Introduction to MultiSim – Part 2

Prepared by: Mohamad Eid Summer 2007

The purpose of this tutorial is to demonstrate the use of MultiSim to simulate operational amplifier circuits (uA741 OpAmp), such as inverting and non-inverting amplifiers, filters, and oscillators. The circuit shown in Figure 1 is used as a simulation example in the current tutorial.



1) Start the MultiSim program as shown in the previous tutorial (For Windows users the default location can be found by clicking: Start ->All Programs -> Electronics Workbench -> DesignSuite Freeware Edition 9 -> MultiSim 9).

2) You will be creating a new schematic and simulation so choose File->SaveAs, navigate to or create a directory where you can save this schematic and simulation then fill in the filename as shown below. (I called this one tut1.) Click **OK** when you have navigated to the proper directory and entered a name for the project.

3) You should now have a blank schematic. Start placing components by selecting Place->Component from the menu bar. You will see another dialog box as shown. Start with the OpAmp so pick *Analog Components* in the drop down menu as shown:

🀲 Select a Componen	t in the second s		
Database:	Component	Symbol (ANSI)	- 04 1
Master Database 🛛 💌	AC_VOLTAGE		
Group:	AM_VOLTAGE		Liose
🕈 Sources 💌	EXPONENTIAL VOLTAGE	Ť	Search
T Sources	FM_VOLTAGE	$\begin{pmatrix} 1\\ \\ \end{pmatrix}$	Detail Report
H>→ Diodes	PIECEWISE_LINEAR_V0	$ \downarrow $	Model
-⊀ Transistors 1≫ Analog 1≫ TTL	THERMAL_NOISE		Help
CMOS		-	
Misc Digital		Function:	
		ne rollage obaice	
MISE Misc			
Y RF			
Electro_Mechanical		Madel Manuf MDr.	
Ladder_Diagrams	l,	Generic/VACS	•
		Footprint Manuf.\Type:	_
Components: 8			11.

4) You should see a new dialog box. Select *OPAMP* then *LM741CN* as shown below then click **OK**.

Database:	Component:	Symbol (ANSI)
Select a Lomponent Database: Master Database aroup: Analog amily: ANALOG_VIRTUAL OPAMP OPAMP_NORTON COMPARATOR WIDEBAND_AMPS SPECIAL_FUNCTION	Component: ► LM725AH LM725AH/883 LM725CH LM725CH LM725H LM725H LM725H LM725H LM725H LM725H LM721AJ-14/883 LM741CH LM741CH LM741CH LM741CH LM741H LM741H LM741H LM741H LM741J Mo LM741J/883 LM741J/883 LM741W/883 Na LM741W/883 Na	Symbol (ANSI)
	LM747AJ LM748CH LM748CJ LM748CJ LM748CN LM74011 LM759CH IM759CP	auonainLimr41C_2 atprint Manuf, \Type: aneric\DIP-8(NOSE)

5) Place the OpAmp on the schematic. Next you must add a couple of power supply connections. You will add a DC source by again selecting Place->Component. Only this time pick *SOURCES* from the drop down menu. Then select *POWER_SOURCES* and *DC POWER* as shown below then click **OK**.

🍩 Select a Component			
Database:	Component:	Symbol (ANSI)	OK
Master Database	AC_POWER		
Group:	DC_POWER		<u><u>C</u>lose</u>
🔹 Sources	DGND		Search
Family:	GRUUND		
POWER_SOURCES	INUN_IDEAL_BATTERY		Detail Report
SIGNAL_VOLTAGE_SO	U THREE_PHASE_DELTA	*	<u>M</u> odel
(SIGNAL_CURRENT_SC			11-1-
CONTROL FUNCTION	B., VDD		
CONTROLLED VOLTA	- VFF		
	N VSS	Function	
		DC Voltage Source	
		DE Vollage Source	
		Model Manuf. ND:	
		Generic\VDCP	
		Ecotoriot Manuf \Tupe:	
		Comparison of the contract of the participation of	
Components: 11			1

6) Put the battery (*DC power source*) on the schematic somewhat above the OpAmp as shown below.

15-27-41	11 II 1	1.6.1	25			10.1	D- E-	E•Ⅲ	- B- E) 第・日	- E+		
110		181	1	11	11	1.1.1.1.1.1.1		- 1	11.124	11		 111	 1
1													
						主族							
													T.
						100 14	EF +						100
						2	41						
	-					× .	1	144					1
						P.		192					
	100												
	E												
													10.00
2	- I												- P
	Paul											 	
	Andrease 1												
er er pelakel													

7) Double click on the "12V". Change the voltage from 12 to 18 then click **OK**.

