# Smith Chart, Custom Plotting, and TLine Transient Response

ADS Session
2

## Introduction

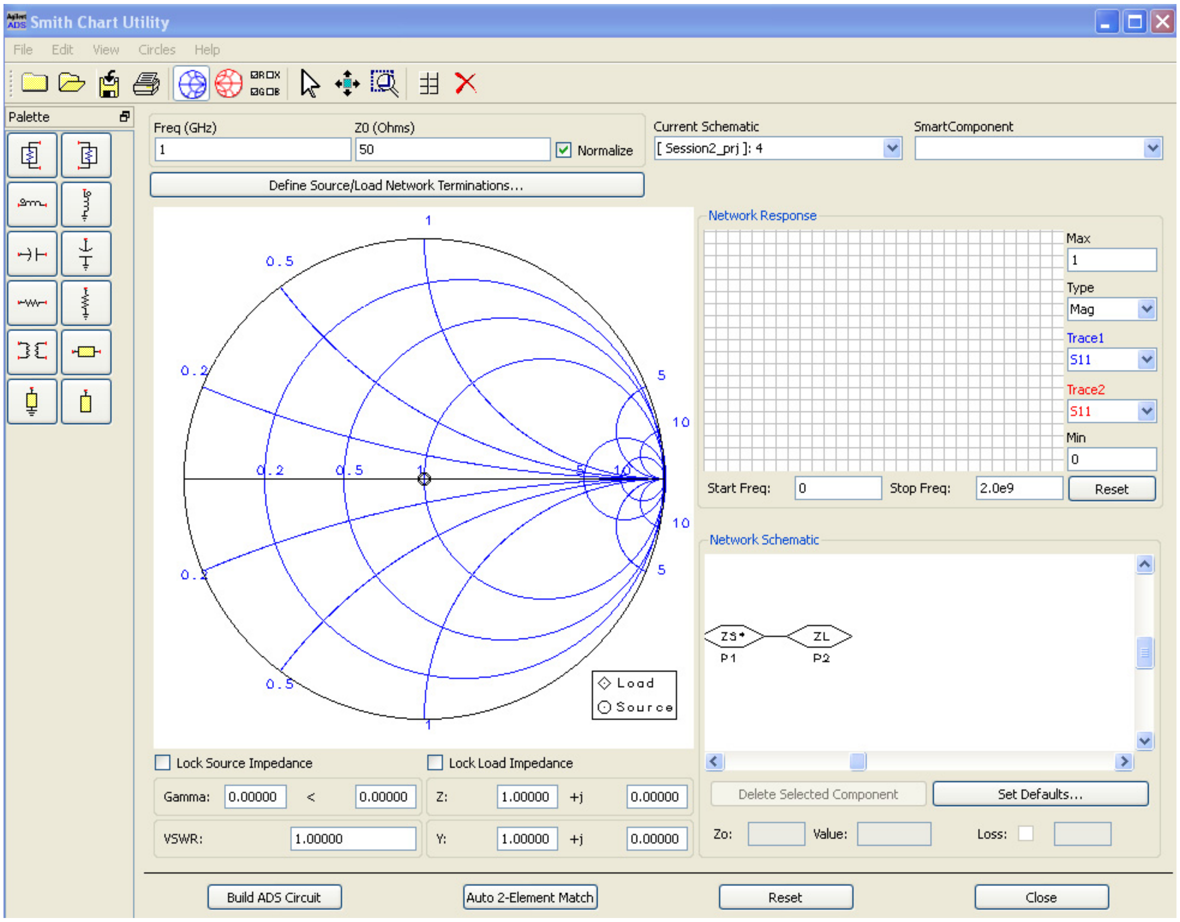
In this second session we consider more details of ADS for specific types of measurements. In particular we will address the needs of Problem Set #2, which reviews the Smith Chart, more transmission line frequency domain modeling, including lossy transmission lines, and transmission line transient analysis.

ADS support for the above topics will have us explore the Smith Chart Utility, using equations and custom plot functions for plot windows, AC Simulation and the use of the *Power Probe* schematic component from the Probe Components palette, and finally Transient Simulation and the use of the Sources-Time Domain palette.

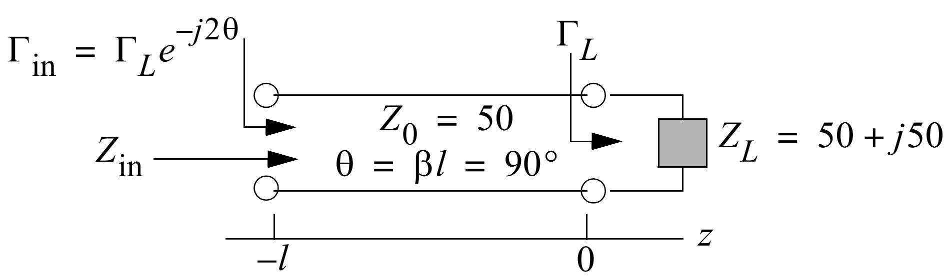
## Smith Chart Utility

- The Smith chart utility available in ADS is available under the Tools menu when a schematic window is open
- When it is first opened the window appears with just a 50Ω

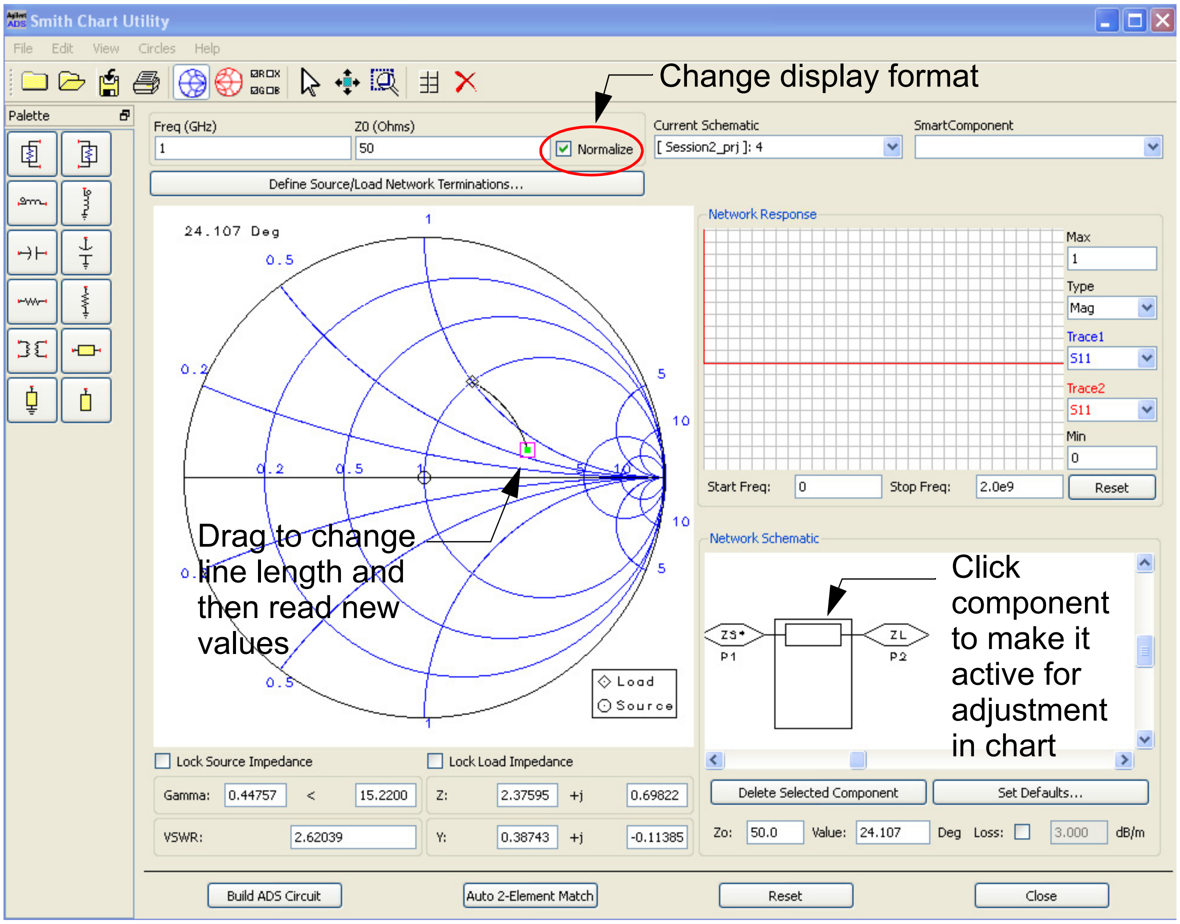
source and load defined




- One of the purposes of the utility is to aid you in the graphical design of matching networks
- Circuit elements placed between the source and load in the tool can then be placed in the schematic
- Here we will only consider the utility as a stand-alone tool
- Suppose we wish to model the following circuit:

