

ECE 404L: RF ELECTRONICS LABORATORY

DEPARTMENT OF ELECTRICAL AND COMPUTER ENGINEERING

MICHIGAN STATE UNIVERSITY

I. TITLE: Lab VII - Introduction to Ansoft Designer

II. PURPOSE:

Ansoft Designer is a suite of design tools that fully integrate high-frequency, physics-based electromagnetic simulation, modeling, and automation into an environment for circuit and system analysis.

As with any robust software package, it is best to start with known examples and re-create these in simulation. In this way we can see how our existing knowledge meshes with the tools developed. It can also clear up misconceptions.

The concepts covered are:

1. first time use of Designer;
2. schematic creation and parameter adjustment;
3. creating and editing graphs.

The examples covered are:

1. Lab VI network analyzer plots;
2. Input impedance of a coaxial cable with load;
3. S-parameters of a two-port.

III. BACKGROUND MATERIAL:

See Class notes and Lab VI.

IV. EQUIPMENT REQUIRED:

- 1 Lab PC

V. PARTS REQUIRED:

- 1 Storage Medium or M: drive

VI. LABORATORY PROCEDURE:

- A) Measuring a Long Lead Resistor with Ansoft Designer
1. In Windows goto **start -> All Programs -> Ansoft -> Designer SV2 -> Ansoft Designer** to start the Ansoft Designer program.
 2. A new design space will appear similar to Fig. 1. This space is set by checking all the boxes under **View**. On the left of the screen is located a project window and in the upper part of the project screen is a **Project1**.

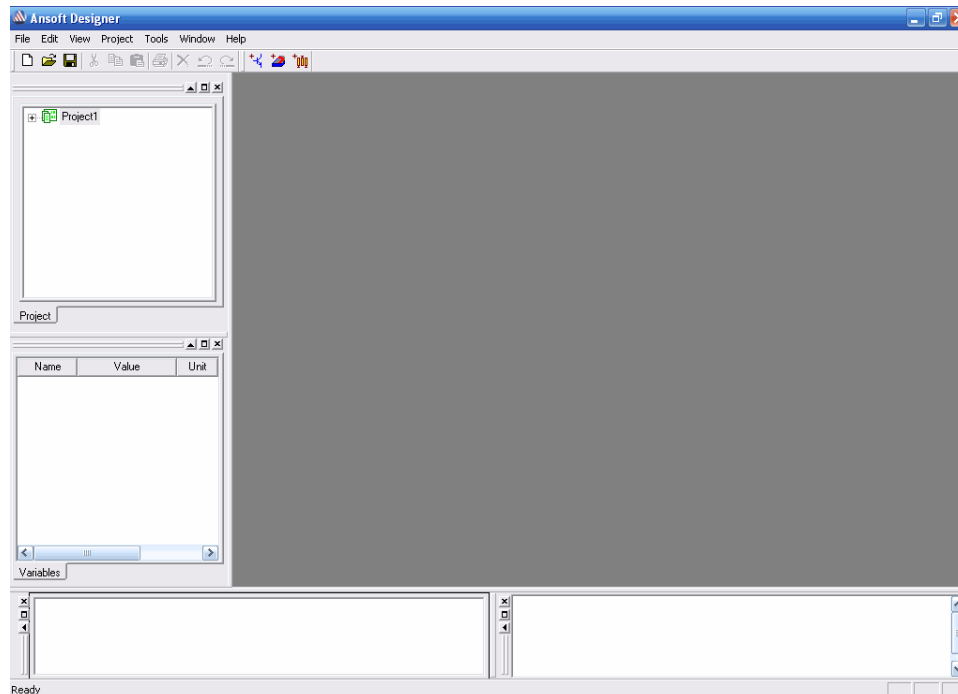


Figure 1. Ansoft Designer startup screen.

3. Right click on the **Project1** and select **Insert -> Insert Circuit Design ...**
4. A **Choose Layout Technology** window appears prompting for the type of printed circuit board that is used in this project, click **None**.
5. Components can now be placed onto the circuit schematic. To see the components, click on the **Components** tab located directly to the right of the **Project** tab. All of the available RF components are listed in the project window. Next an inductor will be placed.
6. Inductors are listed under lumped elements. Expand the **Lumped** elements listing and then expand **Inductors**. Fig. 2 shows how this

option should appear. Now click and drag the **IND: Inductor** onto the schematic area. Hit **Esc** on the keyboard to end the symbol on the cursor. If you make a mistake just click on the element and hit delete to erase.

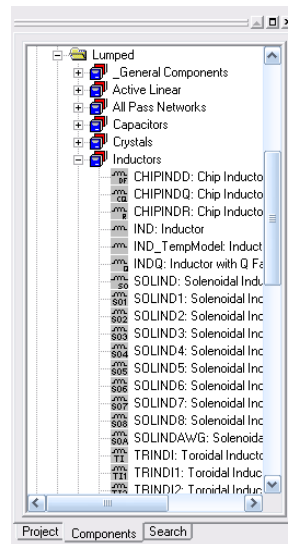


Figure 2. Selecting an inductor from the components list.

- Resistors are also listed under lumped elements. Return to the **Components** window and select **Resistors**. Click and drag the **RES: Resistor** onto the schematic area but don't release as yet. To rotate the resistor like that shown in Fig. 3, press the key R, place and hit **Esc**.

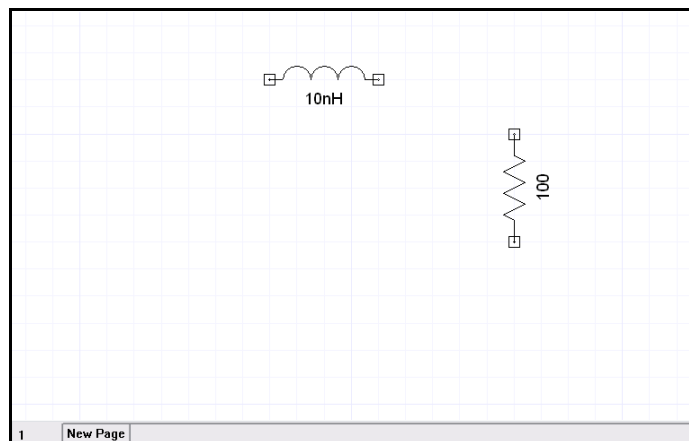



Figure 3. Inductor and resistor placement.

- Grounds  are located in the tool bar on the upper part of the screen. To place a ground, click the tool bar symbol and then click anywhere on the schematic to place it. *Do not click on this yet.*

9. Making connections in Designer is very simple, move the cursor on top of the terminal of the component. A cross symbol \times will appear. By clicking once, the cross symbol stays and a wire can now be dragged to another component terminal. Click onto the other component terminal to end the wire and make the connection. *Do no click on this yet.*
10. Now wire your inductor and resistor to look like Fig. 4. To create the long wire with no connection on the left of the inductor, start at the inductor and drag the wire to the desired length. Double click on the end to place the long wire.

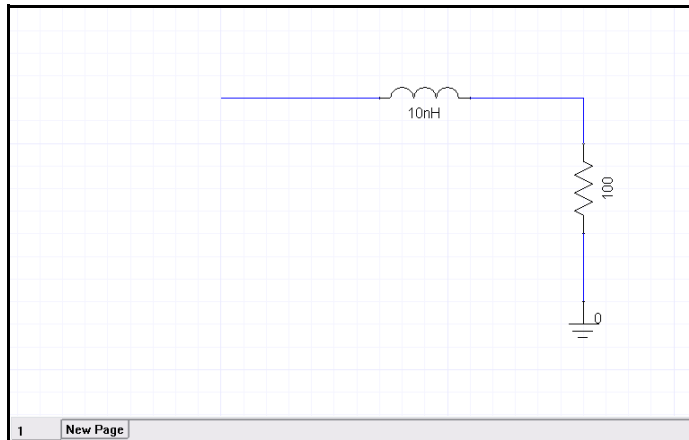


Figure 4. Wired components.

11. Proper values must be entered for each component. Double click on the inductor value to bring up the component editing menu. Change the inductor to 40 nH as shown in Fig.5.

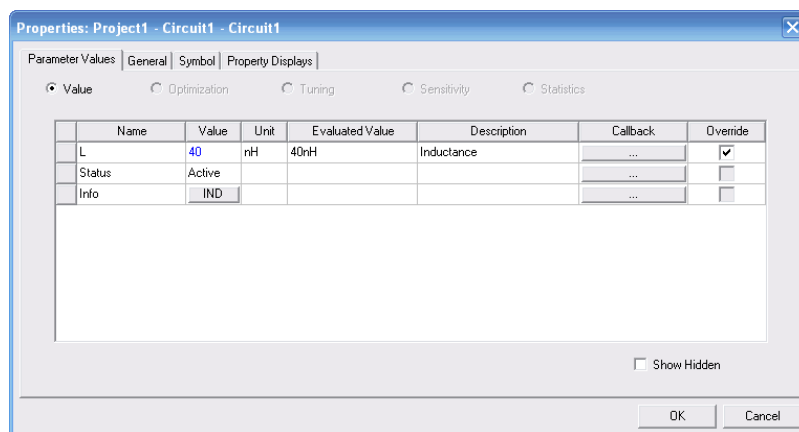



Figure 5. Editing the inductor's value.

12. Ansoft specifies sources (and loads) through the port symbol  which is located on the tool bar next to the ground symbol. Place a port at the left most end of the long wire from the inductor.
13. Double click the source port to open its editing window. Change the options to a **Microwave Port** under **Symbol** in the upper right hand corner of the editing box. Add a source definition by clicking the **Add ...** button in the Source Definition box and then type in **1 mW** (which is 0 dBm) for property **p**. The editing box will look like Fig. 6 when everything is entered correctly.