Voltage (V):	18	V ÷
AC Analysis Magnitude:	0	V ÷
AC Analysis Phase:	0	Deg
Distortion Frequency 1 Magnitude:	0	V ÷
Distortion Frequency 1 Phase:	0	Deg
Distortion Frequency 2 Magnitude:	0	V ÷
Distortion Frequency 2 Phase:	0	Deg

8) Place the rest of the components on the schematic as shown below. Use *Place->Wire* to add wires for connecting components together. Resistors are under the Basic components drop down menu. Note you must specify "k_" in the filter to locate the 10K resistors. Note *GROUND* symbols are at SOURCES->POWER_SOURCES. Your completed schematic should resemble the one below.

9) Be sure all components are connected as shown above. Now you will name the input and output signals to make them easier to locate in simulation. First highlight the wires above the 5V 1KHz source V3. Do this by placing the cursor on the red wire and left clicking once. Now right click once and select "properties" from the popup menu. Change the net name to Vin as shown on the next page then click **OK**. Use the same method to change the output net name to "Vout".



10) You will now run a simulation. First select Simulate->Analyses->Transient Analysis from the top menu bar. Change the End Stop Time (*TSTOP*) to 0.002 as shown below.

Automatically determi	ne initial conditions		•	Reset to <u>d</u> efaul
Parameters				
Start time (TSTART)	0	Sec		
End time (TSTOP)	0.002	Sec		
Maximum time step	o settings (TMAX)			
C Minimum nur	nber of time points	99		
C Maximum tim	e step (TMAX)	1e-005	Sec	
		I construction		

11) Next choose the *Output* page and select the two signals you want to see on the output. Your dialog box should now look like the one on the next page.

Variables in circuit		Selected variables	for analysis
All variables		All variables	<u> </u>
\$2 \$3 vv1#branch vv2#branch vv3#branch	>> <u>A</u> dd << <u>H</u> emov	\$vout	
Filter Unselected Variab	les		

12) At this point you are ready to simulate. Left click on the **Simulate** button to run the simulation. You should get a result window that looks like the one below.



Now run an AC analysis to make a Bode plot of the response of your circuit. For an ideal Op Amp the gain would be always be 1. In the real world, capacitive and inductive

effects at higher frequencies cause the gain (and phase) to shift. The Bode plot is a graph of gain and a graph of phase shift relative to input frequency.

13) Select Simulate->Analyses->AC Analysis as shown.

ani - Lo	ucij											
⊻iew	Place	Simulate	Tr <u>a</u> nsfer	<u>T</u> ools	<u>R</u> eports	Opti	ons	<u>W</u> in	dow	<u>H</u> elp		
i 🗟	X 🛙	🖋 <u>R</u> un			F5	R	0	Q			: =	8
≰ ⊅≻	35 T	Paus	е		F6	E			1		Ξ	
	<u>[</u>	Instr	uments									
2 mg	-5 -	Inter	active Simul	lation Se	ttings	: :	· · · ·	· · · ·	· · ·	· · · ·	· · · ·	· · · · ·
لقاح		Digita	al Simulation	Setting	5		· · · ·	· · ·	· · · ·	· · · ·	· · · · ·	
L		Anal	yses			۱.	D	C Ope	rating	g Point		
		🗒 Post	processor				A	C Ana	lysis			– ∨1. ≟181
		Si <u>m</u> u	lation Error I	Log/Aud	it Trail		Tr	ansie	nt An	alysis.		

14) Make sure the frequency dialog box looks as shown below then click the *Output* tab. Make Vout the only output then click the **Simulate** button as before.

Start frequency (FSTART)	1	Hz	-	Reset to default
Stop frequency (FSTOP)	10	GHz	_	
Sweep type	Decade	<u> </u>		
Number of points per decade	10	-		
/ertical scale	Decibel	N		
		N		

Frequency Parameters Output Analysis Options S	uninary
Variables in circuit	Selected variables for analysis
All variables	All variables
\$1 \$2 \$3 \$vin	\$vout

15) The simulation results should appear as below. Note the top graph is gain in decibels (0 db is the same as unity gain or 1). See how the gain rolls off starting around 1 MHz. The bottom graph is phase shift. Since this is an inverting configuration, expect the phase shift to be 180 degrees. But notice how it drops to around 90 degrees by 10MHz then rolls down to around 0 by 10GHz.



16) Put the cursor in the graph and right click to get the menu show. From there you can turn the grids on or off, add cursors, etc. You can also choose **Properties** and change the axes of the graphs. Use File->Print to print the Bode plot.

Examples

Simulate the following circuits:

a) Average Amplifier:



b) Bandpass Filter:

