MONTANA STATE UNIVERSITY

Guide to *Agilent's Advanced Design System (ADS)*
Department of Electrical and Computer Engineering
Spring 2008

(last revised 1/12/08)

Summary

This is a tutorial on how to create projects, enter schematics, simulate, and view results using ADS.

The following lists the steps that will be covered in this tutorial:

Part 1: Setting up Directory Structures and Launching ADS
Part 2: Creating a Project
Part 3: Creating a Design and Entering schematics
Part 4: Simulating and Viewing Results
Part 5: Using Variables
Part 6: Using Subcircuits
Part 7: Performing AC Sweep Simulations
Part 8: Performing Parameter Sweeps

Part 1: Setting up Directory Structure and Launching ADS

1)　　Log onto the PC's in the computer lab on 6th Floor Cobleigh (COBH 624).

2)　　Setup your directory structure (if your first time using ADS)

- ADS uses *Project*s to hold all of your schematics and simulation results. One Project may contain numerous schematics (called Networks) and simulation results (called Datasets) in addition to other more advanced items. When you create a Project in ADS, you will enter a project name and ADS will create a new folder with the name you provided plus a suffix of "_prj".

- to keep things organized, you should create a folder called "ADS_Projects" in which all of your project subfolders will be created in by ADS. Since there may be permission and size issues with using your Z drive, it is recommended that you use your own personal thumb drive for all your files.

- Create a folder on your thumb drive called "ADS_Projects".

3)　　Start ADS using:

- Start – Programs – Advanced Design System 200x – Advanced Design System

- the first time you launch ADS, you may need to select the *mixed signal* option in a popup window.

Part 2: Creating a Project

1)　　Create a Project

- you'll need to provide a descriptive name, (i.e., EE461_Digital_System_Design)

- when you started ADS, a pop-up window appeared asking you
"What would you like to do?"

- click on the "Create a New Project" button:

- in the Name field, click "Browse" and browse to the folder: x:\ ADS_Projects

- when you click "OK" in the browse window, you will come back to the "New Project" window. You will now need to enter the name of your new project at the end of the path that resides in the "Name" field. You do NOT need to add the "_prj" to the name of your project, it will be done automatically. Type in the name "EE461_Digital_System_Design" (or whatever project name you've selected) at the end of the path making the entire path:

x: \ADS_Projects\EE461_Digital_System_Design

- leave the "Project Technology Files" setting as is, Click "OK"

Part 3: Creating a Design and Entering Schematics

    1)   Create a Design (ex, squarewave_TRAN)

        - after you create the Project, a "Schematic Wizard" will appear. Select the "No help needed" radio button and click "Next".

        - A schematic window will appear that is untitled. In this window you will enter your schematics and simulation setup options. First, let's save this schematic to give it a name.

            - File – Save Design

            - in the "Save As" dialog, fill in the name "squarewave_TRAN" and click "Save".

        A Note on Design Names

        - a design can be used for a particular simulation (TRAN, AC, DC) or as a sub-circuit in another design. It is a good idea to give some type of description in the filename so that you can easily determine what the Design is used for. I suggest the following postfixes"

                *_TRAN   - for transient simulation Designs
                *_AC   - for AC simulation Designs
                *_DC   - for DC simulation Designs
                *_SUBCKT   - for Designs that are sub-circuits to be used in other designs.

    2)   Enter a Schematic (A simple RC Circuit)

        - In the upper left of the schematic window, you'll see a drop-down menu of all of the built-in libraries within ADS. Selecting a library from this pull-down will populate the left-hand side of the schematic window with a set of icons that represent each part available in that library.

        - Enter a Resistor:

            - Choose the "Lumped-Components" library
            - Click on the "R" component
            - a resistor attaches to your mouse, drag and drop into the schematic.
              You can keep clicking to add multiple resistors or hit [ESC] to end.

            - You can click on the resistor to select it and then Rotate it using CNTL-R

            - You can change the value of the resistor by clicking on the value and typing
              or you can double click on the resistor and alter its properties.

            - Change the value of the resistor to 75 Ohms

        - Enter a Capacitor

            - Choose the "Lumped-Components" library
            - Click on the "C" component

- enter the capacitor and rotate it so that the positive terminal is on the top
- change its value to 10pF.

- Enter a Pulse Source

    - Choose the "Sources-Time Domain" library
    - Click on the "Pulse" component
    - enter the pulse source and then set its parameters to:

Vlow	= 0 V	(default)
Vhigh	= 1 V	(default)
Delay	= 0 nsec	(default)
Edge	= Linear	(default)
Rise	= 0 nsec	
Fall	= 0 nsec	
Width	= 2.5 nsec	
Period	= 5 nsec	

- Enter a GROUND

    - along the top of the schematic window, click on the "Insert GROUND" icon.
    - click in the schematic window to place the ground.
    - hit [ESC] to end.

- Position the parts

    - if you click and hold on a symbol, it will move the part.
    - position all of the parts so that they can easily be wired up.

- Enter wires

    - along the top of the schematic window, click on the "Insert Wire" icon
    - when you click on the terminal of a part, it will start a wire. You can drag the
      wire to the terminal of another part and click to automatically connect it. You can
      then reposition the wire as you want it by click-hold-drag.
    - hit [ESC] to end.

