MULTISIM DEMO 7.2: INTRODUCTION TO AC ANALYSIS

In this section, we'll introduce AC Analysis in Multisim. This is perhaps one of the most useful Analyses that Multisim offers, and we'll use it in many of the problems encountered later.

AC Analysis is simply the sweeping of the frequency of the circuit and plotting the values, be they voltages, currents, etc... at each frequency. To get a feel for this type of Analysis we will analyze the AC circuit shown in Fig. 7.2.1 below as a practice problem.



In order to obtain an AC_VOLTAGE source, go to:

Group:SourcesFamily:SIGNAL_VOLTAGE_SOURCESComponent:AC_VOLTAGE

Open up the AC_VOLTAGE source control window, and insert the values shown in Fig. 7.2.2 on the next page...yes there are a lot of fields which you can enter values in. We want to draw our attention to two in particular, however. At the top, is Voltage (Pk) field. This is the amplitude of the sine wave in *Transient Analysis* and the *Interactive Simulation*. Midway down the window there is a field called AC Analysis Magnitude. This is the magnitude of the AC_VOLTAGE source in *AC Analysis*. These are two very different things. Since we will be running an AC Analysis, we need to set the AC Analysis Magnitude to what we want. (You can set Voltage (Pk) to whatever you want; it won't make a difference to us for this simulation). You can also set the phase of the source in AC Analysis through the field called AC Analysis Field, however, as we see in Fig. 7.2.1, the phase of the source is 0° so we can leave it as it is anyways.

Frequency of the source doesn't matter. This is because the AC Analysis will sweep the frequency anyways so any frequency assigned to the AC_VOLTAGE source will have no play in the AC Analysis. The frequency will, of course, influence Transient Analyses and other similar simulations

AC_VOLTAGE	×	
Label Display Value Fault Pins	User Fields	
Voltage (Pk):	10 V -	
Voltage Offset:	0 V •	
Frequency (F):	1 kHz 📩	
Time Delay:	0 sec 🗼	
Damping Factor (1/sec):	0	
Phase:	0 •	
AC Analysis Magnitude:	10 V •	
AC Analysis Phase:	0 •	
Distortion Frequency 1 Magnitude:	0 V •	
Distortion Frequency 1 Phase:	0 •	
Distortion Frequency 2 Magnitude:	0 V •	
Distortion Frequency 2 Phase:	0 •	
Tolerance:	0 %	
Replace OK	Cancel Info Help	
Figure 7.2.2 Changing the		
rigure <i>1.2.2</i> Changing the	AC_VOLTAGE values.	

With the AC_VOLTAGE source set up properly, we can build the rest of the circuit. Once completed, it should resemble Fig. 7.2.3 below.



Now we are ready to run an AC Analysis. To do this, go to the same menu you would go to open a Transient Analysis, the Dot to us shown in Fig. 7.2.4), or you can go to Simulate>Analyses>AC Analysis.

🚰 Ele Edit	View Place MCU Simulate Transfe	r <u>T</u> ools <u>R</u> eports <u>O</u> ptions <u>W</u> indo	w <u>H</u> elp		_ 8
🗅 🥩 🖆 (u ⊕ L : h n c ∽ c E 0	L Q Q Q 'e S A	🛚 🗔 🔽 🕅 🧊 🗔 🖏 📣 🔲 In Use	e List 🔹 🐐 ?	
+ 4₩ -1}-	< t> 天 哭 哭 み & 回 thinks ■	Y A 3 2 F 🗊 🗆	DC Operating Point		
			AC Analysis		
		⊐ a a a a a a •• • •• •	Transient Analysis		
0	1 2	3 4	Fourier Analysis	8	
			Noise Analysis		
A			Noise Figure Analysis	A	
	50		Distortion Analysis		
11	2 R2	1	DC Sweep		
П	3.3kΩ		Sensitivity		
	V1		Parameter Sweep		
°	10 Vpk		Temperature Sweep	в	
		$\geq R1$ $\geq 4.7k0$ $= 10F$	Pole Zero		
Н	Ū		Transfer Function		
			Worst Case		
C		0	Monte Carlo	C	
			Trace Width Analysis		
Н		<u> </u>	Batched Analysis		
			Stop Analysis		
D			Stop Analysis	D	
Н				-	
<u> </u>					
S_3 6	_1 1 6_opt1 9_1 8 8_1 5	4 * 😰 8_2 * 😰 7_2 *			•
×					
Multisim -	- 2008-05-02 09:36:23				
Viev					
reet					
adsh					
Results No	ets Components PCB Lavers				

