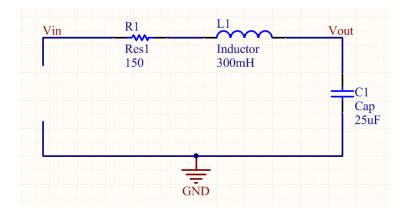
Altium Designer – Simulation Tutorial

Last updated: 8/4/16, John Miller, Altium Designer 14.3

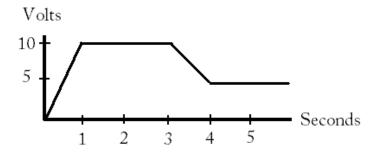
IMPORTANT! See the *Altium - Getting Started* tutorial before reading this one.

The following two sections will use the same circuit (below) with different sources.



Transient Simulation

The first step is to select and configure the input source. For transient analysis, we can use any
of the simulation sources, but we are often interested in a step response. The easiest way to get
a step input is to use a piece-wise linear source or VPWL. This means we will define the voltage
at a series of points in time. The source will produce an output that changes linearly between
each of these defined points. The following waveform would be produced by the sequence
shown in the table.



Time	Voltage
0 s	0 V
1 s	10 V
3 s	10 V
4 s	5 V

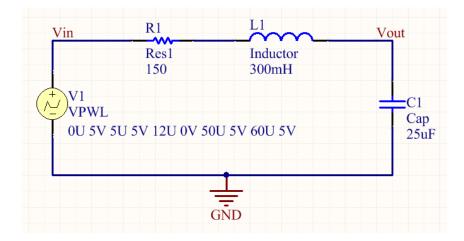
2. To do this in Altium, first place the VPWL source. Then double-click to open the Properties dialog. Click the **Edit** button in the Models section (lower right).

Properties			Paramet	ers				
			Visible	Name	△ Value		Туре	
Designator	V1	Visible 📃 Locked		LatestRevisionDate	17-Jul-2002		STRING	
Comment	VPWL -	Visible		LatestRevisionNote	Re-released for D	XP Platform.	STRING	
		Part 1/1 Docked		Note	PCB Footprint - N	lot required	STRING	
	< < > >>	Part 1/1 📃 Locked		PackageReference	Not Applicable		STRING	
escription	Piecewise Linear Voltage Sourc	e		Published	8-Jun-2000		STRING	•
Inique Id	BTEBEENJ	Deat		Publisher	Altium Limited		STRING	
inque tu	BIEBEEINJ	Reset	L	Sim Note	To add model file	e containing tir	me-ci STRING	
ype	Standard	•						
Link to Librar Design Item ID I Library Name Table Name	vPWL Simulation Sources.IntLib	se Vault Component						
Design Item ID Ibrary Name Table Name	VPWL		<u>A</u> dd	Remoye	dit Add as	<u>R</u> ule		
esign Item ID Z Library Name	VPWL	Choose	Models				Ban Day D	wition
esign Item ID] Library Name] Table Name Graphical	VPWL Simulation Sources.IntLib	Choose) Validate Link		Remoye §		<u>R</u> ule Vault	Item Rev Re	
esign Item ID 2 Library Name 2 Table Name 3 Table Name 3 Taphical ocation X	VPWL Simulation Sources.IntLib	Choose) Validate Link	Models Name	Type 🔺 Descript				
esign Item ID] Library Name] Table Name Graphical ocation X	VPWL Simulation Sources.IntLib	Choose) Validate Link	Models Name	Type 🔺 Descript				
esign Item ID 2 Library Name 2 Table Name 3 Table Name 3 Table Name 4 Table Name	VPWL Simulation Sources.IntLib	Choose Validate Link	Models Name	Type 🔺 Descript				
Design Item ID Ibrary Name Table Name	VPWL Simulation Sources.IntLib	Choose Validate Link	Models Name	Type 🔺 Descript				evision
esign Item ID 2 Library Name 7 Table Name 8 raphical ocation X 0 rientation	VPWL Simulation Sources.IntLib	Choose Validate Link	Models Name	Type / Descript				

3. This brings up a second dialog. Click the **Parameters** tab (red arrow). You should have the dialog below. You can edit the table here or...