- Enter Wire Names

    - along the top of the schematic window, click on the "Insert Wire/Pin Label" icon
    - type in the wire name you want in the dialog box that appears, DON'T CLICK
      CLOSE. With the dialog box still open and your wire name entered, click on a
      wire in the schematic to give it the name you typed.
    - repeat this process by going back to the dialog and typing in a different name.
    - Label the input wire of the RC circuit "Vin" and the output "Vout"
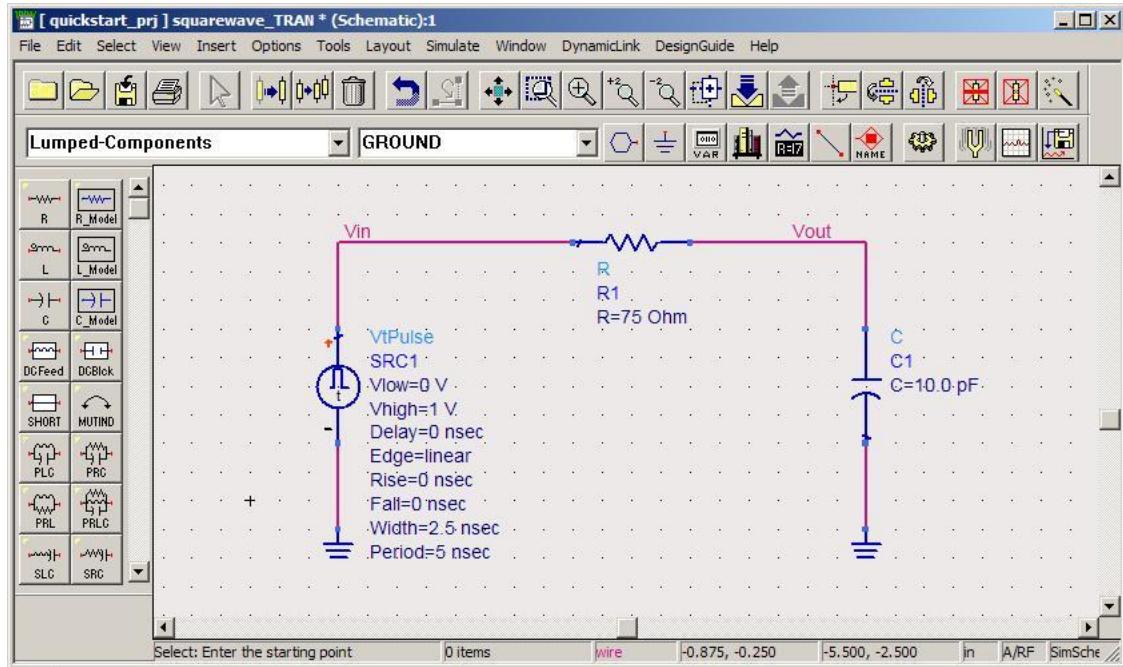
3)   Enter the Simulation Options Element

- the way we enter the simulation setup options is to instantiate a Simulation Component in the schematic

- Enter a Transient Simulation

      - Choose the "Simulation-Transient" library
      - Click on the "Trans" cell
      - A Transient component attaches to your mouse, drag and drop into the schematic.

      - Double click on the Transient component and set the following parameters

          - Start Time = 0 nsec
          - Stop Time = 20 nsec
          - Max time step = 10ps

      - click "OK"

- Save your design

Part 4: Simulating and Viewing Results

1)   Simulate the Design and View the Results

- in the Schematic window, click on the "Simulate" icon

- you'll see the progress of the simulation in the *hpeesofsim* window. When the simulation is complete, a Data window will appear

- The Data window is where we can plot our results.

- Along the left-hand side of the Data window, click on the "Rectangular Plot" icon. A plot will attach to your mouse. Click once to drop the plot in your Data window.

- A dialog will come up that asks what traces to plot. In order to plot a node, you select it in the list on the left (Datasets) and click the "Add" button in order to move it to the list on the right (Traces).

- Add "Vin" and "Vout" to the Traces' list and click "OK".

- You can edit the plot further by doubling clicking on it. You can then add a title, change the axis scale, add more traces, and change the colors.

- You can also enter markers using the "Marker – New" dropdown menu. Once you have chosen to add a marker, you click on the trace to which the marker is to be attached.

- You can also perform a measurement between the markers using the "Marker – Delta Mode On". You then select the reference marker for the measurement. You can click-and-drag the markers to the desired location. Save your Dataset window.
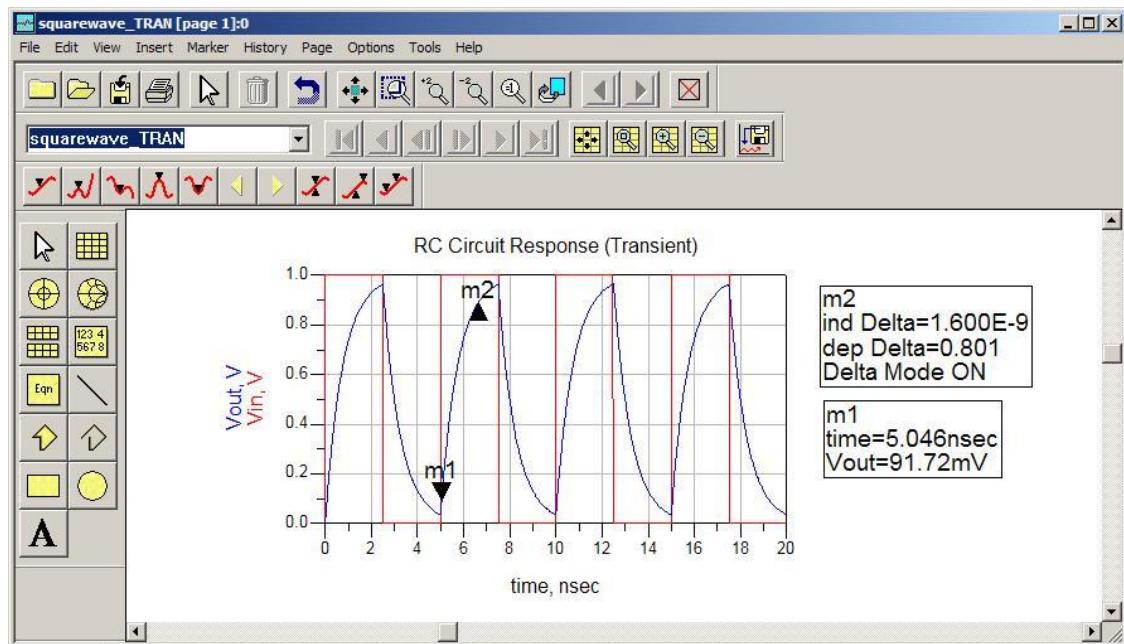
Part 5: Using Variables

    1)   Use the VAR Component to Enter New Variables

        - a variable is a way to pass in a value to a component using a symbolic name (i.e., Cload, Rload..). This is done by entering the VAR component from the "Data Items" library and then entering a list of parameter names and values.

        - Enter the VAR component

                - Choose the "Data Items" library
                - Click on the "VAR" component
                - a VAR item attaches to your mouse, drag and drop into the schematic.

