NOTE: Optimization can be use with other simulators such as Harmonic Balance to achieve other goals. The Examples directory has numerous optimization setups.



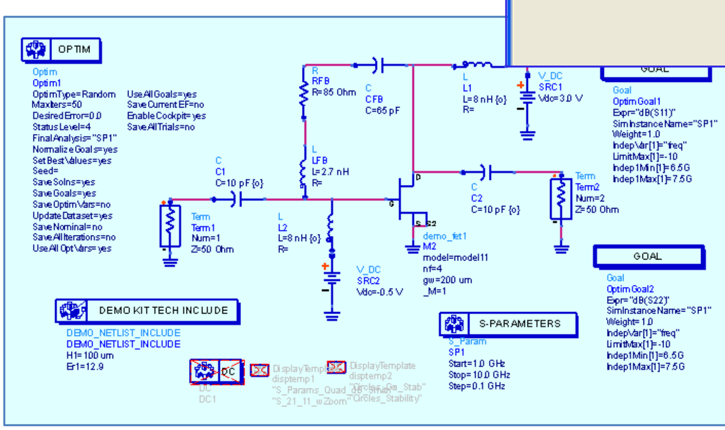
6. Simulate with a Matching Network

a. In the S_Opt schematic, click the command: **File > Save As**.

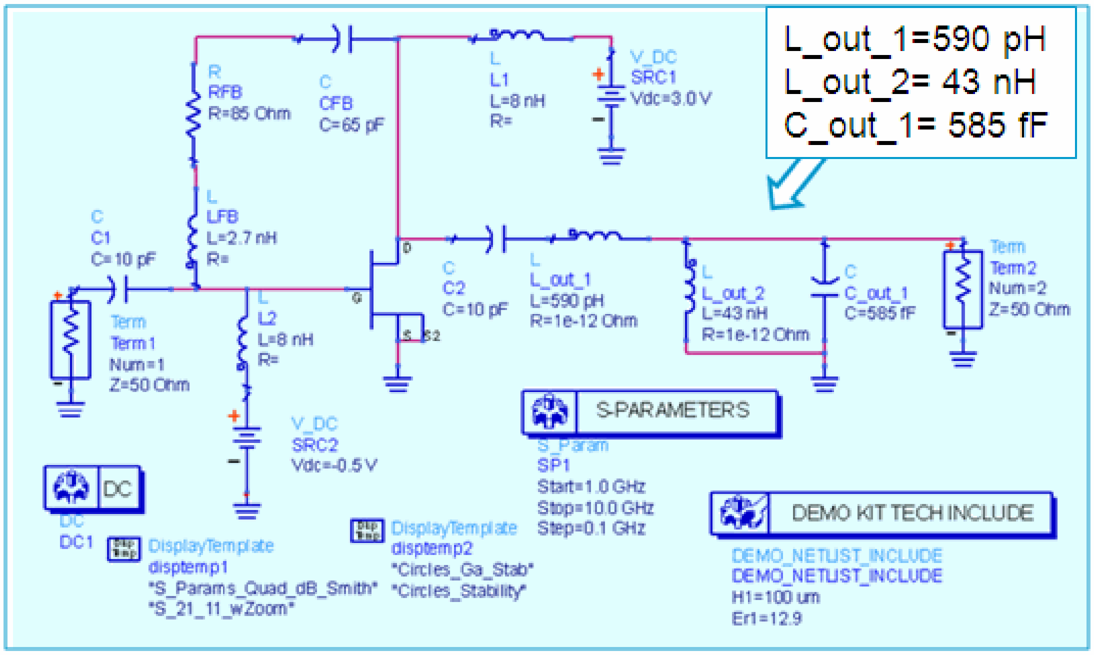
b. When the dialog appears, name the cell: **S_Params_Zmatch** as shown here - then click **OK**.



The copied schematic should appear in a new window ready to use for the next steps.