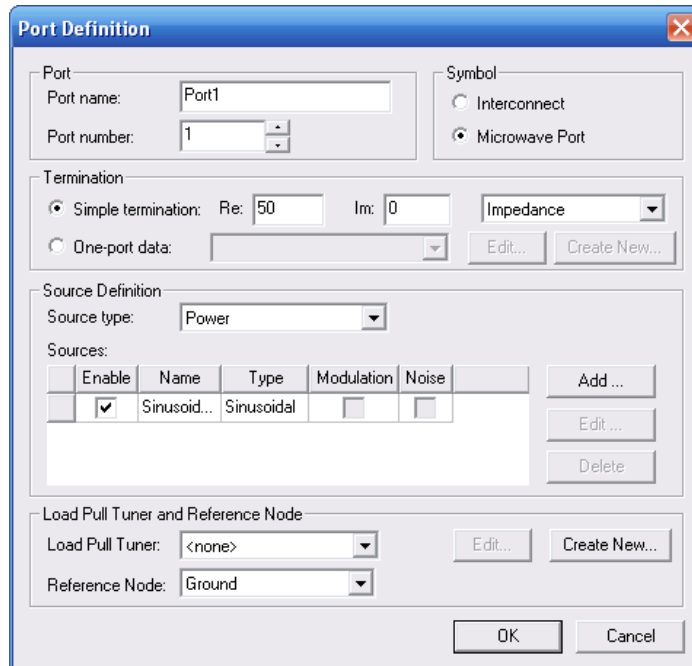


Figure 6. Port editing window.

14. Keep on clicking **OK** until the schematic shows again. The source port's symbol will have changed to an AC source with a series resistor as shown in Fig. 7. The term **PNUM=1** means microwave port number 1; **RZ=50Ohm** means the real part of the source impedance is 50 Ω ; and **IZ=0Ohm** means the imaginary part of the source impedance is 0 Ω .
15. Save the file to your M: drive or an external storage media device.
16. With the schematic built, an analysis profile must be defined which is similar to adding a .AC line in Pspice. To define the analysis, click the **Project** tab in the project window. Expand **Project1***. Expand the **Circuit1** list and right click the **Analysis** symbol. Then select **Add Solution Setup ...**. A new box appears, click **Next**. Click on **Add...** and then the **Edit...** button. Enter data as shown in Fig. 8. Then click on

Add >> and click **OK** when done. Click **Finish** to get back to the schematic.

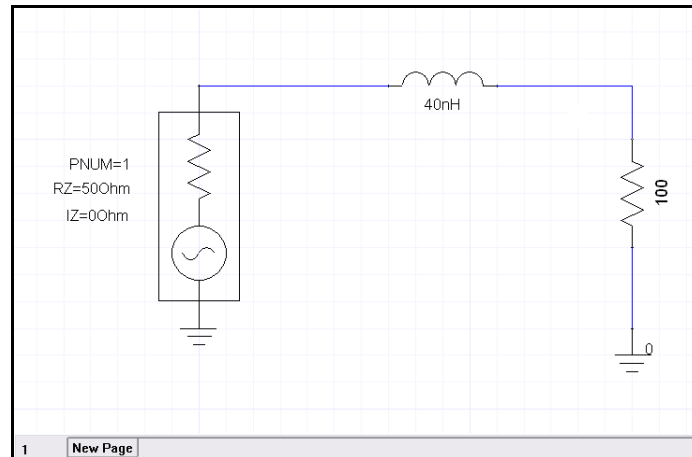


Figure 7. Final wired circuit with port defined.

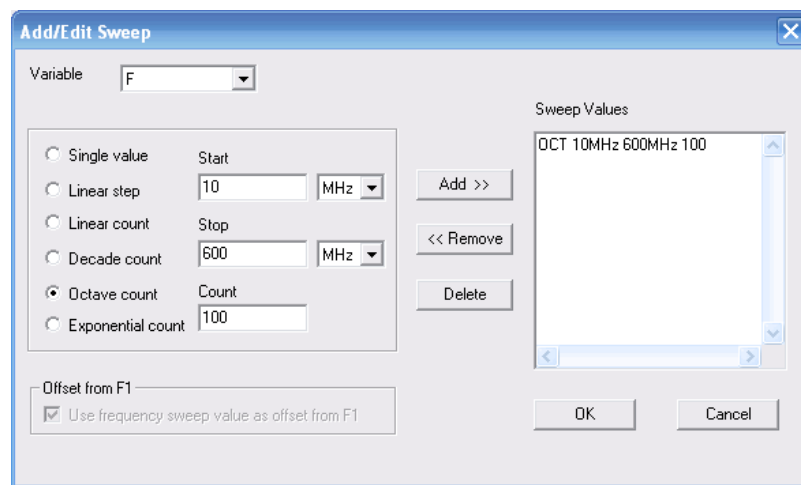




Figure 8. Analysis editor where a frequency span is entered.

17. The circuit can now be analyzed by clicking the analyze  button located on the upper toolbar.
18. After the analysis, the results have to be looked at by using the **Results** option in the project window. Left click on  **Results** to highlight it. Then right click and select **Create Report...** A new menu pops up. The desired response is a **Standard, Rectangular Plot**. Click **OK**.
19. The desired trace is the phase of S_{11} in degrees. As shown in Fig. 9, select **S11** and **ang**, then click **Add Trace**. Click **Done**.