                You can keep clicking to add multiple VAR components, or hit [ESC] to end.

        - Add Variables

                - Double click on the VAR item. You can enter parameter names and values. If you click the "ADD" button, it will add a new parameter to the list. If you click the "Apply" button, it will overwrite the selected parameter. Use the "Cut" button to delete parameters.

                - Setup variables for Vhi, Vlo, trise, tdelay, Rload, and Cload

        - Use the Variables in your components

                - Replace the values in each component with the unique variable name you created
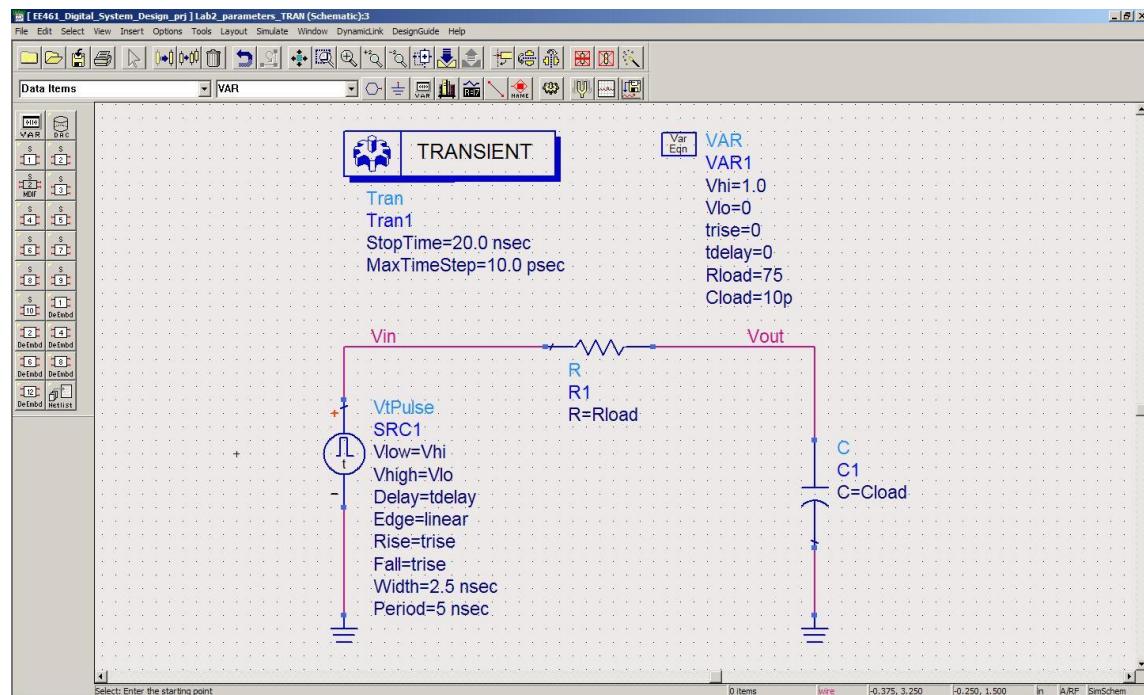
Part 6: Using Subcircuits

    1)   Basics of a Subcircuit

          - a subcircuit is a schematic/network that has an associated Symbol View with it.

          - you save this to a component library and then it is accessible to other schematics

          - you can pass parameters into a Subcircuit.

          - you use "Ports" to pass signals into and out of the subcircuit

    2)   Enter a Gaussian Step Source Schematic

          - we want to create a general purpose, step that only has spectral components in the range of frequencies that we want. This can be accomplished by applying a multi-pole, Gaussian filter to the output of an ideal step.

          - this step sources will be used as a Subcircuit in other designs.

          - enter the following schematic and variables. Note that we use a "Port" for the output of our subcircuit.
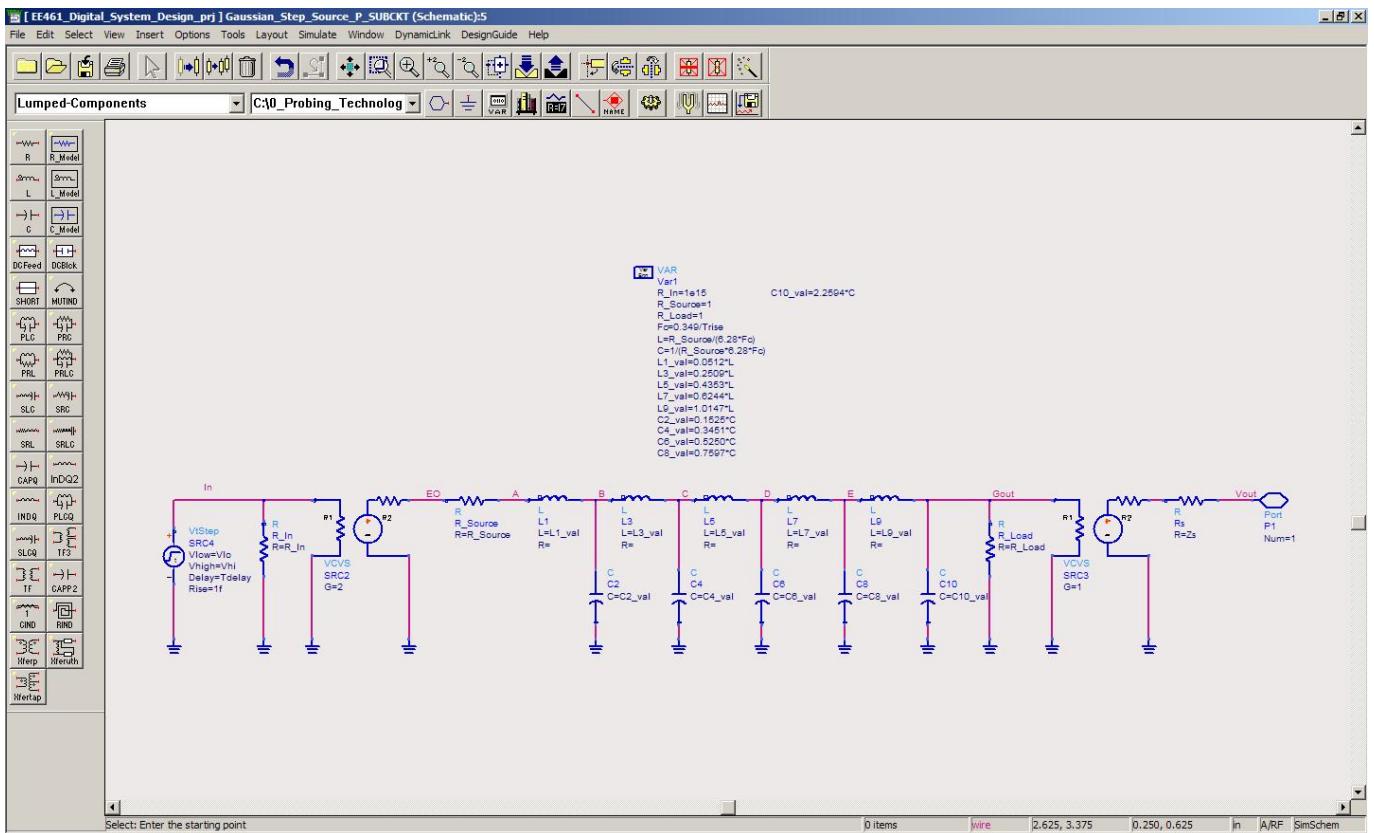
3) Enter a Symbol for the schematic

   - In the schematic window, use the pull-down menu:

      - View –Create/Edit Schematic Symbol

      The first time you do this, ADS will ask if you want to automatically create a symbol. Say yes. Once the symbol is created, you can alter the shapes and add text.

      Create a symbol that looks like this:
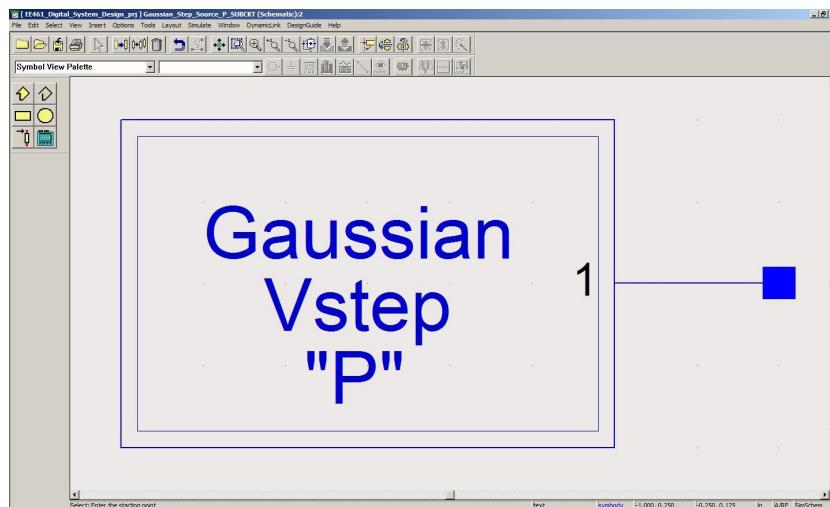
   - Once you are done with the symbol, switch back to the schematic view using:

      - View – Create/Edit Schematic

4) Enter the parameters that will be passed to the subcircuit from the calling entity:

   - Use the pull-down menus to bring up the "Design Parameters" dialog:

      - File – Design Parameters

      - Click on the "Parameters" tab

      - On this tab, you can enter parameters and also give default values for each. When the symbol is instantiated, the user can edit the parameter values on the symbol. Enter the following parameter:

$$
\begin{array}{ll}
\text{Zs} & = 50 \\
\text{Vhi} & = 1 \\
\text{Vlo} & = 0 \\
\text{Tdelay} & = 100\text{p} \\
\text{Trise} & = 35\text{p}
\end{array}
$$

      - Go back to the "General" tab, but DON"T CLICK **OK** YET!!!!
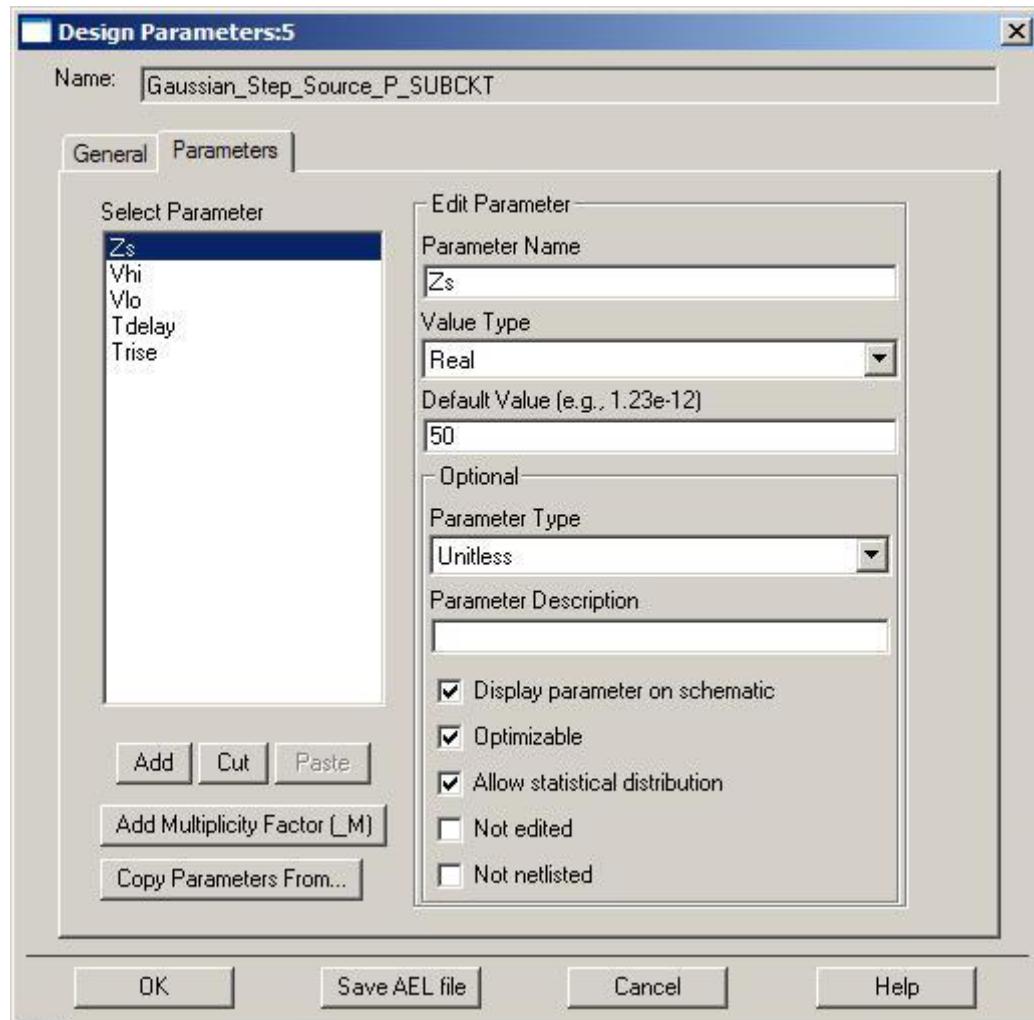
5) Save the subcircuit to a Component Library:

- On the General tab, enter a unique Library Name (i.e., *your_name_ADS_Library*)

- Now click on "Save AEL File", this saves the subcircuit to a component library so that it can be instantiated each time.

NOTE: This step may need to be done every time you copy your project to a new computer. No information is lost since your actual subcircuit is a Network within your project directory.
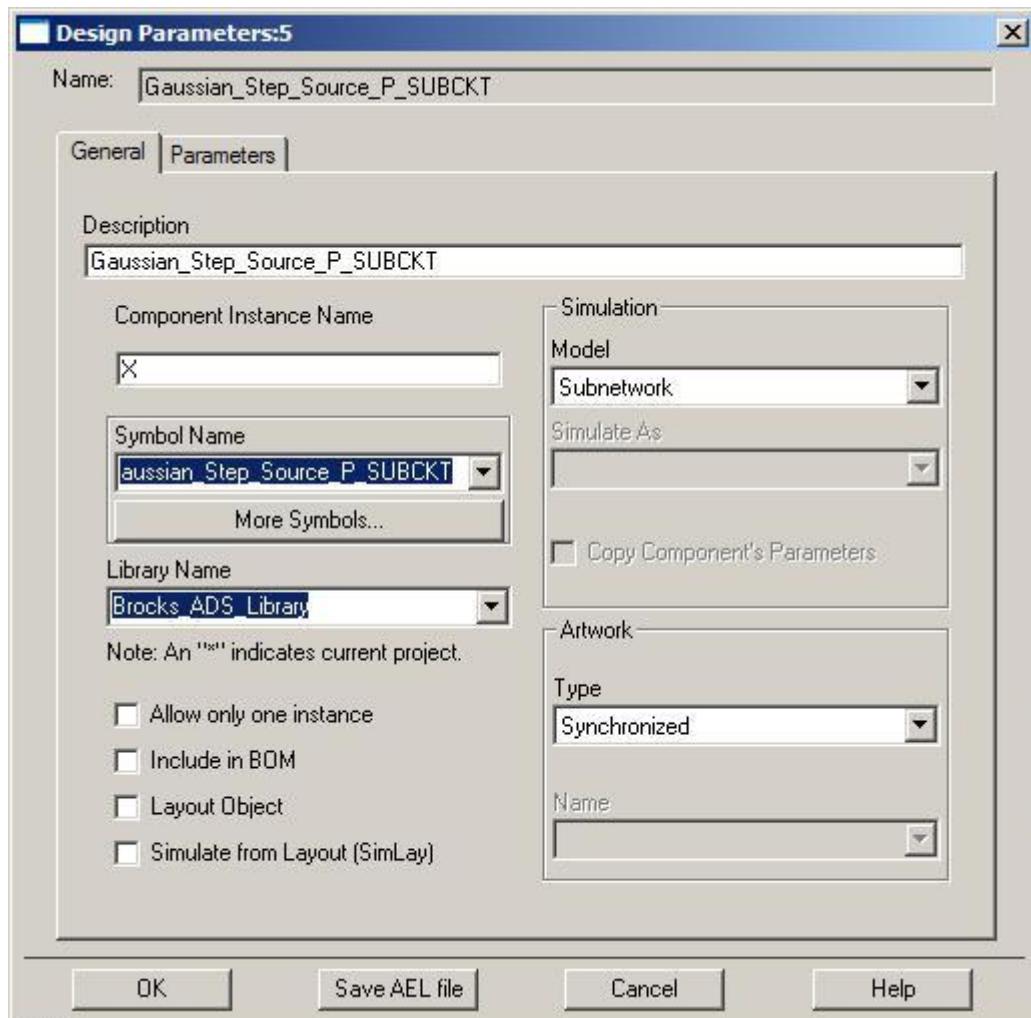
6) Instantiate your subcircuit in a Test Schematic

    - copy your RC circuit schematic to a new design called "subcircuit_test_TRAN"

    - in the schematic window, instantiate your subcircuit using the pull-down menu:

        - Insert – Component – Component Library:

        - The Library you created will be at the bottom of the "Libraries" List. If you click on your library once, you will see your subcircuit show up in the "Components" list. If you click once on your subcircuit name in the Components list, it will attach to your mouse and you can drop it into your schematic.