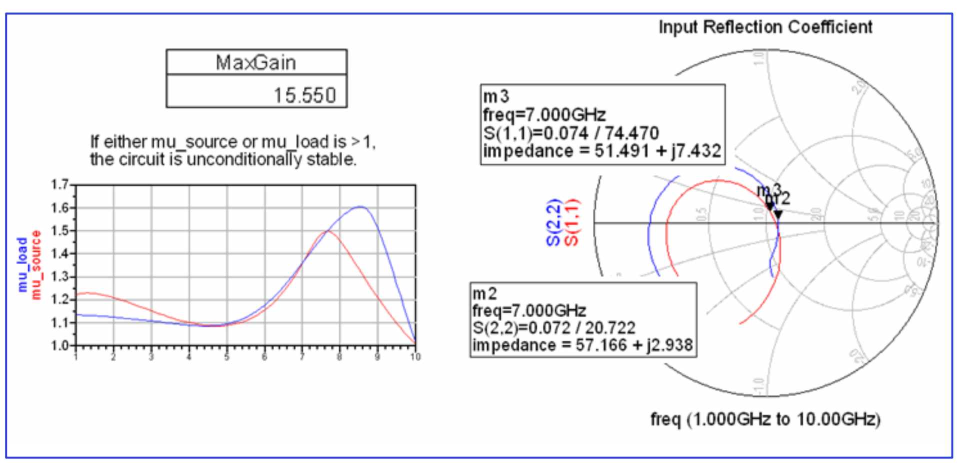
c. In the new schematic, S_Params_Zmatch, **add the output matching network** with the L and C values to as shown. If you have time, rename the components (L_out_1, L_out_2, C_out) as shown. This may be helpful for identifying their purpose. The resistive value for the inductors is negligible but realistic.



d. **Activate** the DC controller and the two templates as shown here.

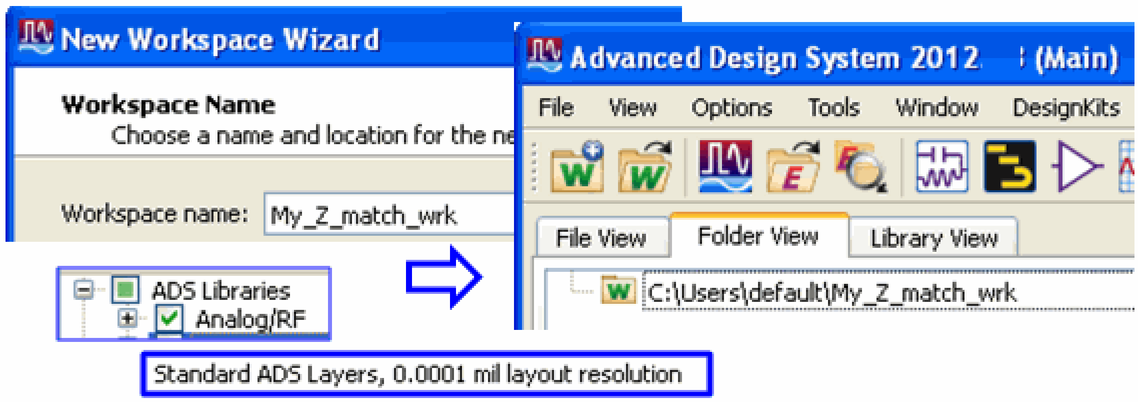
e. **Simulate**. Then verify that your schematic is similar to one shown here and **Simulate**. Then go through the pages of the Data Display - you should see similar results – **add S22 to the Smith chart as shown**. This design shows stability and, at 7 GHz, good gain with a good match to 50 ohms.

f. **Save** and close all your work. - but continue to the Optional Step if you have time.