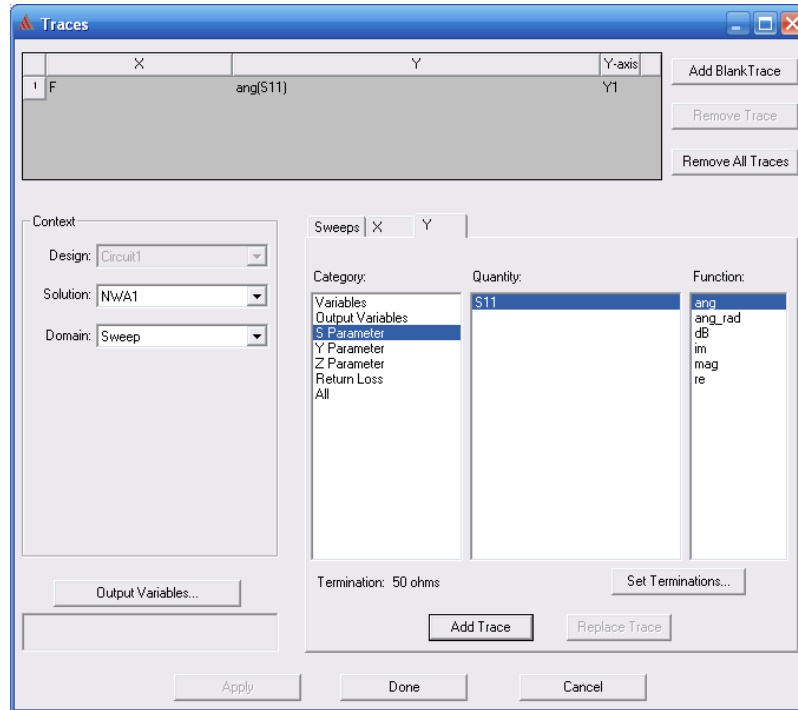


Figure 9. Inserting a trace.

20. The resulting graph is a plot of the angle of S_{11} from 10 MHz to 600 MHz. This should look similar to what you measured in Lab VI on the network analyzer for your resistor, R_S .
21. As with any plot, we may want to change the scales and/or the appearance of the plot. Suppose we do that by starting with changing the y-axis to display -10° to 40° . Double click anywhere on the y-axis and a Y Axis Properties box will appear. Click on **Scaling** and un-check **Autoscale**. As shown in Fig. 10, change the settings to Min. = -10, Max. = 40, Minor Divs. = 5, Major Divs.=10 and un-check **Show Minor Grid**.

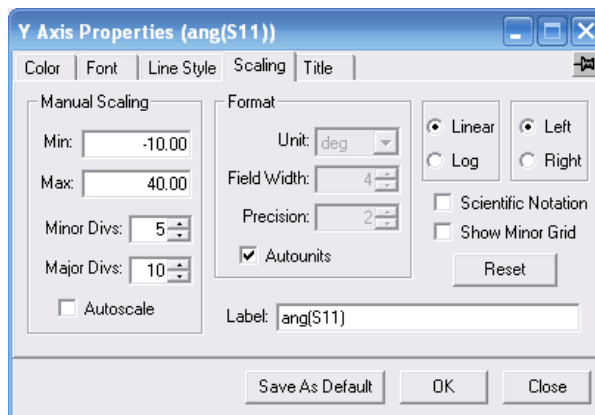


Figure 10. Changing the y-axis properties.

22. Likewise change the x-axis settings to Min. = 0, Max. = 600, Minor Divs. = 5, Major Divs.=10 and un-check **Show Minor Grid**.
23. Lastly the trace is thin and sometimes hard to see when printed in black and white. So double click on the trace of the angle of S11. Select **Line Styles** and change the width to **20**. The resulting plot is roughly set to the scales of the network analyzer (at least when this author did it) and is shown in Fig. 11.

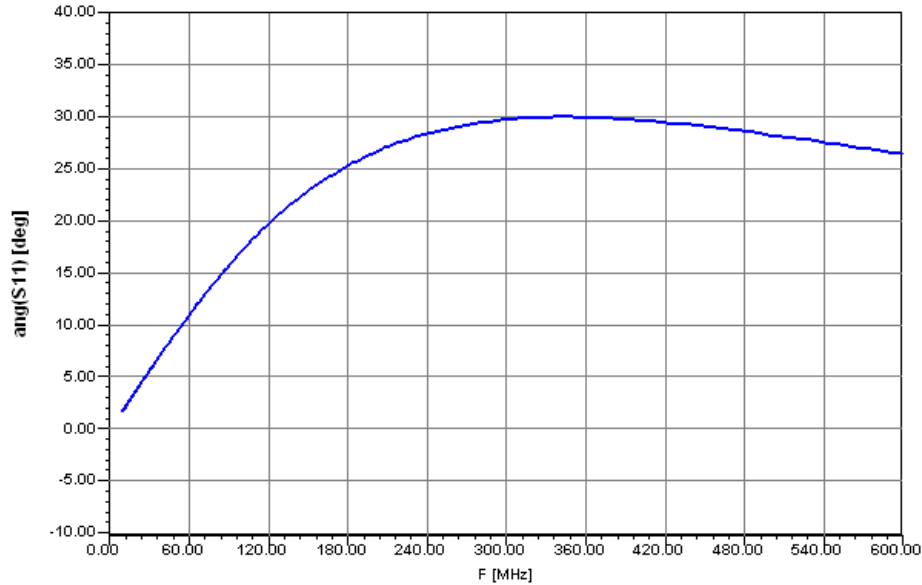



Figure 11. Re-creation of the network analyzer screen of Lab VI.

24. In Lab VI, we also displayed the value of the impedance of our resistor, R_S , versus frequency on a Smith Chart. We can do the same with Ansoft Designer. To this we need to create a new report.