        - use your new Gaussian Step Subcircuit instead of the ideal *pulse* component in your test circuit. Use your variables that you had previously setup instead of the default values of your subcircuit.
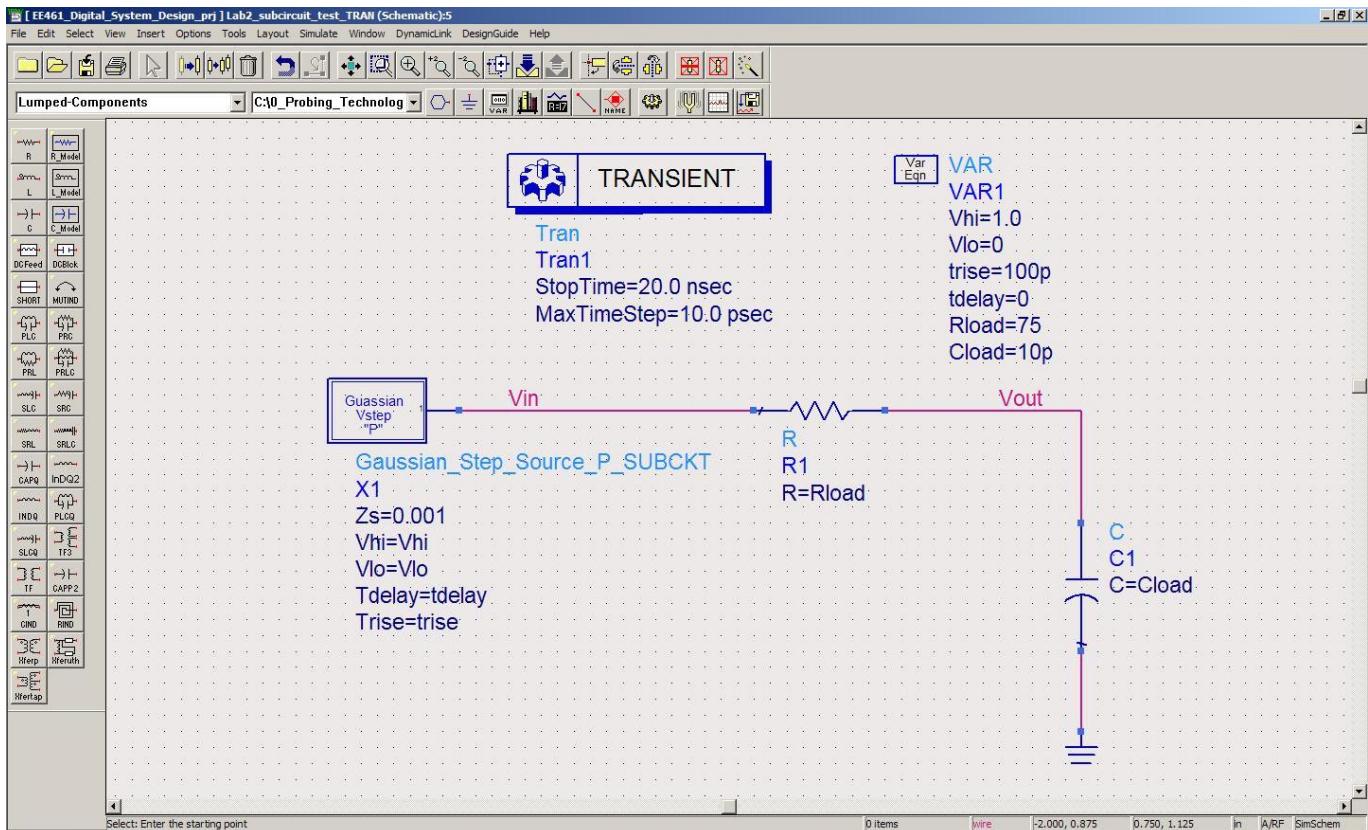
- Test your circuit by running a transient simulation. Change the risetime to 100ps. Run the simulation and place markers so that you can verify that the Gaussian step coming out of your subcircuit is 100ps. The output of the RC circuit can be predicated using the sum-of-squares technique.
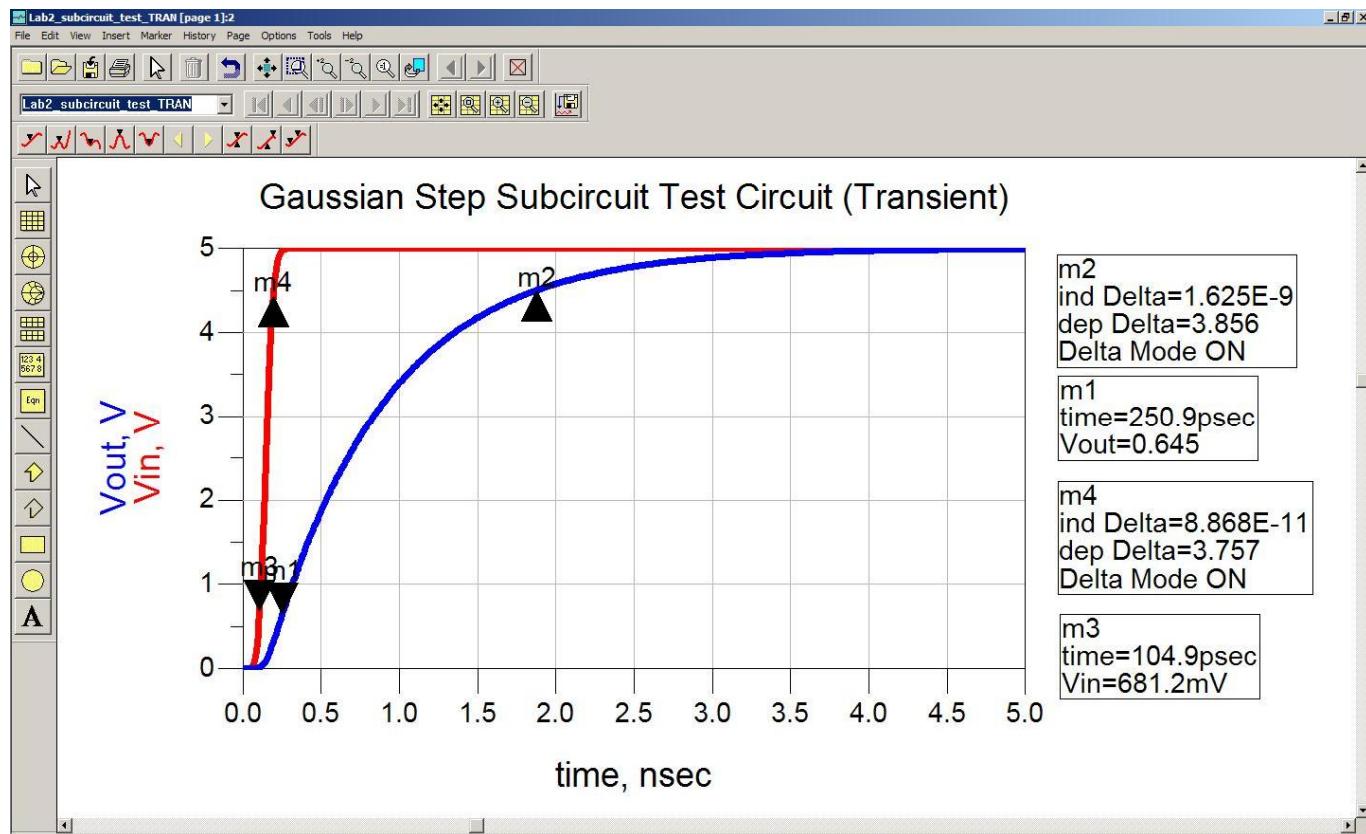
Part 7: Performing AC Sweep Simulations

1)   Create a new design for your AC sweep

- Copy your series RC circuit from previous labs and name it with a postfix "_AC".