		Component parameter		
DC Magnitude	0			
AC Magnitude	1			
-	0			
AC Phase	-			
Time / Value I	~airs	1	Compor	ient para
Time (s)		Voltage (V)		
		5V	Add	
5U		5V	Delete	
12U				
500		5V		
60U		5V		
DESIGNATOR 2	1 %2 ?"DC MAGN	TUDE" DC @"DC MAGNITUDE"	PWL(?MODELLOCATION/FILE=	=@MODE
<				

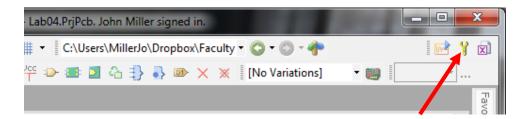
4. Checking the "Component param" box (green arrow above), and clicking OK twice, will display the sequence on your schematic, which you can edit directly.



5. For our simulation, we want to do a simple step function, meaning it transitions from 0 to 1 in an infinitesimal amount of time. Some programs require you to have a non-zero amount of time for the transition, but Altium will actually let you specify a 0 second time step, such as: 0u 0V 0u 1V

This means "at 0 microseconds output 0 V, then at 0 microseconds output 1 V." Double-click the text string and enter that sequence. (It's not case sensitive.)

Click the Setup Mixed-Signal Simulation icon (red arrow) on the simulation toolbar (or go to Design → Simulate → Mixed Sim). You can make the toolbar appear by going to View → Toolbars → Mixed Sim.



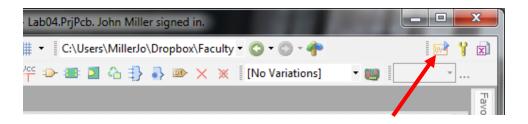
7. The following dialog appears. Your values for the fields *Collect Data for* and *Sheets to Netlist* should be the same as below by default. Also leave the *SimView Setup* option as **Keep last setup** (red arrow). This will maintain any modifications we make to the display of the waveforms. Under *Available Signals*, select **IN**, **OUT**, and **C1[i]** and click the arrow (green arrow) to move them to the *Active Signals* list. Click OK.

Analyses Setup					
Analyses/Options General Setup Operating Point Analysis Transient Analysis DC Sweep Analysis AC Small Signal Analysis Noise Analysis Pole-Zero Analysis Transfer Function Analysis Temperature Sweep Parameter Sweep Monte Carlo Analysis Global Parameters Advanced Options	Enabled	Collect Data For Sheets to Netlist SimView Setup Available C1[p] L1[p] NetL1_1 R1[p] NetL1_1 R1[p] V1#branch V1[p] V1[z]	Node Voltage, Su Active project Keep last setup e Signals	C1[i] VIN VOUT	vice Current and Power
Preferences					OK Cancel

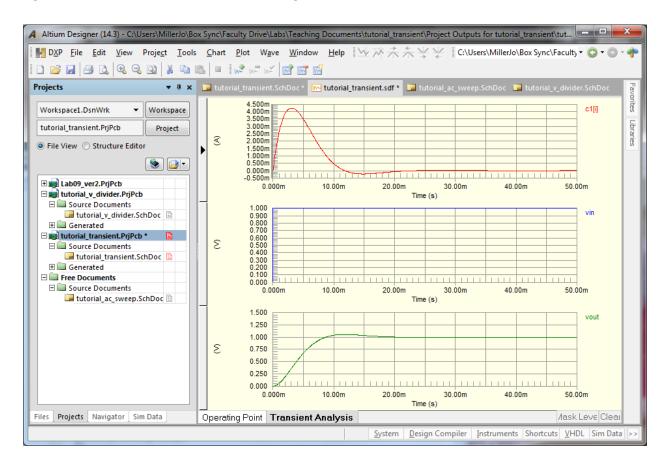
 Click on Transient Analysis to bring up the transient analysis options. Uncheck the "Use Transient Defaults" box (red arrow). Set the Stop Time to be 50 ms and the Step Time to be 50 us. (Note: one order of magnitude smaller than the length of the simulation is usually a good setting for step time. This will give you 1000 points in the simulation.). Be sure the Transient Analysis box is checked (on the left).