In the project window, left click on  Results to highlight it. Then right click and select **Create Report....** A new menu pops up. The desired response is a **Standard, Smith Chart**. Click **OK**.

25. The desired trace is S_{11} . Select (if necessary) **S11** and **<none>**, then click **Add Trace**. Click **Done**.

The Smith Chart shown in Fig. 12 should appear. Note that the highlighted blue curve is hard to distinguish. Also the font size of the numbers inside the Smith Chart is too large. The following will describe how to fix this.

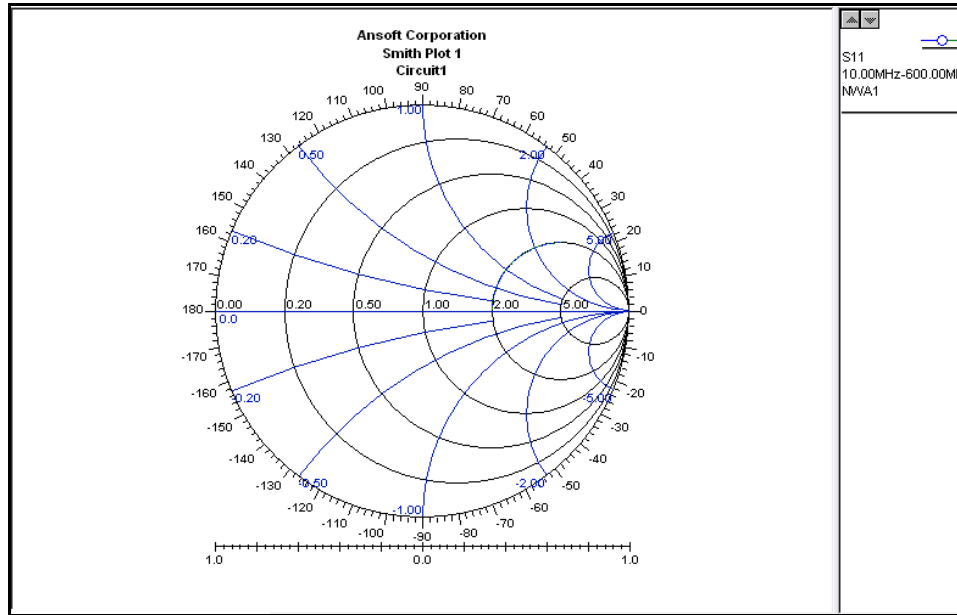


Figure 12. Re-creation of the Smith Chart of Lab VI.

26. Right click anywhere on the Smith Chart and some options will appear. Select **Zoom In**. To enlarge the area like that shown in Fig. 13, hold down the left button of the mouse, drag the square symbol over the desired area and release the left mouse button.

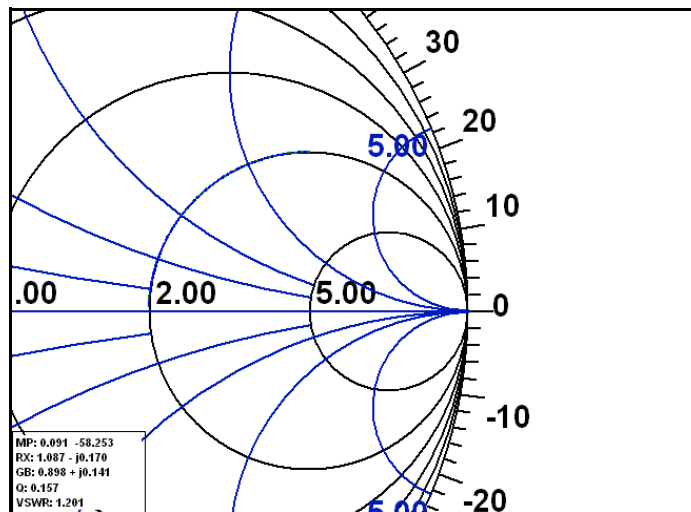


Figure 13. Zooming in on the Smith Chart.

27. Double click on the blue portion of the $2\ \Omega$ circle and change the width of the line to **20**. Right click again and select **Arrows**. Double click on the resistive circle font and change to 10 point. Likewise the for the

reactance arcs. Your plot should now look like Fig. 14.

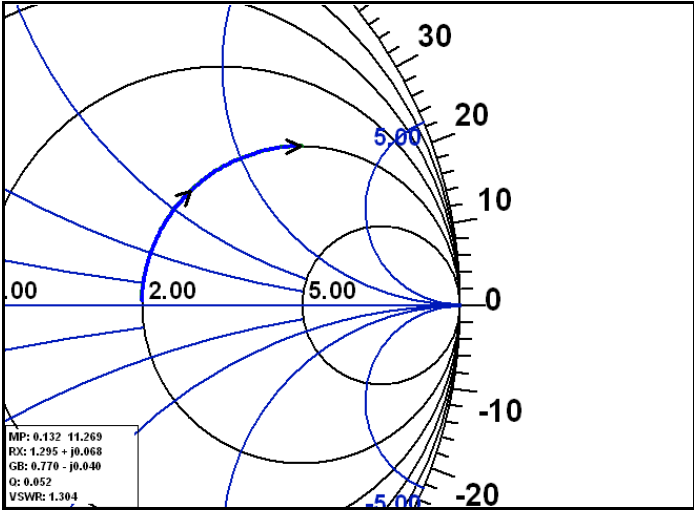


Figure 14. Modifying the Smith Chart.

- 28. Right click again and select **Data Marker**. The highlighted curve should turn to yellow with a marker circle. Right click again and select **Marker Scale >** and select **Imp Scale**. Move the marker with the arrow keys to approximately 300 MHz. The value of S_{11} is used to calculate the normalized impedance in rectangular form using Eqn. 2 of Lab VI assuming that Z_0 is 50Ω . The results are displayed in the upper right corner as shown in Fig. 15. Record. Multiply by 50 to denormalize.

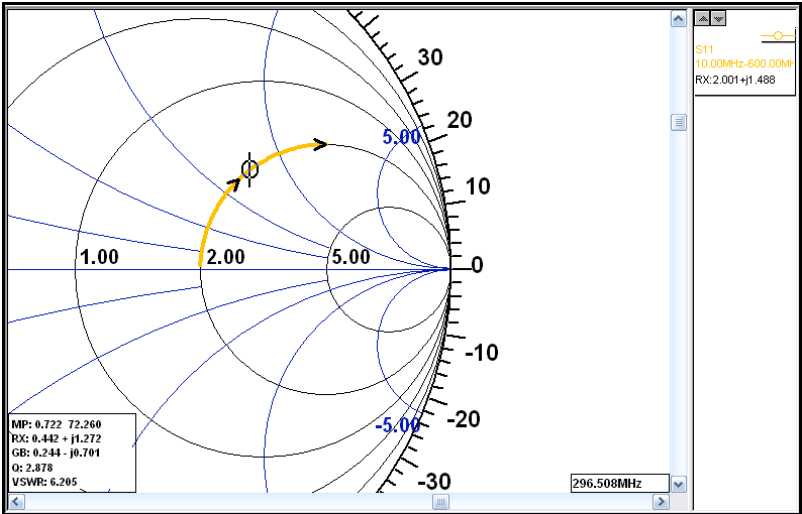


Figure 15. Reading the impedance at 296.508 MHz.

29. Using the denormalized value calculate the value of R and L using the actual frequency of the marker. Record in the Lab Report.
 30. Right click again and select **Exit Marker Mode** and **Fit All** to restore the Smith Chart to full size. Print a copy for each member of your lab group and include as indicated in the Lab Report.
 31. Save your project and close.
- B) Transmission Line
1. Start a new Project.
 2. On page 48 of the Ch. 3 class notes is an example of a 1 foot coaxial cable. This has been drawn in Fig. 16 using a cable which can be found in **Components** under **Coaxial Cable** by selecting **COAXK_REF: Coaxial Cable, K, w / Ref. Nodes**.

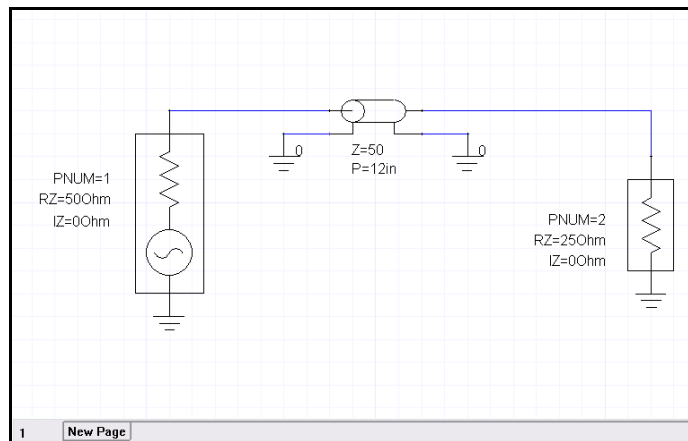


Figure 16. Schematic on page 48 of the class notes.

3. Layout this schematic and double click on the cable to edit its values. Set the value of P, which is the physical length, to 12 inches (avoid using feet - - there seems to be a “bug” with some feet settings.) We also need to set the value of K, which is the dielectric constant of the material filling the line, to the value for Teflon which is 2.21. To see more about the model and/or references, double click the button in the **Info** line under the column **Value** which has the name of the component.
4. Save and run your file to display the magnitude and angle of **ZIN** under **Z Parameters** for a frequency range of 1 MHz to 1 GHz with 1000 points per decade. Change the x-axis to a log scale.

This is the input impedance looking into the cable.

5. We can also mark points on our displayed graphs. Left click on the magnitude graph and it should turn yellow. Right click and select **Data Marker**. Moving the cursor with the mouse or the arrow keys, move the marker as close as you can to .1667 GHz. A left click will mark the data point with the x- and y-axis values. Your result should look like Fig. 17. Mark the other points indicated in Fig. 17.

Print a copy for each member of your lab group and include as indicated in the Lab Report.

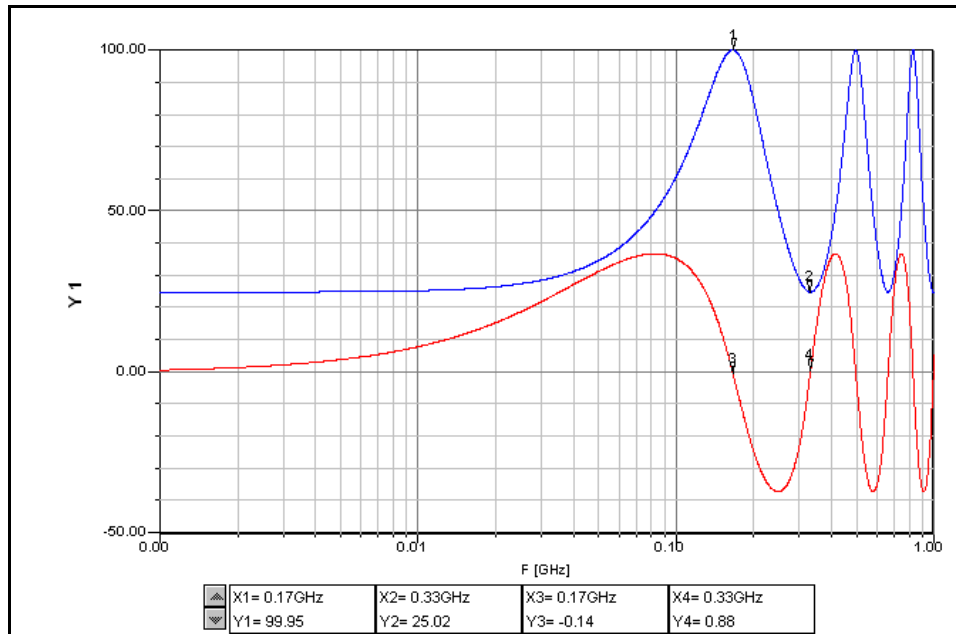


Figure 17. Z_{IN} (d) where $d = 12$ inches

6. Save your project and close.
- C) Scattering Parameters
1. Start a new Project.
 2. On pages 71 - 72 of the Ch. 3 class notes is an example of finding the S-parameters of a shunt 100Ω resistor in a balanced system. We will do this example in Designer.

Fig.18 shows the layout of this system. Here the port symbol is left as the default **Interconnect**. What this means is that this is a transmission line terminated in its characteristic impedance which can be specified by double clicking on the port symbol. The default value is 50Ω . Layout this schematic.

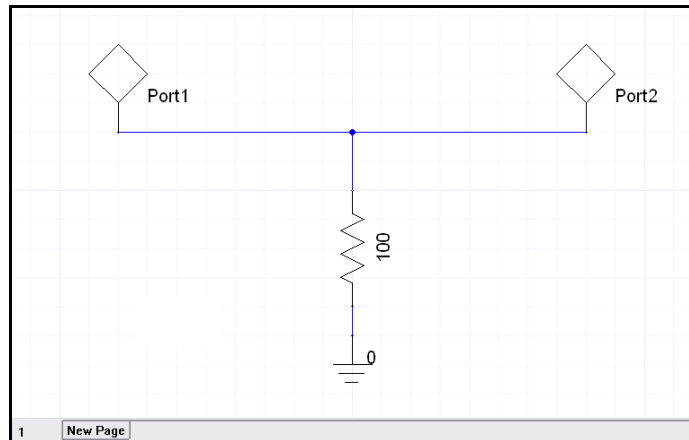


Figure 18. Schematic on page 71 of the class notes.

3. Save and run your file to display the magnitude and angle of the **S Parameters** for a frequency range of 1 MHz to 1 GHz with 1000 points per decade. Change your scales for the y-axis to match Figs. 19 and 20. If you have a hard time clicking on plot so that you can change the thickness of the plot, you might want to try double clicking on the symbol of the graph on the far right side.

Also shown in Figs. 19 and 20 is the option **Mark All Traces** which is found by right clicking anywhere on the graph. The vertical black cursor can be moved with the slide at the top of the graph. Do this to re-create Figs. 19 and 20. These results match the calculated results found in the class notes on page 72.

Print a copy of Figs. 19 and 20 for each member of your lab group and include as indicated in the Lab Report.

4. Change the characteristic impedance of port 1 to a value of $75\ \Omega$ and port 2 to $600\ \Omega$. If you display the plot shown in Fig. 19, there are crosses through each parameter on the right. The reason for this is that we have changed the schematic and our former plots are no longer valid. Re-run by clicking the **Analysis** button and the crosses should disappear.

Print a copy of the new Figs. 19 and 20 for each member of your lab group and include as indicated in the Lab Report.

5. Using your class notes, derive the s-parameters with $Z_{o1} = 75\ \Omega$ and $Z_{o2} = 600\ \Omega$. (We need to re-derive, since $Z_{o1} \neq Z_{o2}$) Record your calculations in the Lab Report.

How does this compare to the results you measured?