2)   Enter an AC Current Source

- In the "Sources – Freq Domain" library, enter the "I_AC" current sources.

- orient the current source so that it drives current into the series RC circuit

- give the current sources a magnitude of 1 (i.e., polar(1,0) A)

- name the node connecting the current sources to the RC circuit "Zin_Series_RC"

3)   Enter the AC simulation options component

- In the "Simulation-AC" library, enter the "AC" component

Note: you will need to delete the TRAN component that was in the design when you copied it over.

- Double click on the AC Simulation component and setup as follows:

Sweep Type   = Log
Start   = 1 kHz
Stop   = 1 GHz
Pts/decade   = 100
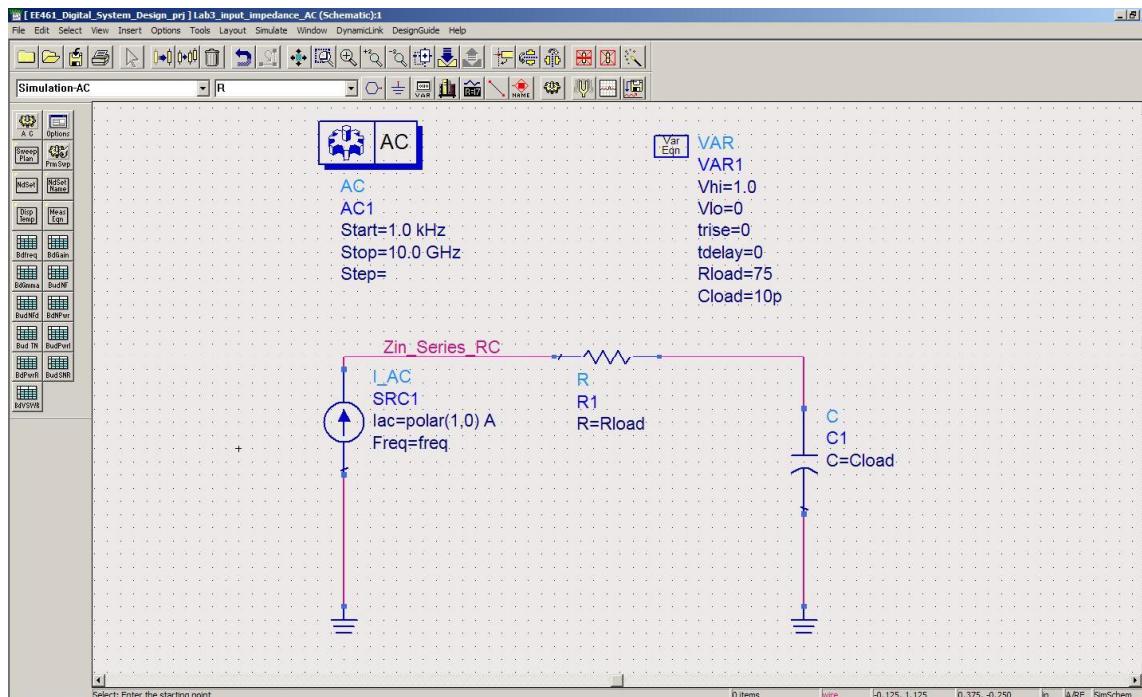
4)   Run the Simulation and View the results

- Simulate the circuit just as in the Transient simulation by clicking the "Simulate" button.

- View the results. When you add traces to the plot, you will be asked how to display the numbers (remember they are complex in the frequency domain). Choose Magnitude.

- On the plot, you will need to change the scale to "LOG". Double click on the plot and change both the X-axis and Y-axis to "LOG".
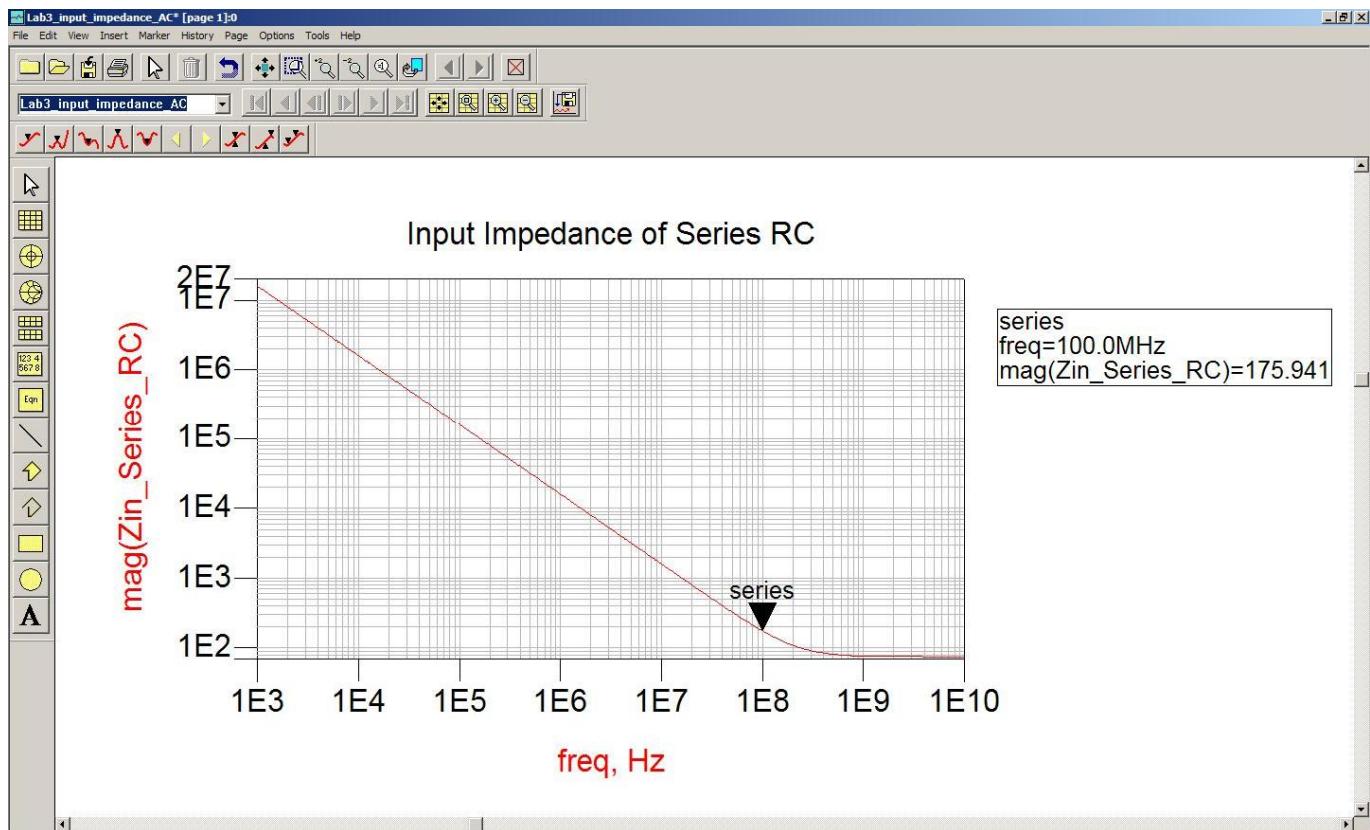
Part 8: Performing Parameter Sweeps