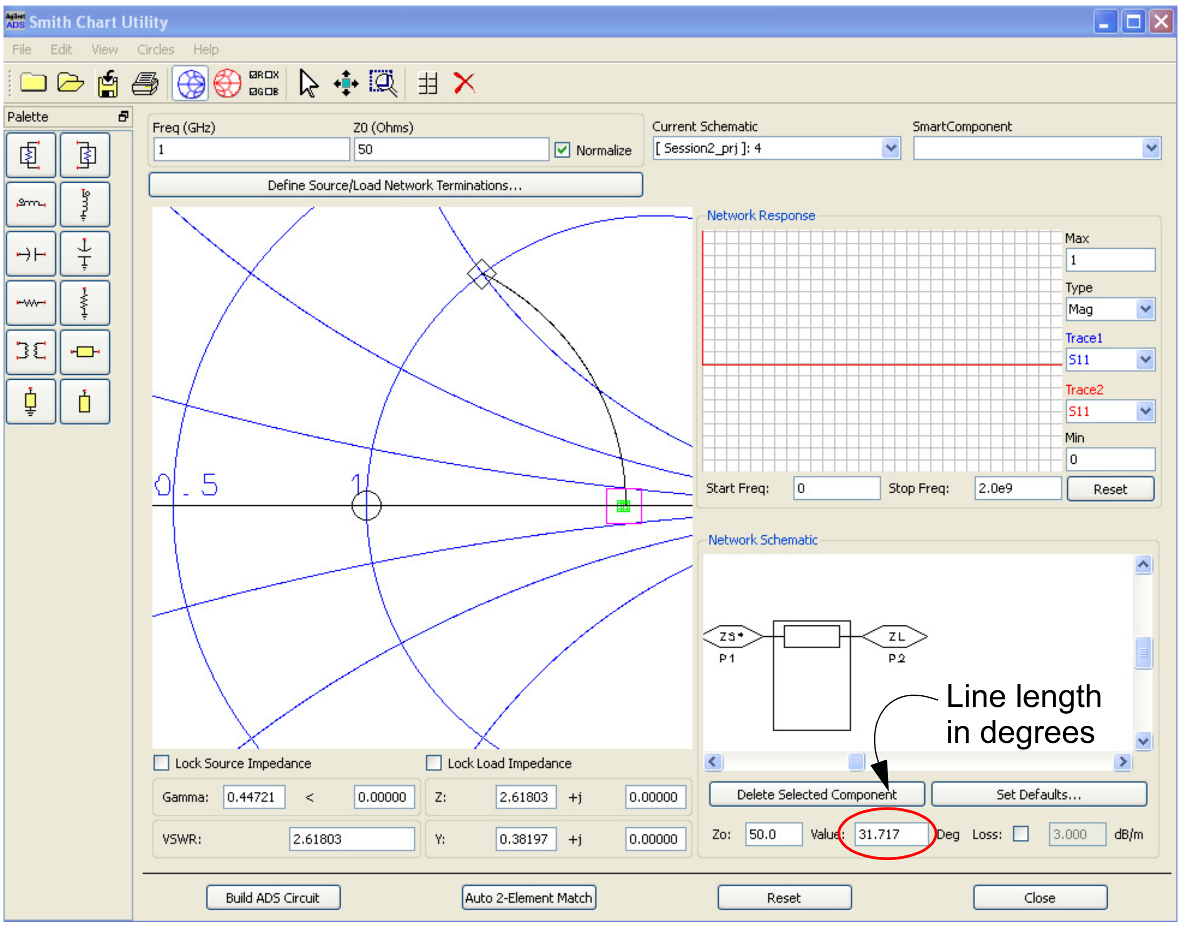


- The Smith chart provides a graphical means to determine $Z_{\text{in}}$ and/or $\Gamma_{\text{in}}$ for a given $\Gamma_L$, $Z_0$, and $\theta$
  - Clearly to obtain $\Gamma_{\text{in}}$, all we are doing is rotating $\Gamma_L$ by $-2\theta$ degrees (counter clockwise) in the $\Gamma$-plane
  - Other elements, such as series and shut inductors or capacitors are also easily managed via the graphical capabilities of the Smith chart (and the Smith chart utility); here the $\Gamma$-plane trajectories are along constant resistance or constant conductance circles, depending upon whether we have a series or shunt element (more on this when lumped element matching circuits are discussed)
- Moving forward with the transmission line example, we change the load `ZL` to $Z_L = 50 + j50$ in the Smith Chart Utility
- Next we pick a series transmission line element from the component palette with `ZL` selected, and then click on the Smith chart itself (this insures that the line section be placed in front of the load `ZL`, that is between the `ZL` and `ZS)`




- Note that you can drag the point you just clicked in the Smith chart constrained to a circle of constant $|\Gamma|$, and it will adjust the line length as it constrains the locus of points on the Smith chart to represent a trajectory towards the generator
- This is the *graphical-like* action of the utility, similar to using a real Smith chart; what do you think?
- All of the displays at the bottom of the window update dynamically
- The display of impedance/admittance related quantities can either be as normalized values or actual

- An interesting fact is that the maximum and minimum standing wave magnitude values occur when the value of $\Gamma$ looking into the line of length $\theta$ sits on the real axis at values either greater than one or less than one, respectively
- In the Smith Chart Utility we move to the greater than one crossing point first, and find that the corresponding line length in degrees is 31.71°



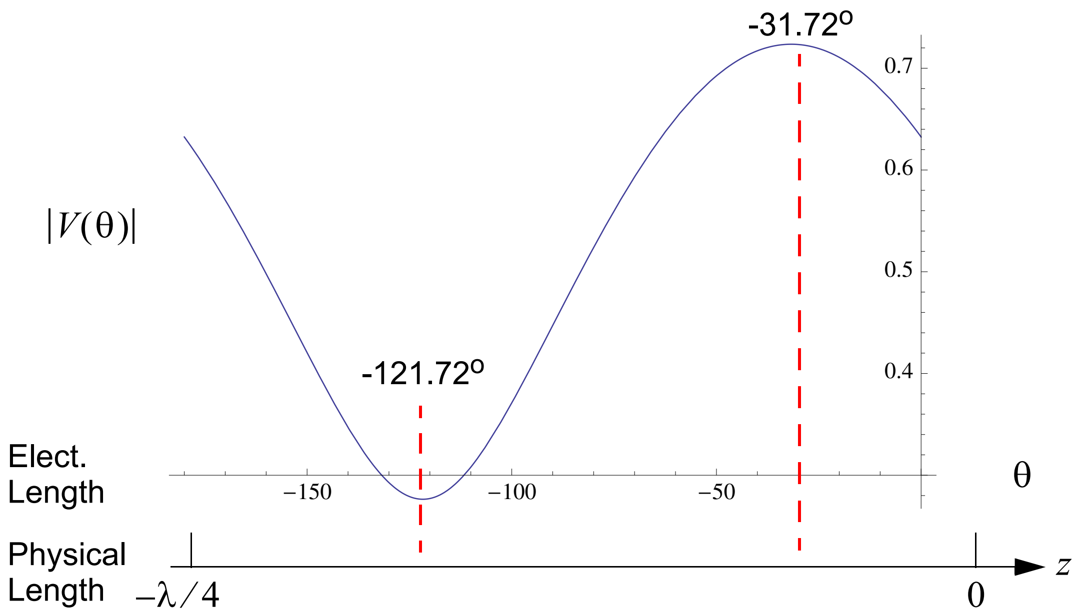

- We can use the total voltage anywhere along a line of electrical length $l$ to perform a mathematical check

- We construct the following mathematical model in Mathematica from notes equation (1.127)

```
V[θ_, θl_, Z0_, Zg_, ZL_] := Module[{Γg, ΓL, Vg = 1},
  Γg = (Zg - Z0)/(Zg + Z0);
  ΓL = (ZL - Z0)/(ZL + Z0);
  Vmag = Abs[Vg Z0/(Z0 + Zg) (1 + ΓL e^(j 2 θ))/(1 - Γg ΓL e^(-j 2 θl)) e^(-j (θ+θl))]]
```