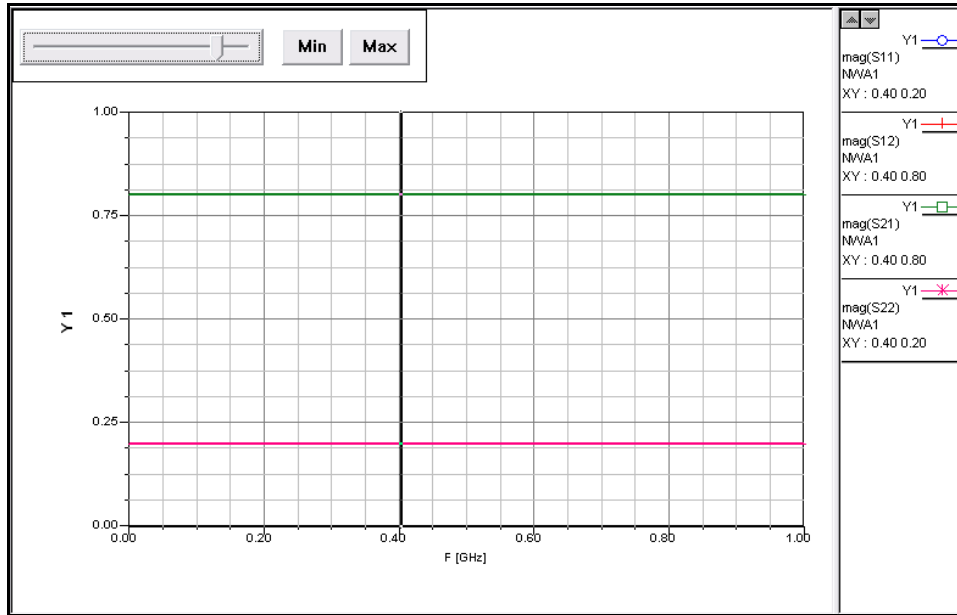


Figure 19. $|S_{11}|$, $|S_{12}|$, $|S_{21}|$ and $|S_{22}|$

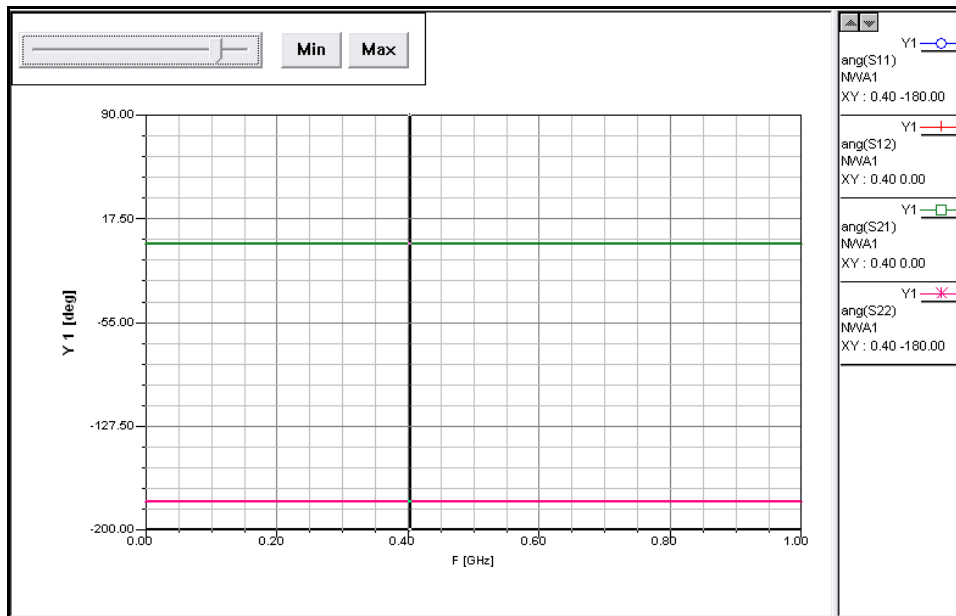


Figure 20. $\angle S_{11}$, $\angle S_{12}$, $\angle S_{21}$ and $\angle S_{22}$

6. Save your project and close.

D) Lab Report

Finish your Lab Report at home.

There is a free student version of Designer which has all the features but is limited to linear circuits. This lab was done using the student version which you can obtain from Ansoft at:

<http://www.ansoft.com/downloads.cfm>

Add the following sections to end of the Lab Report.

1. Try to re-create as much of your network analyzer lab results of Lab VI as possible using Designer. Don't use Designer's lossy models but instead use the lumped element models found in the class notes.

This due at your next lab meeting.

Lab Report

Lab VII - Introduction to Ansoft Designer

Name:

Partner:

Date:

Lab Section Number

Lab Station Number

Code of Ethics Declaration

All of the attached work was performed by our lab group as listed above. We did not obtain any information or data from any other group in this lab or any other lab section.

Signature

VI-A-28

Measured frequency = _____

Valued measured = _____ + j _____

Denormalized Valued = _____ + j _____

VI-A-29

R = _____

L = _____

Show work:

VI-A-30

Mark VI-A-30 on the top right side of your plot and attach as the next page.

VI-B-5

Mark VI-B-5 on the top right side of your plot and attach behind VI-A-30.

VI-C-3

Mark VI-C-3 on the top right side of your plot and attach behind VI-B-5.

VI-C-4

Mark VI-C-4 on the top right side of your plot and attach behind VI-C-3.

VI-C-5

s-parameter calculations:

$$|S_{11}| = \underline{\hspace{2cm}} \quad \angle S_{11} = \underline{\hspace{2cm}}$$

$$|S_{12}| = \underline{\hspace{2cm}} \quad \angle S_{12} = \underline{\hspace{2cm}}$$

$$|S_{21}| = \underline{\hspace{2cm}} \quad \angle S_{21} = \underline{\hspace{2cm}}$$

$$|S_{22}| = \underline{\hspace{2cm}} \quad \angle S_{22} = \underline{\hspace{2cm}}$$