    1)   Add the Parameter Sweep Component

        - In the "Simulation – Transient" library, add the "PrmSwp" component to your schematic

        - Double click on the "Parameter Sweep" component to setup the sweep:

            - On the "Sweep" tab, enter the variable name of the parameter you are going to sweep. Note that whatever you are going to sweep needs to be setup as a variable using the VAR component. We'll sweep "Cload"

            - Setup the sweep to start at 5p and stop at 15p with a step size of 5p. This will run three simulations with Cload = 5pF, 10pF, and 15pF.

            - on the "Simulation" tab, you need to enter the instance name of the simulation component that will be swept. The default for the "simulation-AC" block is "AC1".
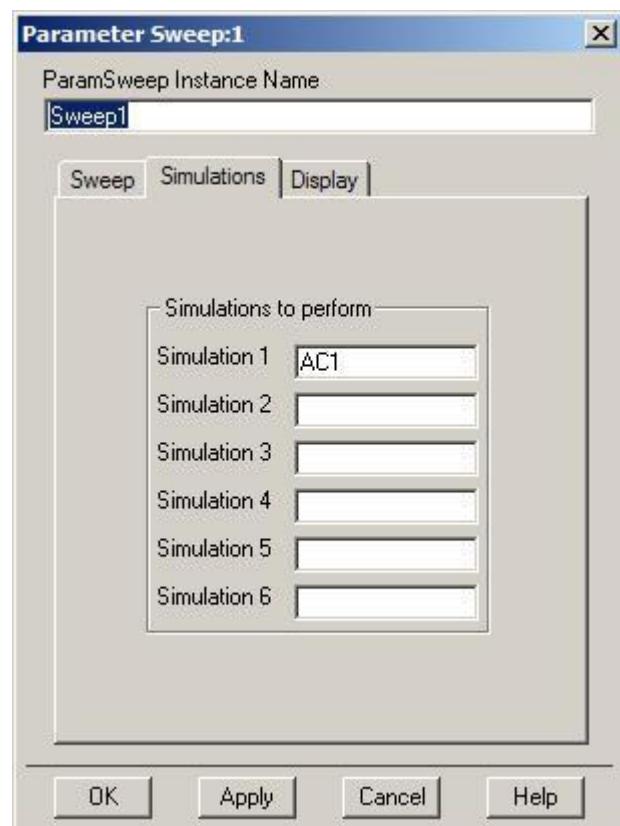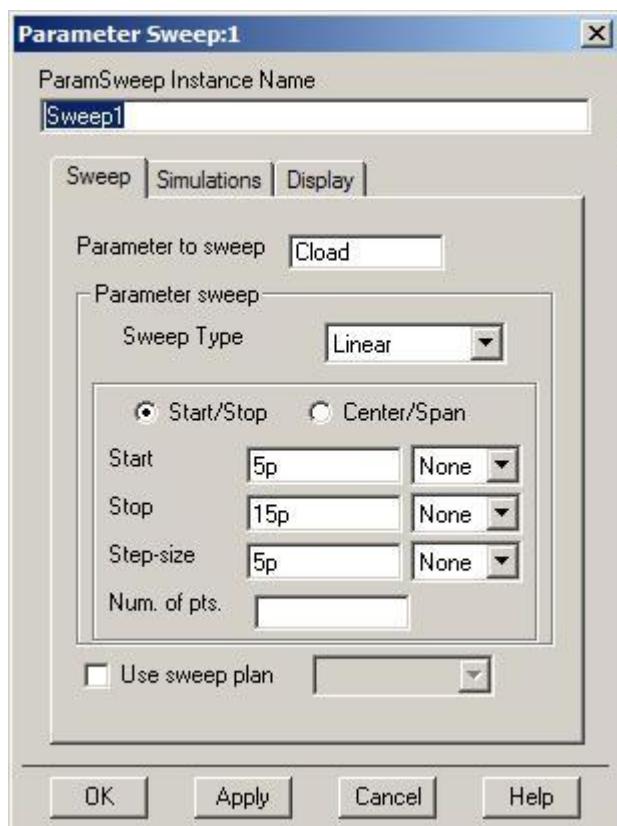
2) Run the simulation and view the results.

- you should see 3 results plotted on the same graph.

Input Impedance of Series RC

series
freq=100.0MHz
mag(Zin_Series_RC)=327.026
Cload=5.000000E-12