- Using this function we can then plot the standing wave pattern as $|V(z)|$ or equivalently as $|V(\theta)|$ we move back from the load towards the source
- For the parameters we are considering, the plot is as shown below

```
Plot[V[θ π / 180, π / 2, 50, 50, 50 + j 50], {θ, -180, 0}]
```




- With the solving capability of Mathematica we can now find the location of the minimum and maximum values as shown below:

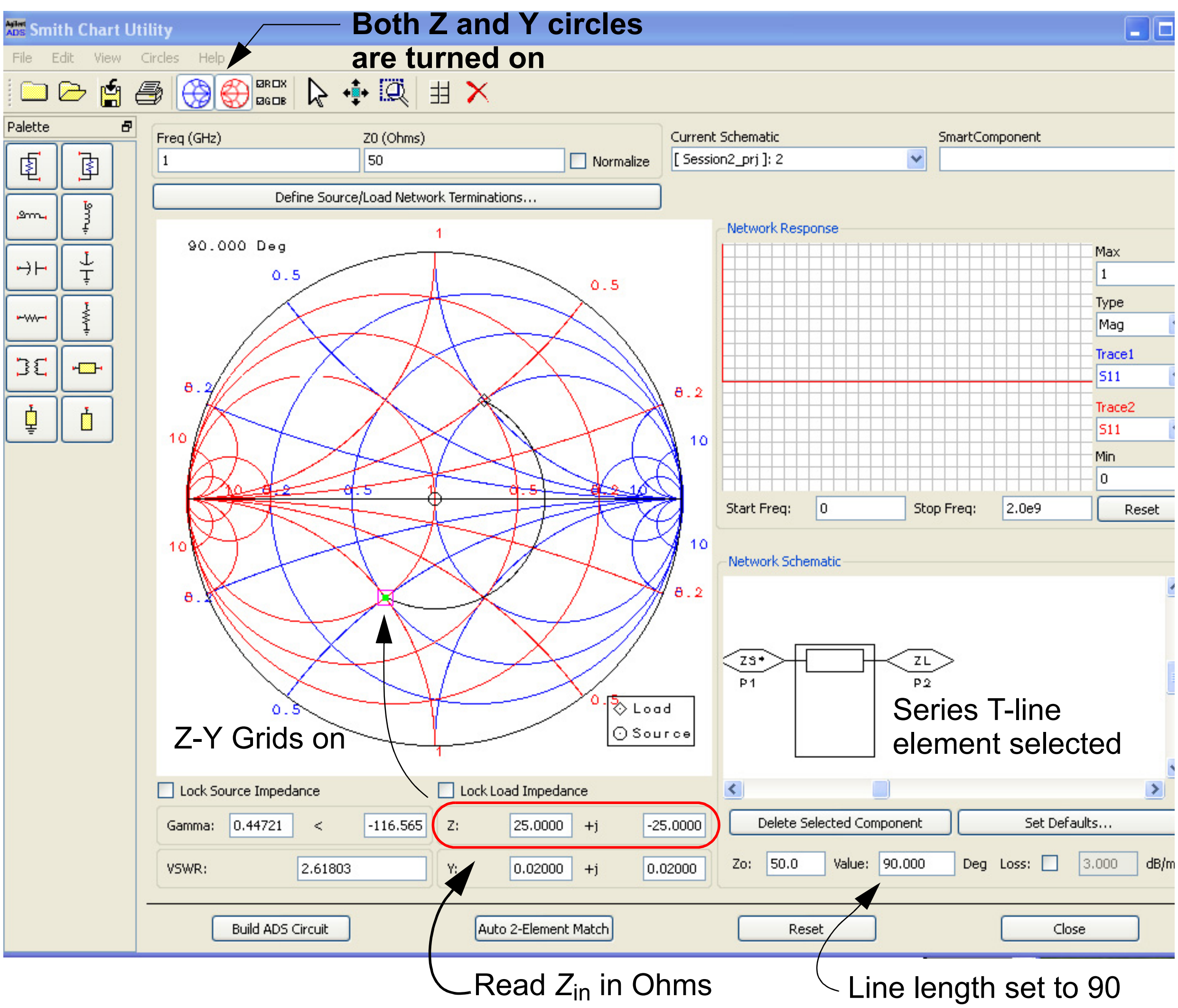
```
FindMaximum[V[θ π / 180, π / 2, 50, 50, 50 + j 50], {θ, -30}]
{0.723607, {θ → -31.7175}}
```

← Agrees with ADS

```
FindMinimum[V[θ π / 180, π / 2, 50, 50, 50 + j 50], {θ, -100}]
{0.276393, {θ → -121.717}}
```

← Agrees with ADS (not shown here)

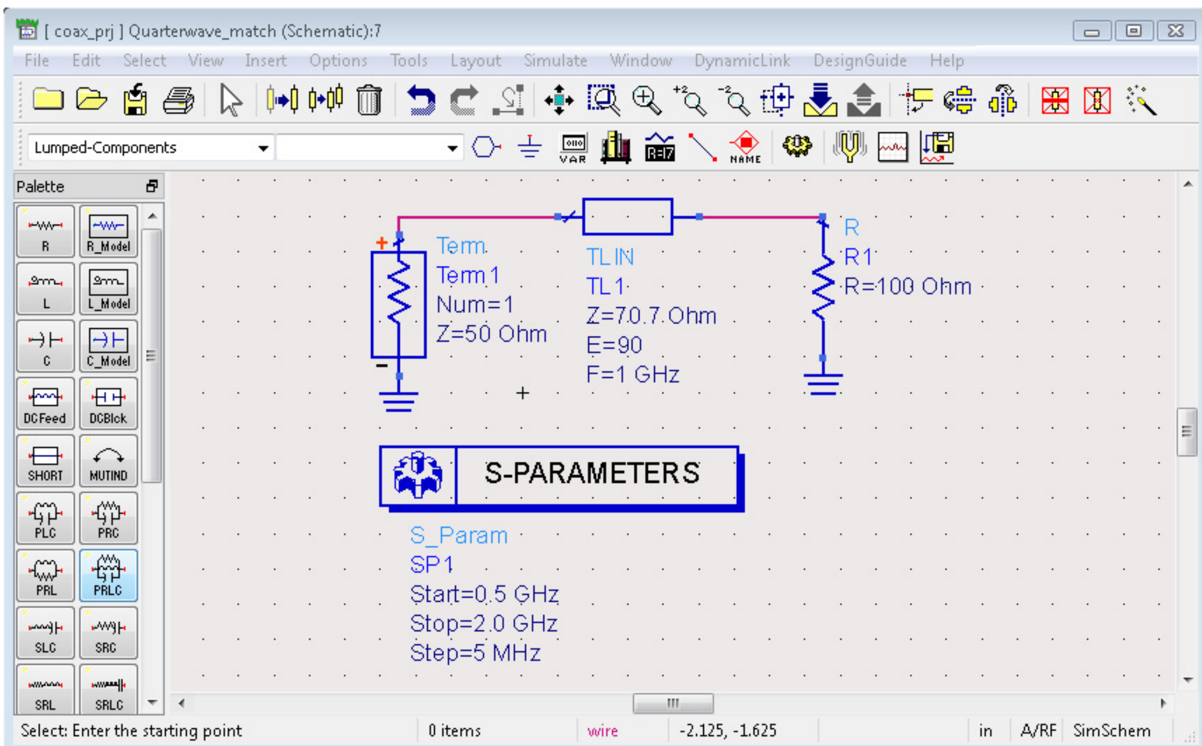
- If we want to find the impedance looking into a line of length $\lambda/4$, we simply drag the marker or type the line length of $90°$ into the window, and then we can read the new $Z_{in}$ at the bottom of the chart