Analyses/Options Enabled		Transient Analysis Setup				
Seneral Setup		Parameter	Value			
Operating Point Analysis Transient Analysis	✓✓	Transient Start Time	0.000			
C Sweep Analysis		Transient Stop Time	50.00m			
C Small Signal Analysis		Transient Step Time	50.00u			
loise Analysis		Transient Max Step Time	50.00u			
ole-Zero Analysis		Use Initial Conditions				
ransfer Function Analysis						
emperature Sweep		Use Transient Defaults				
Parameter Sweep Monte Carlo Analysis		Default Cycles Displayed	5			
Slobal Parameters		Default Points Per Cycle	50			
dvanced Options						
		Enable Fourier				
		Fourier Fundamental Frequency	1.000meg			
		Fourier Number of Harmonics	10			
		,	Set <u>D</u> efault			

9. Click the Run Mixed Signal Simulation icon on the toolbar or press F9 (shortcut).



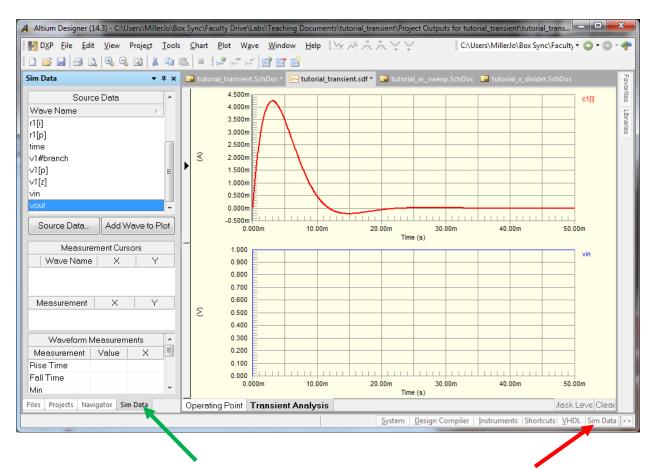
10. You should get the following results, showing the transient response for the three different signals we chose. We can now do lots of things with this.



11. Let's start by improving the formatting. Right-click anywhere on the chart and choose **Document Options** to get the following dialog. Check the "Bold Waveforms" box. This makes the waveform lines thicker and easier to see.

Document Options	×
Colours	View
Grid	Bold Waveforms
Foreground	☑ Highlight Similar Waves
	Show Data Points
Background	Show Designation Symbols
Swap Foreground/Background	Show Chart Title
	Show Plot Title
Fast Fourier Transform	Show Axis Labels
FFT Length 128 🚔	🔄 Show Sim Data panel
	Number of Plots Visible 3 👻
Apply to Active Chart Only	OK Cancel

12. Next, let's combine the Vin and Vout waveforms on the same plot (since they are both voltages and we are interested in the relationship between them). Right-click on the Vout plot (bottom) and click Delete. Click the **Sim Data** button (red arrow) or tab (green arrow) to bring up the Sim Data panel.



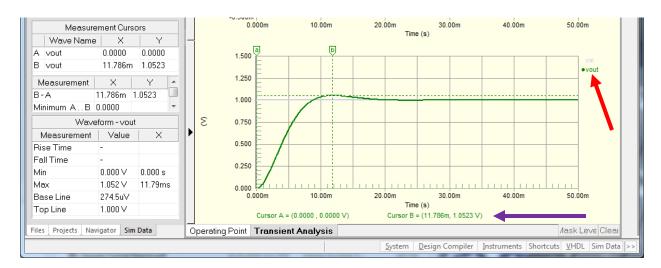
Click the lower plot to "select" it (or make it "active"), as indicated by the small black arrow (being highlighted by the red arrow). Select Vout in the Source Data list and click Add Wave to Plot.

🛃 DXP Eile Edit View Proje <u>ct T</u> ool			p I₩₩Å	***	C:\Users\N	∕lillerJo\Box Syn	ic\Faculty 🔹 🕥 🕶	0 - 4
		ial_transient.SchDoc * 📴 tutor	ial_transient.sdf *	🖬 tutorial_ac_swe	eep.SchDoc 🗔 t	utorial_v_divide	r.SchDoc	Fave
Source Data Wave Name / r1[i] // r1[p] // time // v1#branch // v1[p] = v1[z] // vout * Source Data Add Wave to Plot	(3)	4.500m 4.000m 3.500m 3.000m 2.500m 1.500m 1.500m 0.000m 0.000m 1.000m				40.00m	c1[]	Favorites Libraries
				Time (s)			00.00	
Measurement Curso s Wave Name X Y Measurement X	L €	1.000 0.900 0.800 0.700 0.600 0.500 0.400					vin	
Wave Name X Y	A 40	0.900 0.800 0.700 0.600 0.500		Time (s)		40.00m		

You should now have both Vin and Vout on the same plot.



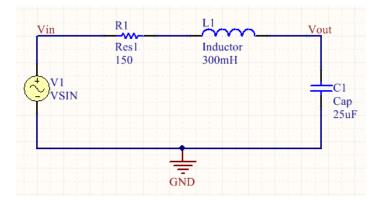
13. We can also make measurements on these waveforms. Click the name "vout" on the right side of the plot (red arrow). This selects just that signal or "wave." Go to Wave → Cursor A and Cursor B to turn on both cursors. You should now see measurements in the Sim Data panel, as well as on the bottom of the chart (purple arrow).



AC Phasor/Fourier Analysis (Sinusoidal Steady-state)

The transient analysis simulation in Altium is not limited to step functions. Any continuous function can be applied as the input. This is simply a matter of changing the input source. In addition, there is a feature for doing Fourier or phasor analysis of sinusoidal steady-state inputs.

14. Create a new project and copy your circuit. Replace the VPWL source with a **VSIN** source.



15. We need to setup the source, just like we did before in transient simulation. Double-click the source to open the Properties dialog. Click the **Edit** button in the Models section (lower right).

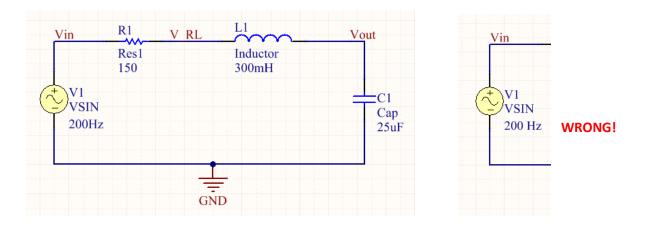
Properties		Paramet	ters			
		Visible	Name	Value	Туре	
Designator	VI Visible Locked		LatestRevisionDate	15-Nov-2004	STRING	
Comment	VSIN 🗸 🗸 Visible		LatestRevisionNote	Corrected Netlist.	STRING	
	<< <>>> Part 1/1 Locked		Note	PCB Footprint - Not requ		
	< < > > Part 1/1 Cocked		PackageReference	Not Applicable	STRING	
escription	Sinusoidal Voltage Source		Published	8-Jun-2000	STRING	
Jnique Id	CUGUPOEU Reset		Publisher	Altium Limited	STRING	
уре	Standard 👻					
ink to Libra. Design Item ID	ry Component Use Vault Component					
-	Choose					
🖉 Library Name	Simulation Sources.IntLib					
✓ Library Name ✓ Table Name	Simulation Sources.IntLib					
_	Simulation Sources.IntLib Validate Link	<u>A</u> dd	Remo <u>v</u> e	lit Add as <u>R</u> ule		
Table Name		Add	Remoye E	iit Add as <u>R</u> ule		
Table Name	Validate Link	Models Name	Type 🛆 Description		ault Item Rev	
_		Models			ault Item Rev	
Table Name	Validate Link 400 Y	Models Name	Type 🛆 Description		ault Item Rev	
Table Name	Validate Link 400 Y 455 0 Degrees V Locked	Models Name	Type 🛆 Description		ault Item Rev	
Table Name	Validate Link 400 Y	Models Name	Type 🛆 Description		ault Item Rev	Revision [Unknowr
Table Name	Validate Link 400 Y 455 0 Degrees V Locked	Models Name	Type 🛆 Description		ault Item Rev	
Table Name	Validate Link 400 Y 455 0 Degrees V Locked Normal V Lock Pins Mirrored	Models Name	Type / Description ▼ Simulation VSIN		ault Item Rev	

16. This brings up a second dialog. Click the **Parameters** tab (red arrow). You should have the dialog below. You can edit the table here or check the "Component parameter" box to display them on the schematic (where you can edit them directly). For this example, only the Frequency will be displayed on the schematic, and the other properties we will leave as their default values.

Note: The Transient and Fourier analyses use the last 6 parameters (from Offset downward).