Once the AC Analysis window appears, as shown in Fig. 7.2.5, you can adjust several fields. The Start Frequency (FSTART) is the frequency at which the simulation starts, and the Stop Frequency (FSTOP) is, you guessed it, the frequency where the simulation stops. Changing the Number of points per decade will change the resolution of the simulation accordingly. For the simulation which we want, set FSTART to 100 Hz, and FSTOP to 100 MHz. Leave the Number of points per decade at 10.

The Analysis
Frequency Parameters Output Analysis Options Summary
Start frequency (FSTART) 100 Hz ▼ Stop frequency (FSTOP) 100 MHz ▼ Sweep type Decade ▼ Number of points per decade 10 ↓ Vertical scale Logarithmic ▼
Simulate OK Cancel Help
Figure 7.2.5 AC Analysis setup window.

Under the Output tab (Fig. 7.2.6), you can select the variables which we want to analyze, just like you would in a DC Sweep or Transient Analysis. For us, we want to plot the voltages first, so enter V(1) and V(2) from the circuit in Fig. 7.2.3.

Frequency Para	meters Output Analysi	is Options Summary	Coloria durati blan far ana		
variables in c			Selected variables for ana		
All variables			All Variables		
I(1) I(v1)			V(1) V(2)		
		Add			
	<u> </u>	Remove			
		Edit Expression			
Filter Unsele	cted Variables	Add Expression	Filter selected variable	es	
- More Option	s				
Add devi	e/model parameter		how all device parameters	at end	
		l l	n sinulauon in the dualt tha	",	
Delet	e selected variable		Select variables to save		
			1.5		
	Simulate	ОК	Cancel H	Help	

Once you're ready, press Simulate. The resultant plot should be similar to that in Fig. 7.2.7 below. Notice that there are cursors, which you can enable with the button which you can use to measure the plots just like you would in a Transient Analysis.

AC Analysis			🖻 Grapher View					
	V(2)	V(1)	File Edit View Tools					
x1	58.5555k	58.5555k) 🚅 🔲 🚑 🖪 🗼 🖻 🖻			🛯 🔊 📩 🗹 🖉	x 🛼 🐘 🗖	
y1	10.0000	8.4023m						
x2	100.0000	100.0000	AC Analysis					
y2	10.0000	1.9047m			72			
dx	-58.4555k	-58.4555k		A.(Analysia			
dy	0.0000	-6.4976m	V	A	Allarysis			
1/dx	-17.1070µ	-17.1070µ	100		Set X	_Value		
1/dy		-153.9041	10-14	Δ <u>Λ</u> Δ	Δ- Set V	Value - >	Δ	Δ
min x	100.0000	100.0000	u ¹ 1		Sect	_value _>		
max x	100.000M	100.0000M	2 100m		Set Y	_Value <=		
min y	10.0000	4.8229µ	E 10m		Gate	next V MAY ->		
max y	10.0000	5.2254	20 1m		00 10	D NEXT Y_MAX =>		
offset x	0.0000	0.0000	2 100		Goto	next Y_MAX <=		
offset y	0.0000	0.0000	10		Goto	next Y_MIN =>		
			тор		Gett	next V MIN <=		
r			100 11-	101-	0010	THEAT T_IVILIA <=		1003.6
		8	100 1K	TOK	Selec	t Trace ID	/101	1001
		V(1)			Free	Coloret Marilia		
x1	100.0000	100.0000	100		Show	/ Select Ivlarks		
y1	0.0000	89.9814			Hide	Select Marks		
x2	100.0000	100.0000	50-					
¥2	0.0000	89.9814	െ					
dx	0.0000	0.0000	deg					
dy	0.0000	0.0000	5 0- <u>4</u>	Δ	Δ Δ	Δ Δ	<u>م</u>	
1/dx			198					
1/dy			a					
min x	100.0000	100.0000	-50-					
max x	100.0000M	_00.000M						
max v	0.0000	-90.0000	-100					
offset x	0.0000	0.0000	100 1k	10k	100k	1M	10M	100M
offset v	0.0000	0.0000			E	-)		
	0.0000	0.0000			Frequency (H	2)		
			elected Diagram:AC Analysis					11
				c · ·				
	Figure 7.2.7 AC Analysis Output of circuit.							