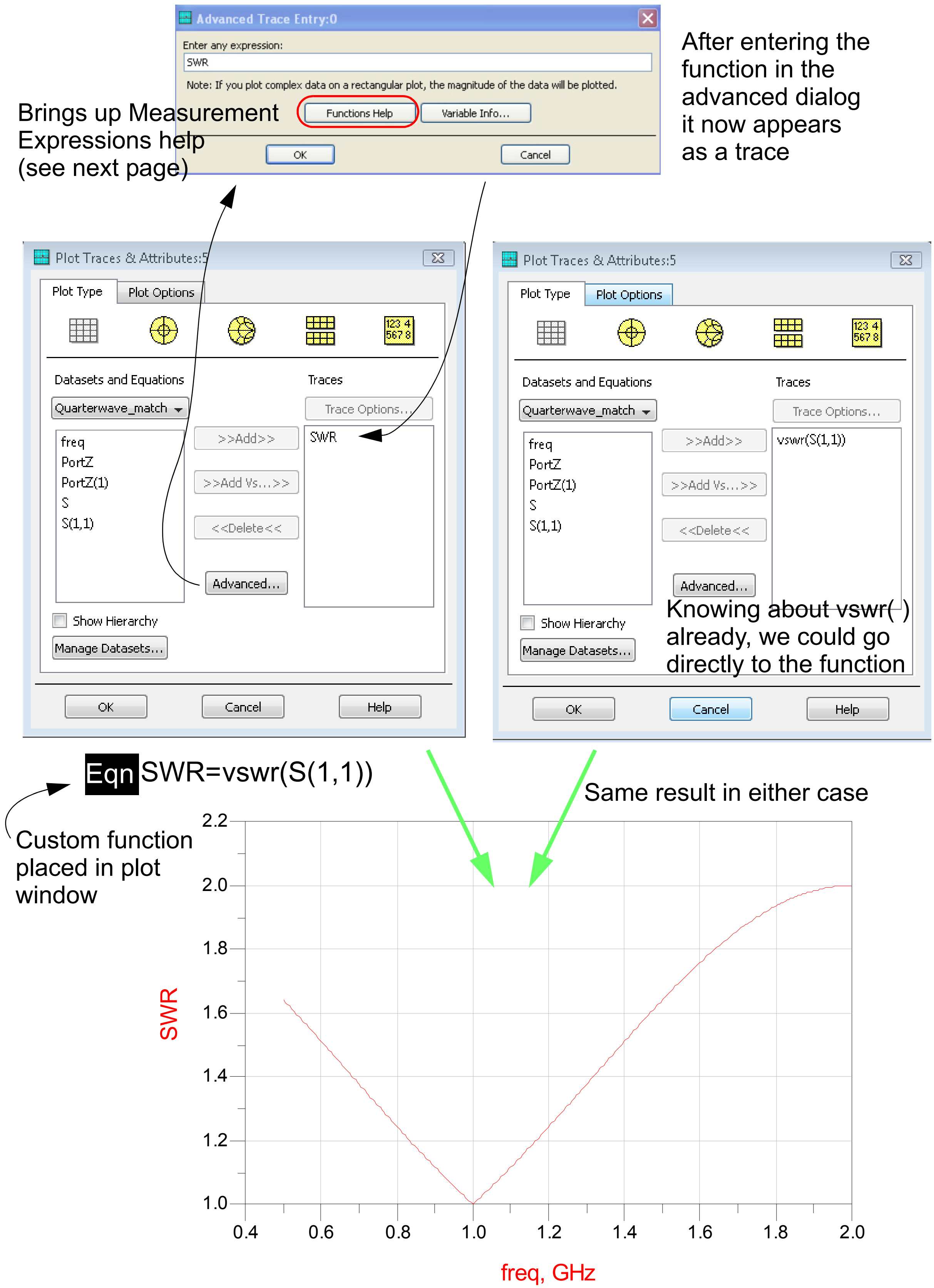
- Explore other capabilities on your own

# Custom Plotting

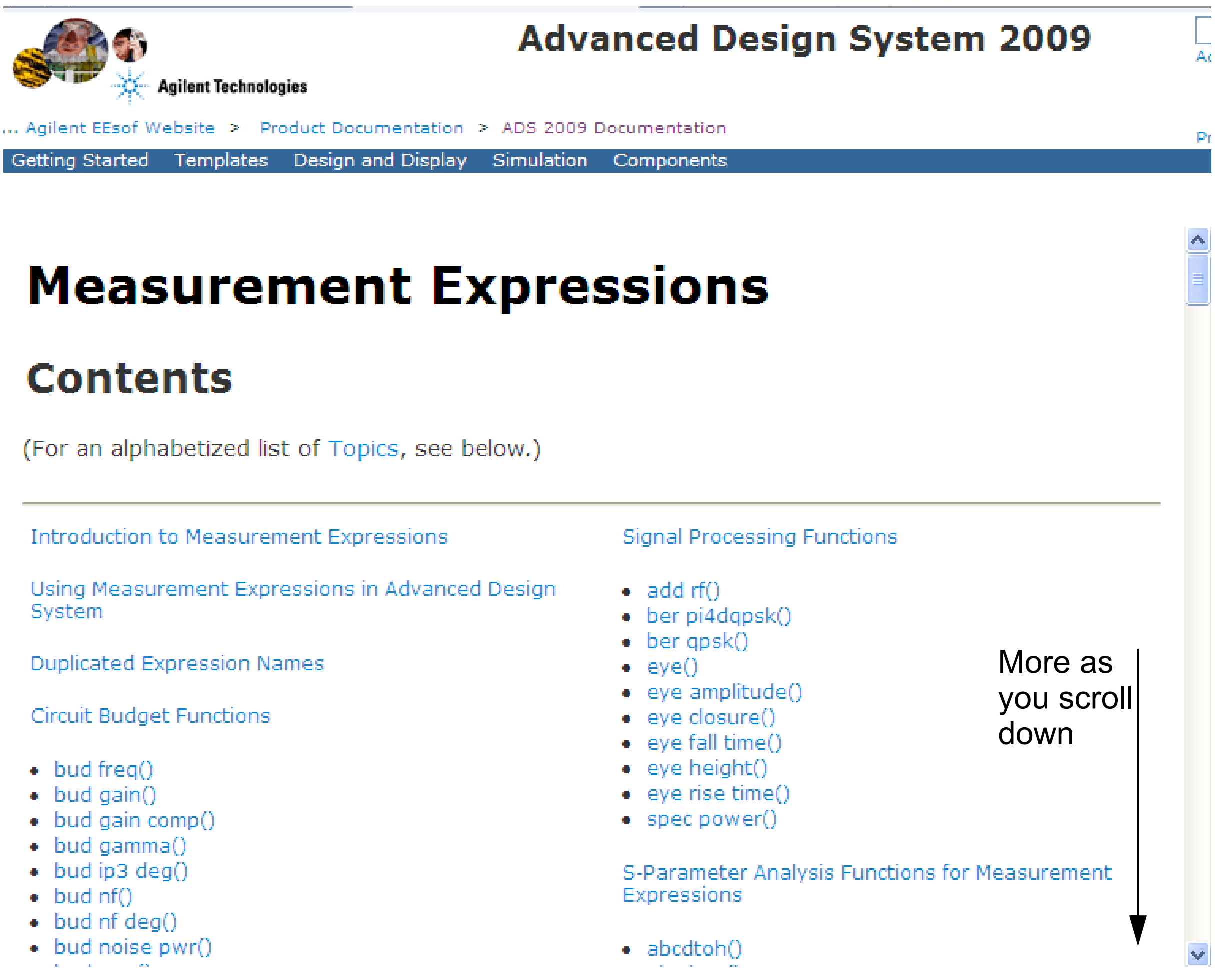
- Consider the analysis of a quarter-wave matching circuit to match 100 ohms to 50 ohms, at 1 GHz
  - Note in particular here that we have set up a one-port circuit, since we have let the output be terminated in a 100Ω resistor, as opposed to using a termination block




- It is possible to write custom functions for plotting or use pre-built functions
- In the above we see that when analyzing with S-parameters we may not have the options we wish immediately available, e.g., VSWR


Advanced Trace Entry:0
Enter any expression:
SWR
Note: If you plot complex data on a rectangular plot, the magnitude of the data will be plotted.
Functions Help
Variable Info...
OK
Cancel
Brings up Measurement Expressions help (see next page)
After entering the function in the advanced dialog it now appears as a trace
Plot Traces & Attributes:5
Plot Type
Plot Options
Datasets and Equations
Traces
Quarterwave_match
Trace Options...
freq
PortZ
PortZ(1)
S
S(1,1)
>>Add>>
>>Add Vs...>>
<<Delete<<
Advanced...
SWR
vswr(S(1,1))
Show Hierarchy
Manage Datasets...
OK
Cancel
Help
Knowing about vswr( ) already, we could go directly to the function
Eqn SWR=vswr(S(1,1))
Custom function placed in plot window
Same result in either case
SWR
freq, GHz
2.2
2.0
1.8
1.6
1.4
1.2
1.0
0.4
0.6
0.8
1.0
1.2
1.4
1.6
1.8
2.0