Sim Model - Voltage So	ource / Sinusoidal	×
Model Kind Paran	neters Port Map	
		Component parameter
DC Magnitude	0	
AC Magnitude	1	
AC Phase	0	
Offset	0	
Amplitude	1	
Frequency	1К	
Delay	0	
Damping Factor	0	
Phase	0	

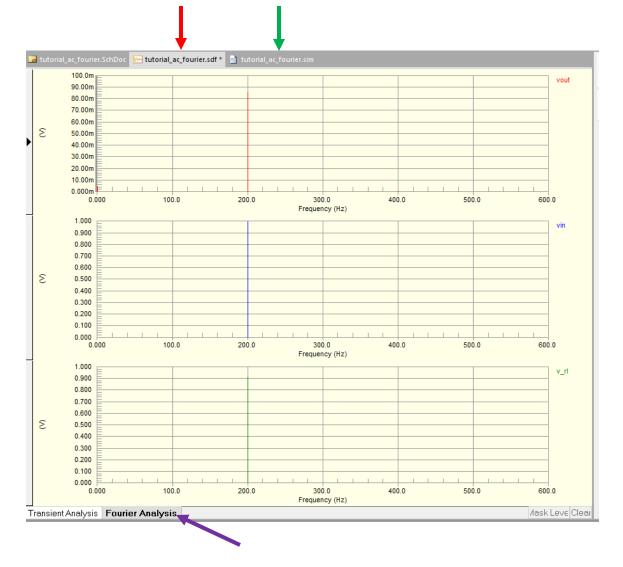
17. Change the frequency to 200 Hz. Note that the unit label "Hz" is not required, but without it, it's not clear to which parameter that value corresponds. DO NOT put a space between the number and the unit! (Altium may still partially simulate but will display an error.)



18. Click the simulation setup icon. Be sure the "Transient Analysis" box is checked, as we are still using that simulation type. Click on **Transient Analysis** to bring up its options. Again, uncheck the "Use Transient Defaults" box. Check the "Enable Fourier" box. This will perform the Fourier/phasor analysis in addition to the transient simulation. Set the Fundamental Frequency to the same frequency as your source (200 Hz) and the Number of Harmonics to 4. (The meaning of this setting will be discussed shortly.) Also, set the Stop Time to 20 ms.

Analyses/Options	Enabled	Transient Analysis Setup				
General Setup		Parameter	Value			
Operating Point Analysis		Transient Start Time	0.000			
Transient Analysis		Transient Stop Time	20.00m			
DC Sweep Analysis		Transient Step Time	20.00u			
AC Small Signal Analysis Noise Analysis		Transient Max Step Time	20.00u			
Pole-Zero Analysis		Use Initial Conditions				
Transfer Function Analysis						
Temperature Sweep		Use Transient Defaults				
Parameter Sweep		Default Cycles Displayed	5			
Monte Carlo Analysis Global Parameters		Default Points Per Cycle	50			
Advanced Options						
		Enable Fourier				
		Fourier Fundamental Frequency	200.0			
		Fourier Number of Harmonics	4			
			Set <u>D</u> efaults			

19. Run the simulation. You should get two (2) new documents that appear as tabs at the top: the simulation waveforms (.sdf) [red arrow] and a text document listing the phasor values at each node (.sim) [green arrow]. The simulation document is split further by the various types of simulation, shown as tabs at the bottom of the screen. Fourier Analysis (purple arrow) should open by default when you select that document (top tab).



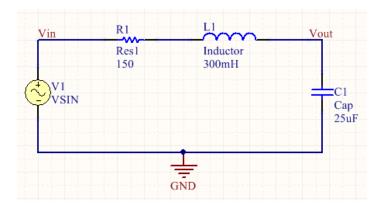
This is a graphical depiction of the amplitude (magnitude) of the node voltage at various frequencies. The number of points (bars, really) shown here depends on the number of "harmonics" we entered in the simulation setup. Altium counts harmonics starting at 0. The harmonic frequency is given by multiplying the fundamental by the harmonic number. So, 0 Hz is the zero-th harmonic, the fundamental (200 Hz in this example) is the first, and then they increase as multiples of the fundamental from there. So for this example, the second is 400 Hz, and the third is 600 Hz. (Other quantities could be plotted as well, like phase, etc. This is not shown.)

20. The other document (.sim) is a listing of the phasor voltage at each node (blue arrow). It shows magnitude (green) and phase in radians (purple) for each harmonic frequency value (red).