Now let's say we wanted to find the peak of the blue trace. By default, all cursors monitor Trace 1. We need to change that, so after you have dragged Cursor 1 out into a visible portion of the graph, right click on it, and click on Select Trace ID. This will bring up the window shown in Fig 7.2.8 below. Under the pull-down menu, change the Trace value to V(1). (Note that if you entered your variables in the opposite order in the AC Analysis set-up window, you won't have to do this.)

Select Trace ID on Crosshair_1
ОК
Trace: V(1) Cancel
.::
Figure 7.2.8 Selecting a new Trace ID.

Now with the proper trace ID selected, you can right click on the cursor and click on Go to next Y_MAX (There will be a => or <= depending on which side of the maximum you are on...pick the proper direction). For example in Fig. 7.2.7, the cursor (hidden behind its window) is to the right of the peak in the blue trace, so you would click on Go to next $Y_MAX <=$. The cursor will move to the peak of the V(1) trace, as shown in Fig. 7.2.9. Then in the measurement panels to the left of the screen you can observe where the peak is, and what the magnitude of the peak (the voltage) is. We see that the peak occurs at about 5.012 kHz.

Why is there a peak in the Magnitude trace? It is because of resonance! As you learned in Chapter 6 of the textbook and in this manual, the inductor, capacitor, and the 4.7 k Ω resistor form a parallel RLC, which has a resonance peak at the inverse of the square root of the product of LC.

$$f_0 = \frac{1}{2\pi\sqrt{1 \times 10^{-3} \cdot 1 \times 10^{-6}}} = 5.033 \text{ KHz}$$

This value is pretty close to what we see in the simulation (5.012 KHz) shown in Fig. 7.2.9 below. So while this chapter isn't really supposed to go into resonance, it's still pretty cool to see it pop up during our AC Analysis.

So we've measured voltage in the AC Analysis. Can we measure current as well? Of course. In fact, all we have to do is add 0V DC sources like we did in DC circuits. Add a 0V DC source to the branches of the circuit containing the 4.7 k Ω resistor, the capacitor, and the inductor. In order to ensure clarity for this example, rename the three voltage sources in series with the resistor, capacitor, and inductor to "Res", "Cap" and "Ind" respectively. As a result, your circuit should look like Fig. 7.2.10 on the next page.





Now when we get ready to re-run the AC Analysis in order to measure currents, replace the voltages V(1) and V(3) with the currents I(res), I(cap), and I(ind) as the variables selected for analysis. When you are ready, press Simulate. You will produce a plot similar to that shown in Fig. 7.2.11 on the next page.

You can also create equations based on variables for analysis. Let's measure the voltage across R1. To do this, under the Output tab on the AC Analysis set-up window, insert the Equation V(2) - V(1) and then simulate. The result should be similar to Fig. 7.2.12. The resolution of the plot looks bad, however. Under the Frequency Parameters tab of the AC Analysis set-up window, adjust the Number of points per decade to 50. Then re-simulate. Your plots should now look like Fig. 7.2.13.









Measurement Probes in AC

Now before we end this Multisim Demo, let's look at one more thing. In DC circuits we were able to use Measurement Probes in order to get some basic values. We can also use probes in the AC domain. Some of the default parameters measured by probes, however, are not suitable for AC, so you have to de-select them as shown in Probe Properties window in Fig. 7.2.14. Place a few probes onto the schematic as shown in Fig. 7.2.15 Run the Interactive Simulation by pressing F5, and then observe the values which appear.