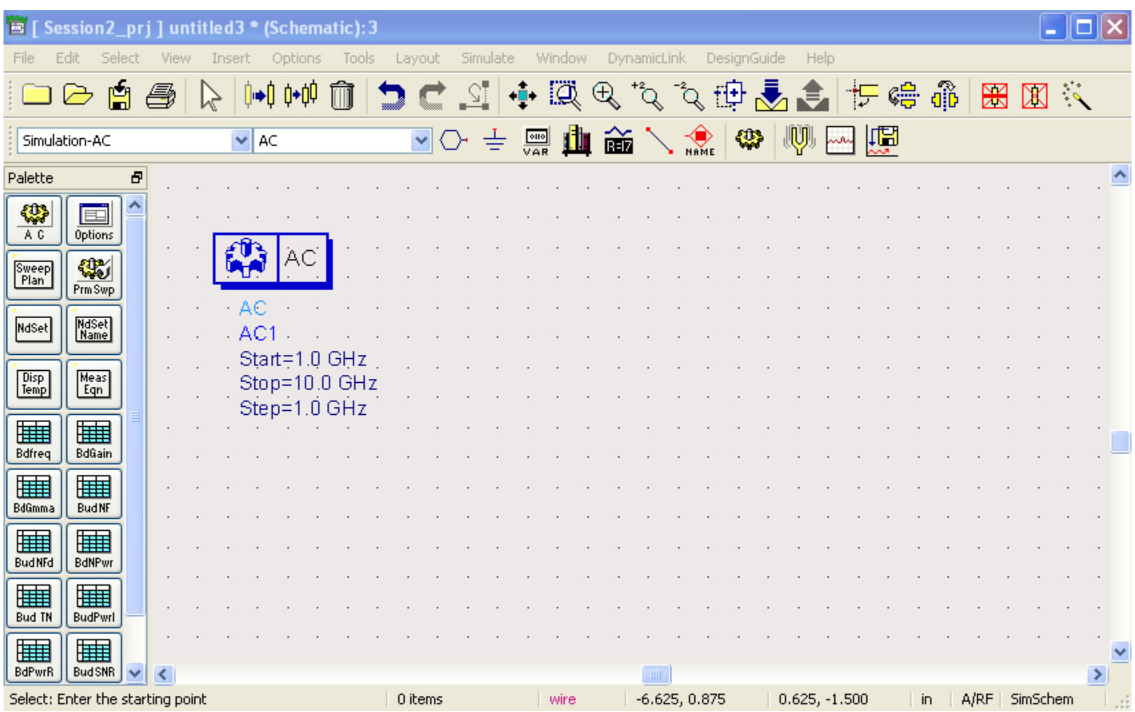
- To explore the function library for writing equations visit the ADS HTML help system and look under Measurement Expressions



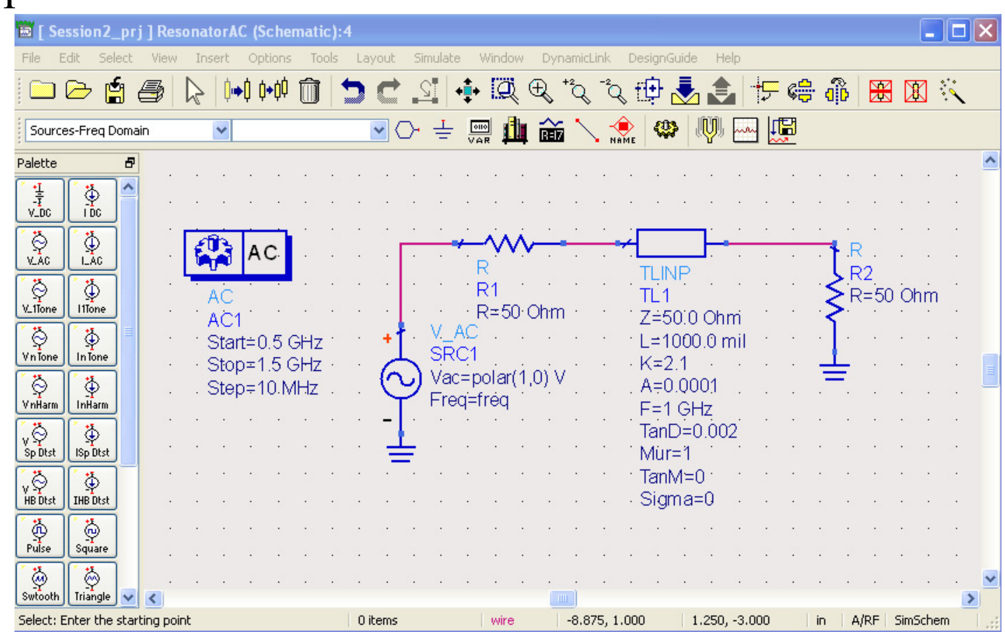

# The AC Simulation and Power Measurements

- To take actual power measurements in a linear circuit, the AC controller can be useful
- AC circuit simulation is very much like AC analysis available in Spice

- Analysis is performed using sinusoidal sources with the node variables being voltages and currents, each of which are complex quantities
  - Node voltages are available by placing probe points on any wire connecting two components
  - Branch currents are obtained by placing current probes in series
  - Power delivered to a portion of a circuit can be obtained using a power probe (grounded or floating)
- The frequency of the sinusoidal source is swept over a start/stop range, so all measure measurements are obtained as steady-state values versus frequency
- We begin by placing an AC controller from the Simulation-AC palette



- We are going to model a simple bandpass filter created using a single quarter-wave length section of transmission line
  - We will also include loss in the resonator
- We need to place an AC signal source down as well as load and source resistances and a transmission line that incorporates loss, but does not assume a particular geometry
  - We obtain the AC voltage source is obtained from the Sources-Freq Domain palette
  - The resistors are obtained from the Lumped-Components palette (note `ctr-r` rotates components)
  - The transmission line is obtained from the TLines-Ideal palette



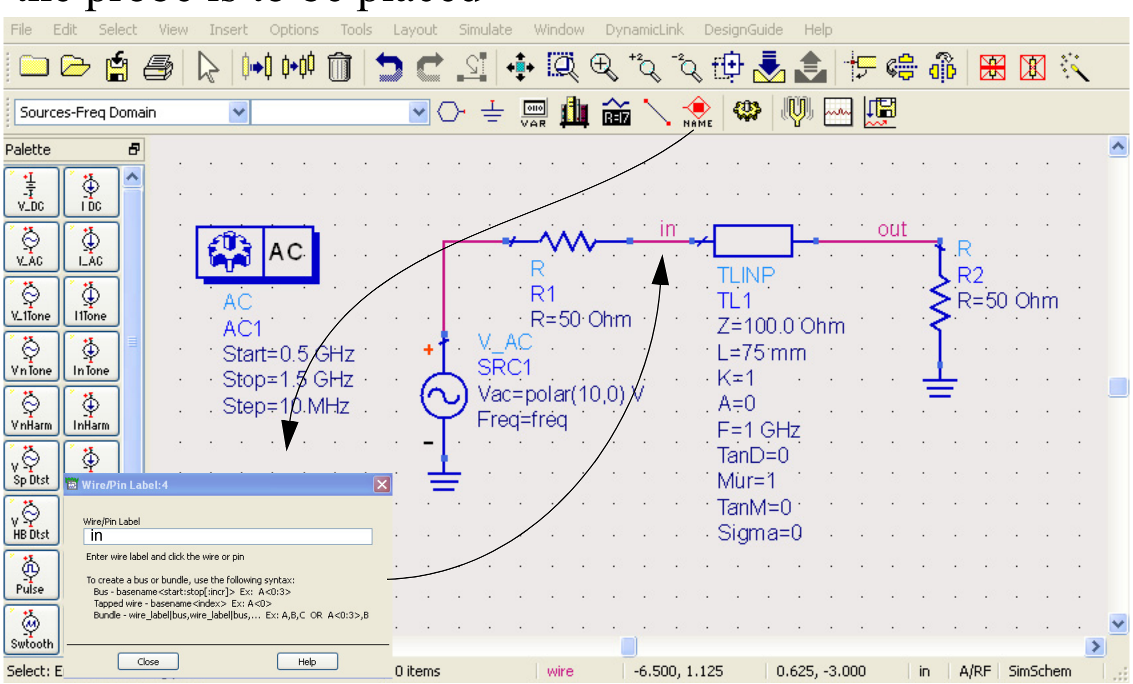

  - Note we have also set the sweep to run from 0.5 GHz to 1.5 GHz in 10 MHz steps
- We want the quarter-wave frequency of the resonator to be at