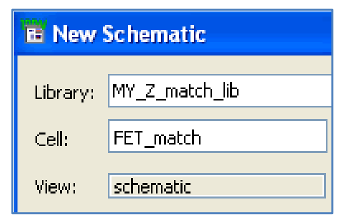


7. OPTIONAL: Impedance Matching tool

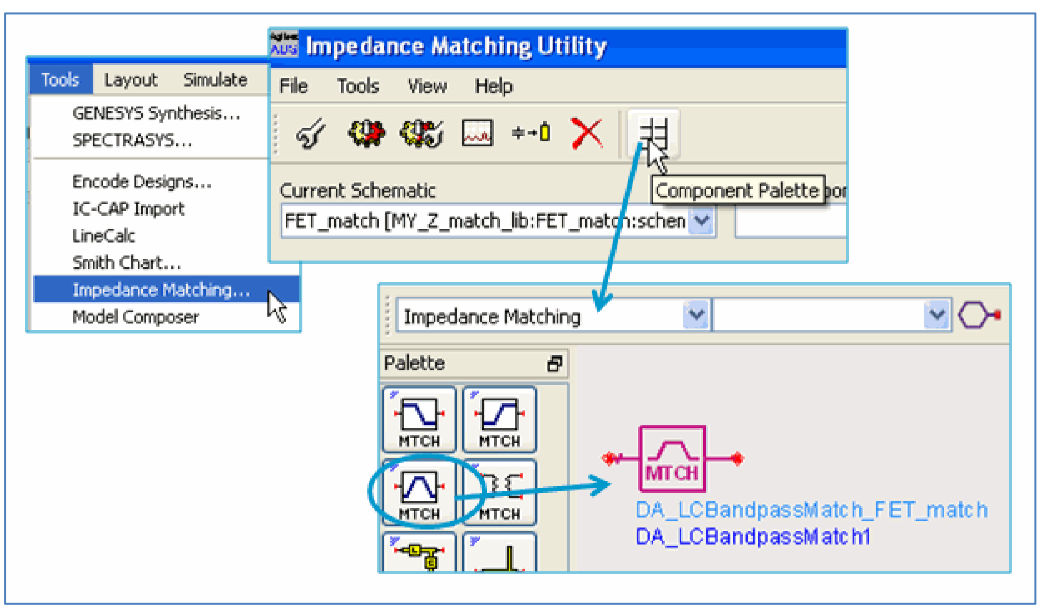
a. In the Main window, use the wizard create a **New Workspace** and name it: **My_Z_match** and select **only the ADS Analog/RF library** and only the ADS Analog/RF technology as shown here.



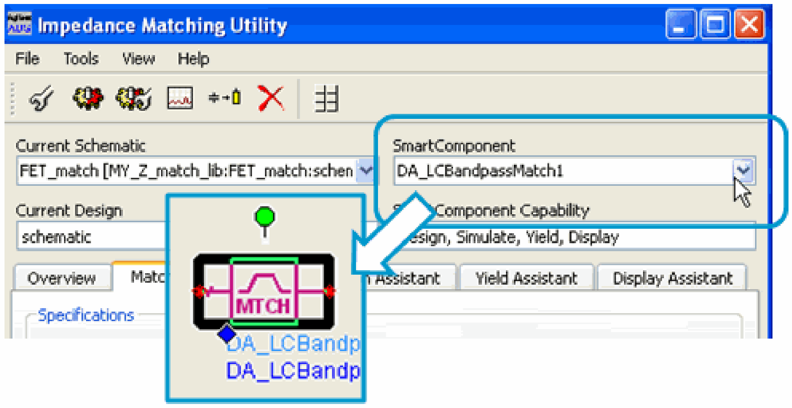
b. In the new workspace, create a new cell/schematic named: **FET_match**.



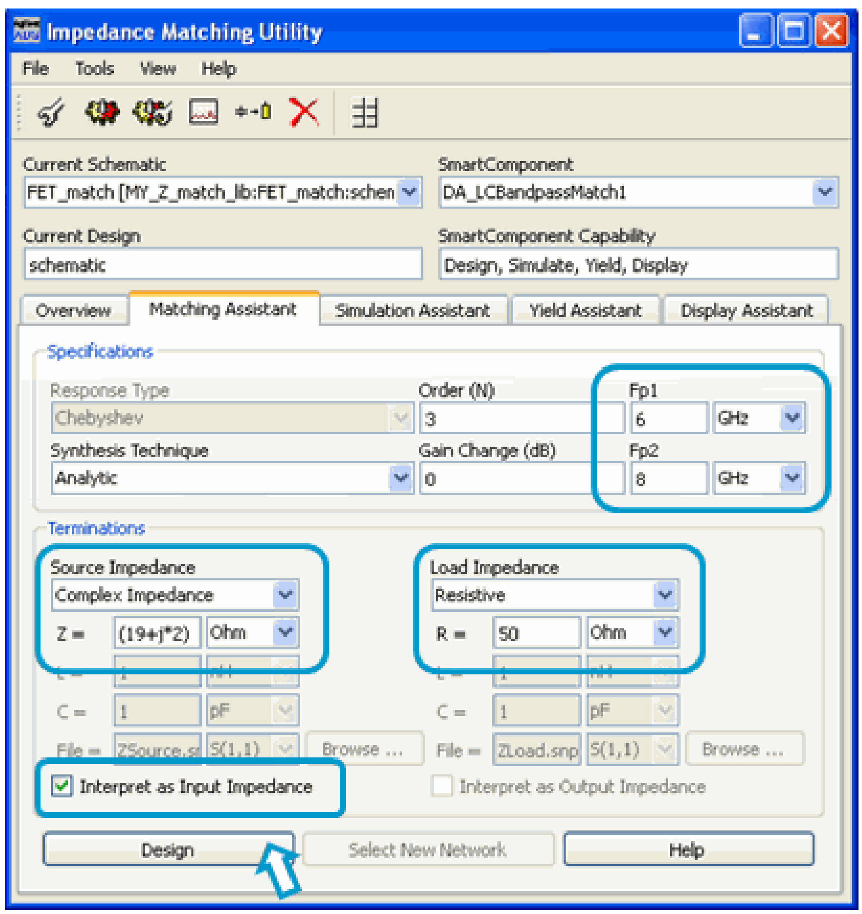
c. In the new schematic, select the command: **Tools > Impedance Matching**. When the Utility dialog appears, click on the **Component palette icon** and the Impedance Matching palette will appear in schematic. Then **insert** the **bandpass component**.