No. Har	malysis for vo monics 4, THD	ut: : 1.24529 %, Gri	.dsize: 200, Int	erpolation Deg	ree: 1
		-		Norm. Mag	Norm. Phase
	0.00000E+000				
1	2.00000E+002	8.63313E-002	-1.57054E+002	1.00000E+000	0.0000E+000
2	4.00000E+002	8.98845E-004	1.79726E+002	1.04116E-002	3.36780E+002
3	6.00000E+002	5.89799E-004	-1.78809E+002	6.83181E-003	-2.17554E+001
Fourier a	analysis for vi	n:			
	-	: 9.52534E-014 %	, Gridsize: 200	, Interpolation	n Degree: 1
		Magnitude		Norm. Mag	Norm. Phase
		-3.46390E-016			
		9.99947E-001			
2	4.00000E+002	8.08820E-016	-1.62078E+002	8.08863E-016	-1.62078E+002
3	6.00000E+002	5.03026E-016	-1.67322E+002	5.03052E-016	-1.67322E+002
Fourier a	analysis for @v	1[0]:			
	-	: 4707.54 %, Gri	dsize: 200, Int	erpolation Deg	ree: 1
	Frequency		Phase	Norm. Mag	Norm. Phase
		5.28315E-004			
0	2.00000E+002	2.82632E-005	-2.02027E+000	1.00000E+000	0.0000E+000
		1.33050E-003			
1	4.00000E+002		1 4200551002	2.63037E-002	1.45015E+002
1 2		7.43426E-007	1.429952+002		
1 2 3			1.429952+002		
1 2 3 Fourier a	6.00000E+002 analysis for v1			terpolation Dec	gree: 1

AC Sweep Simulation

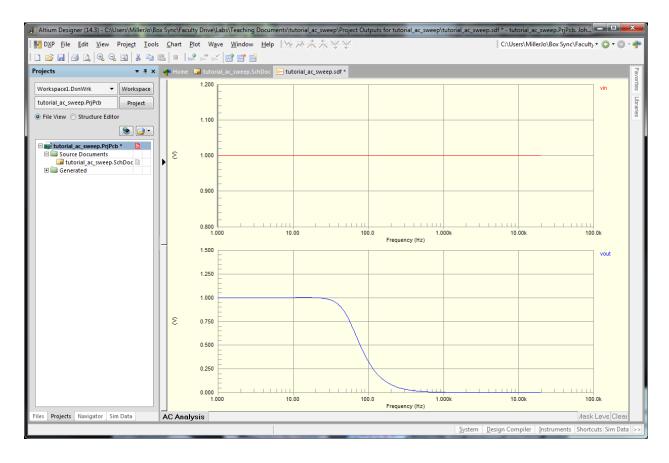
21. Use the same circuit and source (VSIN) as in the previous (AC Fourier) section.
 Note: AC sweep uses the first 3 parameters of the VSIN model (DC Mag, AC Mag, and AC Phase). Frequency is specified as part of the simulation setup (see next step).



22. Click the simulation setup icon. Check the **AC Small Signal Analysis** box (sometimes referred to as an "AC sweep") and click on the name. Setup the analysis as shown below. (Note: You can run all of the various simulations simultaneously if you want.)

Analyses/Options	Enabled		AC Small Signal An	alysis Setup
General Setup			Parameter	Value
Operating Point Analysis			Start Frequency	1.000
Fransient Analysis		=	Stop Frequency	20.00k
DC Sweep Analysis			Sweep Type	Decade
AC Small Signal Analysis	✓			
Noise Analysis			Test Points	100
Pole-Zero Analysis				
Transfer Function Analysis Temperature Sweep		Ŧ	Total Test Points	431

23. Your result should look like this. This plot shows how the voltages vary as frequency changes.



- 24. Very often when doing AC sweep analysis, we are looking for a Bode plot. This is a plot of the magnitude (in dB) and phase of the ratio of output-to-input. Altium can do this for us in a few clicks. Click on the name of each wave and select **Remove Wave** to delete it from the plot (should have two empty plots left).
- 25. Right-click the top plot and choose **Add Wave to Plot...** In the dialog, select **Magnitude (dB)** and create the expression "vout/vin" either by typing it directly or clicking the corresponding elements in the dialog (i.e. click "vout", then "/", then "vin"). Click Create.

Add Wave To Plot	
Wave Setup Waveforms c1[i] c1[p] frequency l1[i]	Functions () + - *
1[p] net1_1 r1[i] r1[p] ∨1#branch ∨1[p]	/ UNARY() Complex Functions Magnitude Magnitude (dB)
∨1[z] <mark>vin</mark> vout	 Real Imaginary Phase (Deg) Phase (Rad) Group Delay
Expression vout/vin	Add to new Y axis
Create Cancel	

26. Do the same thing in the bottom plot, but choose **Phase (Deg)** (expression is still "vout/vin"). You should end up with the following.