1 GHz and choose the characteristic impedance to obtain a particular 3 dB bandwidth via design equations (here we have simply set $Z_{0,\text{ resonator}} = 100\Omega$)

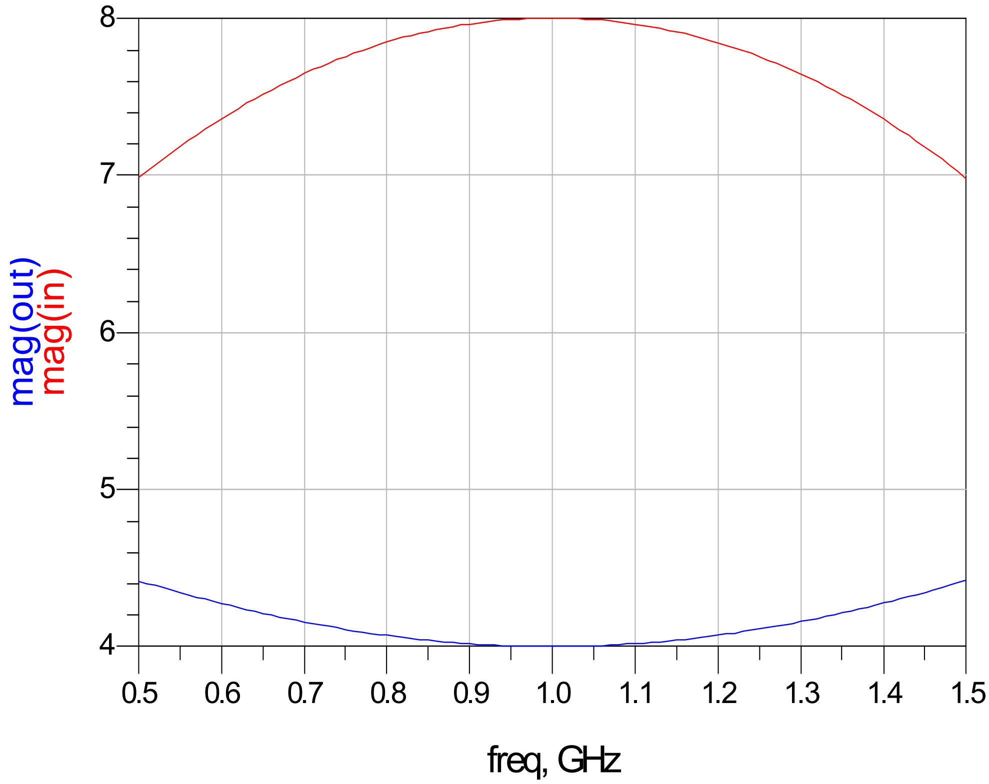
- The `TLINP` element has parameters similar to the coax
- We will set the line up with an air dielectric ($\varepsilon_r = K = 1$) and set the loss tangent to zero (`TanD` $= 0$)
- Loss can then be controlled via `A`, which sets the line loss in dB per meter
- To obtain the proper line length we use the fact that $\lambda_g = c/f_0$, where $c = 3\times10^8$ m/s, so $\lambda_g/4 = 75\,\text{mm}$

• Set the amplitude and phase of the source to $10\angle 0$

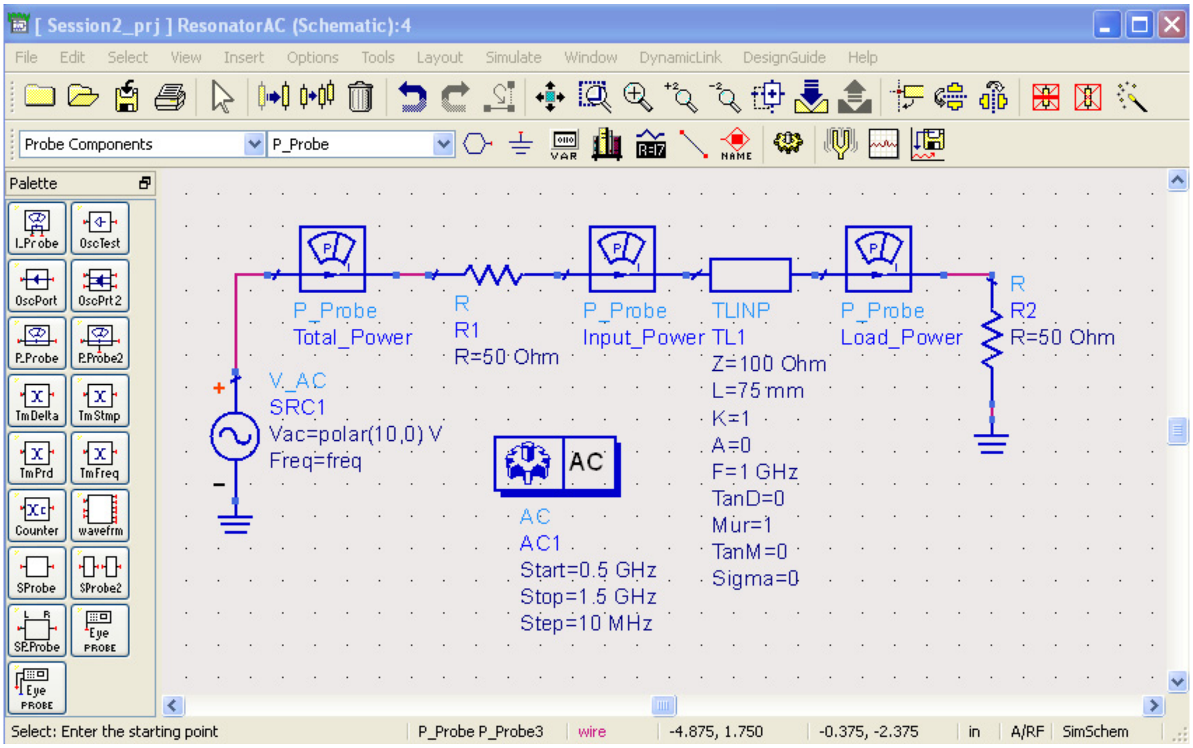
• Place voltage probes at the input and output toolbar probe button (diamond with red square and name below); click and then enter a name for the probe and click on the wire where the probe is to be placed



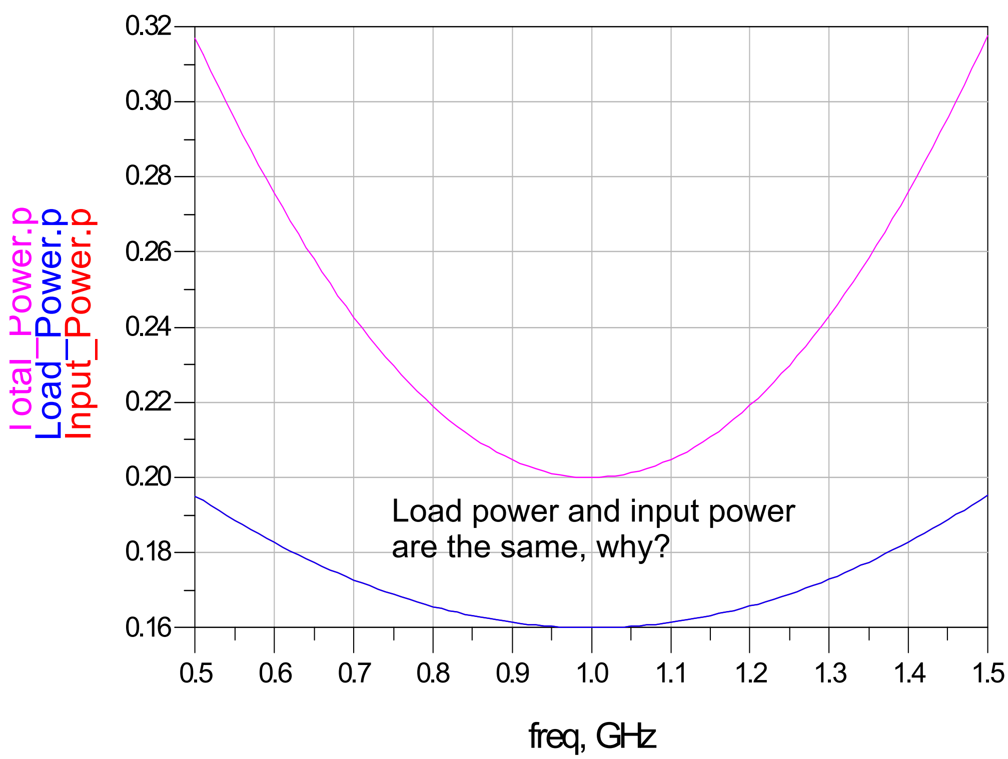
- We are now ready for the first run of the simulation (with no line loss)
- We plot the input and output voltage magnitude