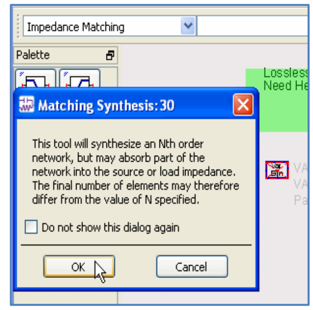
d. In the Impedance Matching Utility, use the pull down arrow to select the **SmartComponent: DA_LCBandpassMatch1** as shown here – it will be highlighted in schematic. This is necessary so that the utility recognizes it.



e. In the **Matching Assistant** tab, set the frequency range **Fp1=6 GHz** and **Fp2= 8 GHz** for this exercise. Set the Source Impedance to **Complex Impedance** and the **Z = (19+j*2)** - be sure to include the parentheses – and check the box to **Interpret as Input Impedance**. Finally, set the Load Impedance to **Resistive 50 Ohm** as shown here. Then click **Design** at the bottom.

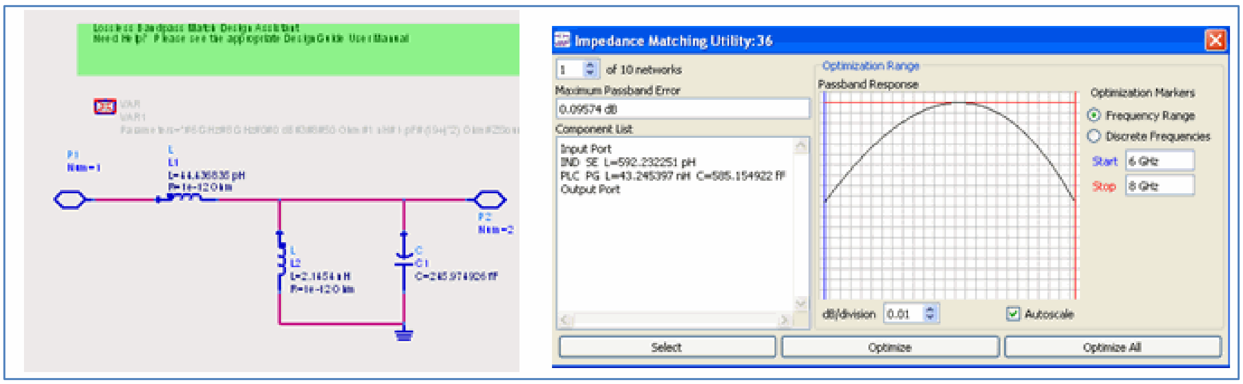


NOTE: 19+j*2 is an approximate value of load impedance from the load pull results (lab 1). This value will be seen by the amplifier output, and 50 ohms will be seen by the next stage. In other words, this is transforming the optimum load match to 50 ohms..



f. If the settings are correct, you should immediately see the tool working. Click **OK** to the prompt and you will see the network. It is a lower level hierarchy to the component you inserted.

g. Next, go ahead and click the **Optimize** button and you will see the results similar to those shown here. This is the network that will be connected to the amplifier output. Also, notice that you have some new cells added to the workspace.



h. **Save** and **close** the **schematic**s or other design windows.

You now know how to use this tool for creating matching networks. Also, this is the matching network that will be used for the FET amplifier in the next lab.

i. In the Main window, use the command **File > Recent Workspaces** and go back to the **My_FET_AMP workspace**.

END of LAB EXERCISE.