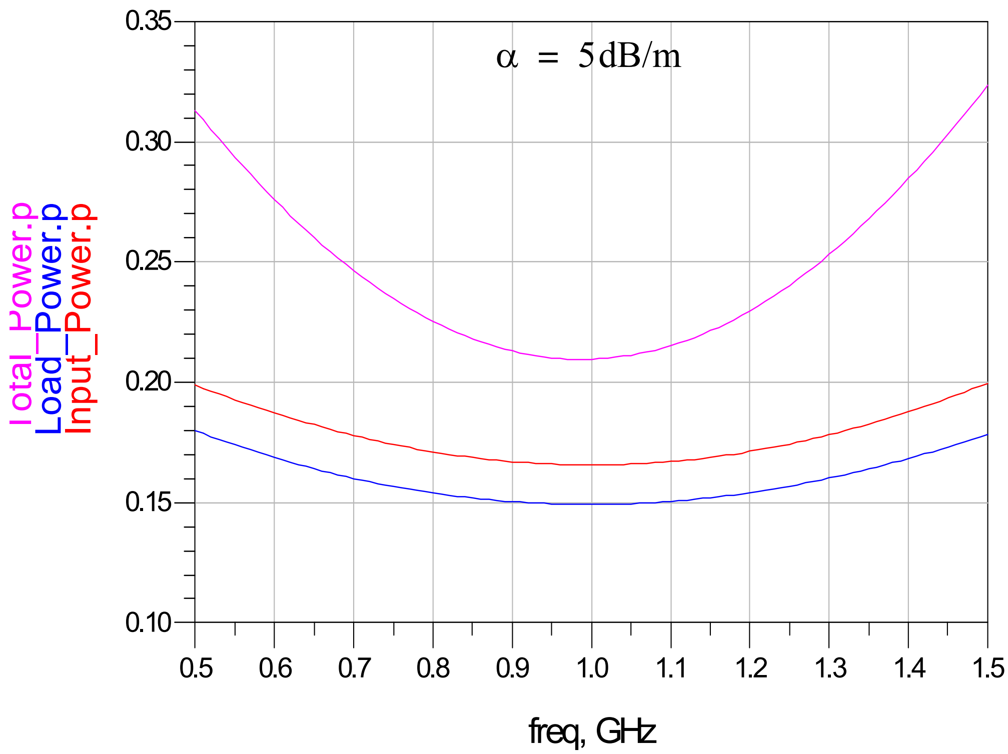

- Interesting results, but perhaps we want to know for the specific input source how much power is delivered to various parts of the circuit
- We select the Probe Components palette and insert grounded power probes at three locations in the circuit (note holding down the right mouse button allows you to drag the schematic page around in the window and the mouse scroller allows you to zoom in and out)
- The reworked schematic is shown below
  – When wires were deleted the voltage probes we lost too




- Now we sweep once more and plot the measured power values (the power probe labels have be edited to give them more meaningful names)



- Power is absorbed by all of the resistive elements in the circuit
- Note that the load power and input power are the same when the line loss is 0 dB/m
- Now we will increase the line loss to 5 dB/m

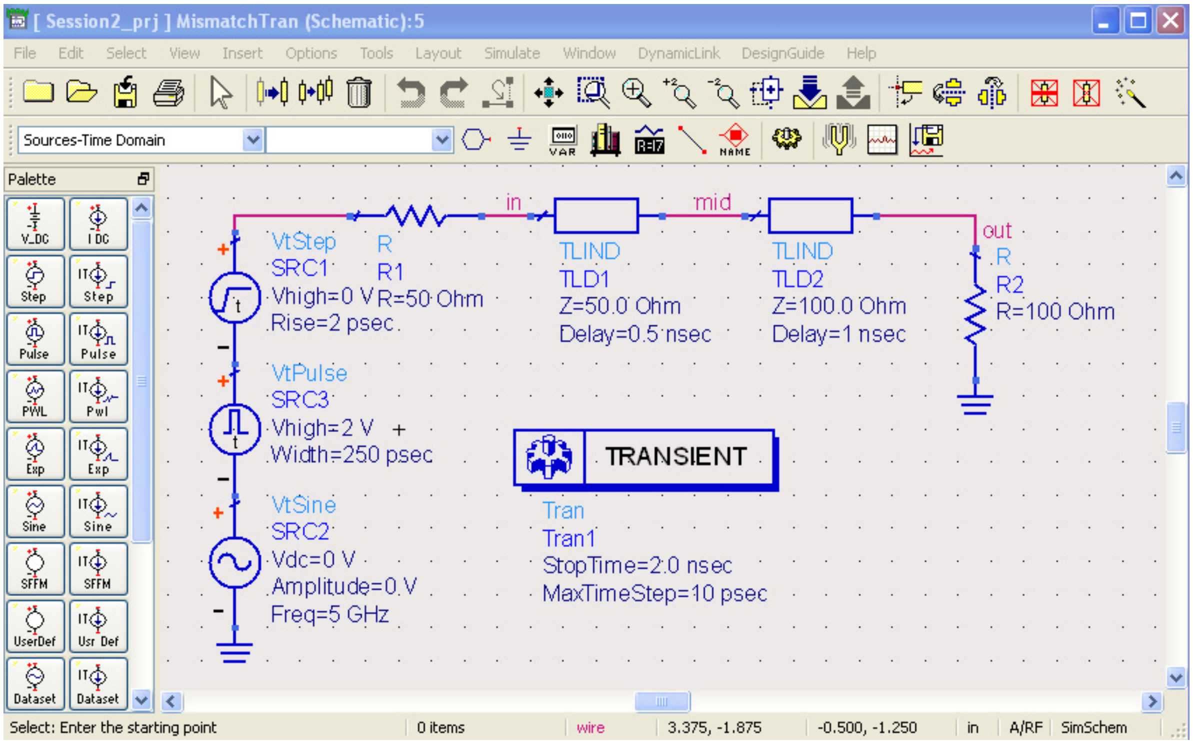



- We now see that the $\lambda_g/4$ resonator absorbs power too, as the load power is less than the input power

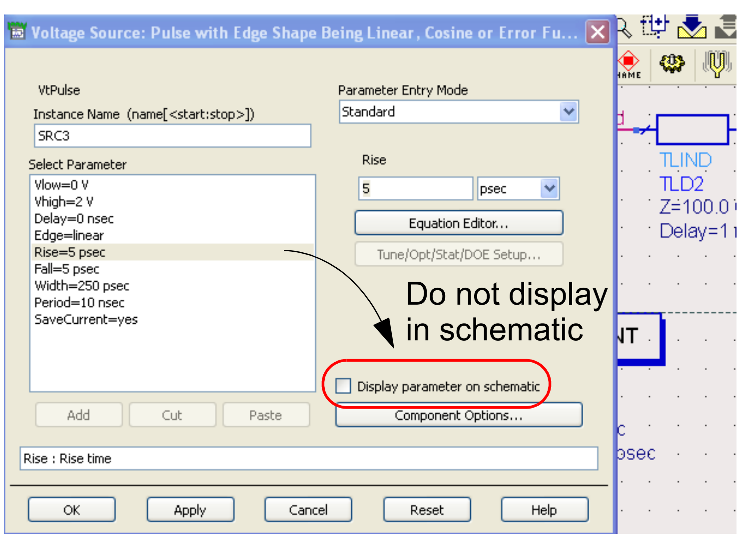
# Transient Analysis using the Transient Controller

- In the final segment of this tutorial, we will consider time domain simulation using the Transient controller
- Transient simulation in ADS is much like transient simulation is Spice
  - Sources are selected from the Sources-Time Domain palette
  - The transient controller is obtained from the Simulation-Time Domain palette
  - Measurements consist of node voltages, branch currents, and power, much like is available for AC analysis
- The real important facet of transient simulation is that you get to work with the actual waveform, and include transmission line elements and other general distributed parameter elements that you do not find in ordinary Spice
- Here we will model a circuit composed of ideal transmission elements and consider what transients are created by line impedance discontinuities
  - Specifically we will drive are circuit with pulse, step, and sinusoid
  - The transmission lines will be ideal lossless lines whose length is given in terms of the equivalent time delay introduced by the line (we use the element `TLIND`)

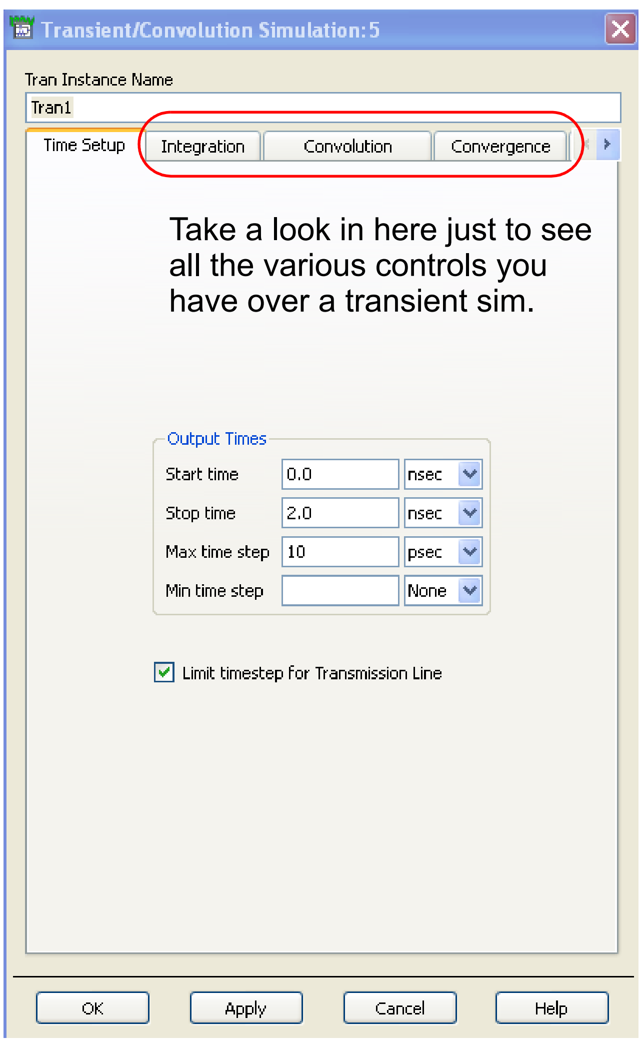
- The schematic for the circuit of interest is shown below:




- Note in particular that three sources are all wired in series
  - The intent is to turn on and off the sources by controlling the respective amplitude values
  - To make the schematic cleaner some of the default schematic display values have been turned off, e.g.,

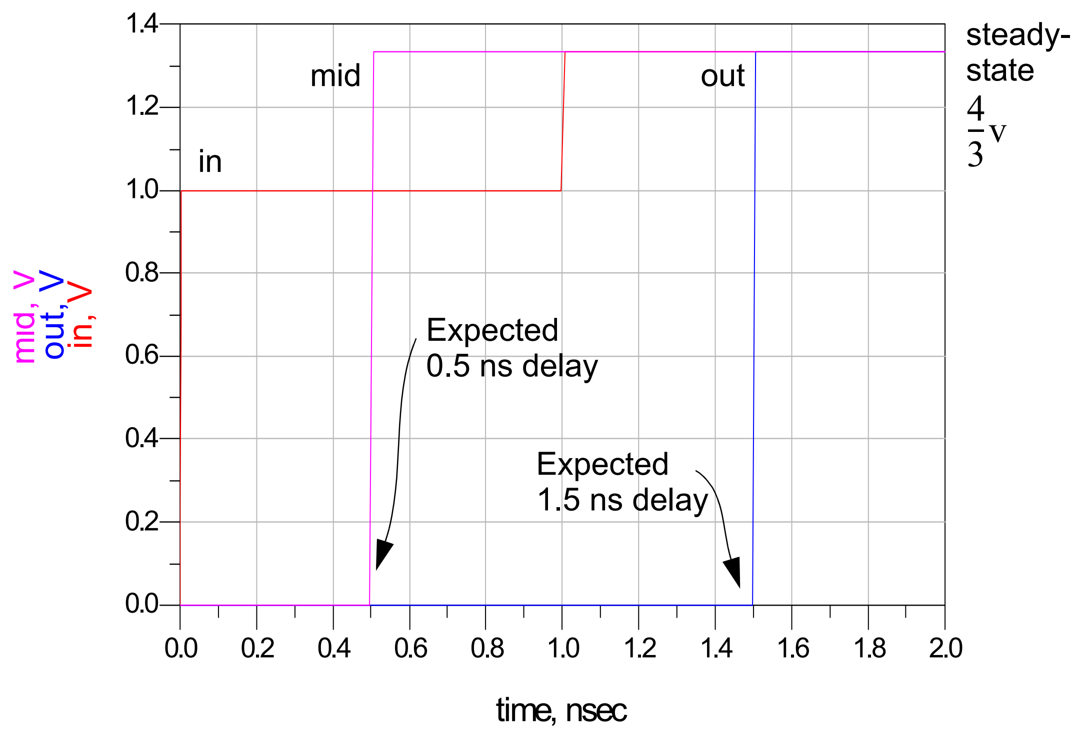


– Probe points for voltage have been added at three locations: (in) the input, (mid) the point of impedance discontinuity, and (out) the output load
– In the screen capture the pulse source is turned on with an amplitude of 2v, while the other sources are set to zero
– The Transient controller block is set to simulate from zero out to 2 ns, with the maximum time step allowed set to 10 ps
– Just as in Spice, you have to think about how to set the time step limits (upper and lower)




- There are a lot of extra settings that can be dealt with for transient simulations
- Here we have chosen default values for all of the tabs in the dialog box, except for the maximum step size
- Read the help and play with settings if you feel you are not getting what you expect

- Time domain simulations characterize both linear and nonlinear behavior of circuits and systems
- The downside is the fact that simulation time can be very long
- Getting on with the simulation, click run and plot the three voltage waveforms




- The source is approximately of the form $v_{in}(t) = 2u(t)$
- At the node `in` we observe that the applied step launches onto the first line section with amplitude 1v, since there is a voltage divider created between the 50 ohm source and the 50 ohm line ($2 \times 50/100 = 1$)
- At `mid` we do not see anything until 0.5 ns have passed due to the `TLD1` have a delay of 0.5 ns

- The voltage at `mid` initially rises up to 1.33 v (why?)
- The incident wave coming down the line from `TLD1` sees an impedance mismatch (50 ohms to 100 ohms) which has a reflection coefficient of

$$\Gamma = \frac{100 - 50}{100 + 50} = \frac{50}{150} = \frac{1}{3} \tag{2.1}$$

- The incident step of 1v is reflected with amplitude $1 \times \Gamma = 1/3$
- The reverse traveling wave of amplitude 1/3 now combines with the incident wave, to form a total amplitude of 1.33, as measured by `mid`
- The reverse wave from the discontinuity travels back to the source impedance where it is completely absorbed due to the perfect impedance match ($\Gamma_g = 0$)
- The wave incident on TLD2 is launched with the incident amplitude times the transmission coefficient $T = 1 + \Gamma$, so the incident wave arriving at the load at 1.5 ns is also 1.33v
- There are no reflections at the load due to impedance match ($\Gamma_L = 0$)

• Backing up from the above theory of operation discussion, we can also look at the circuit and say that if we apply 2v to a circuit composed of a 50 ohm resistor in series with a 100 ohms resistor, in steady-state we will have $2 \times 100/150 = 1.33\,\mathrm{v}$ appearing at the output (it checks)

- Just to see what happens we repeat the simulation with first the 250 ps wide pulse source turned on and then the sinusoid turned on (can you explain these results?