Multisim Manual

Multisim is the schematic capture and simulation application of National Instruments Circuit Design Suite, a suite of EDA (Electronic Design Automation) tools. It is similar to PSpice, but it is more easy to use in practical sense and has lots of features to make circuit drawing/simulating, a really simple task. Here is window of multisim, as it appears first time when you start the software.



- 1. The **Menu Bar** is where you find commands for all functions.
- 2. The **Design Toolbox** lets you navigate through the different types of files in a project (schematics, PCBs, reports), view a schematic's hierarchy and show or hide different layers.
- 3. The **Component** toolbar contains buttons that let you select components from the Multisim databases for placement in your schematic.
- 4. The **Standard** toolbar contains buttons for commonly-performed functions such as Save, Print, Cut, and Paste.
- 5. The **View** toolbar contains buttons for modifying the way the screen is displayed.
- 6. The **Simulation** toolbar contains buttons for starting, stopping, and other simulation functions.
- 7. The Main toolbar contains buttons for common Multisim functions.
- 8. The **In Use List** contains a list of all components used in the design.
- 9. The **Instruments** toolbar contains buttons for each instrument.
- 10. Scroll Left -right is to ensure ease in handling larger designs.
- 11. The **Circuit Window** (or workspace) is where you build your circuit.
- 12. Active tab indicates the current active circuit window.

Let's take an example of a RC circuit. We will simulate the circuit to perform the transient analysis. At any point of time you can click **F1** for help on the tool.

Please follow the steps:

- 1. Select Start»All Programs» National Instruments» Circuit Design Suite 10.1» Multisim 10.1. A blank file opens on the workspace called Circuit1.
- 2. Select **File**» **Save As** to display a standard Windows Save dialog.
- 3. Select Place» Component to display the Select a Component browser, navigate to the group Sources and click on POWER_SOURCES. Then choose the Family: AC_POWER option. The component appears as a "ghost" on the cursor. (Once you have selected the desired Group and Family, start typing the component's name in the browser's Component field. As you type, the string appears in the Searching field at the bottom of the browser.)

Circuit2 - Multisim - [Circuit2]				F X
Eile Edit View Place MCU Simulate Transfer	Tools Reports Options Window Help			_ 8 ×
口戸戸日日本 / 日日日日 日 ④				
+ ··· +> + + +> 12 12 40 10 10 10 10 10 10 10 10 10 10 10 10 10	· · · · · · · · · · · · · · · · · · ·	I (]= ^t = +≡	11	
Design Toolbox				
	🏶 Select a Component			
	Database: Component:	Symbol (ANSI)		313) 210
Creuit ☐ Creui	Master Database Grup: Grup: Sources Panily: Select all families Select all families Select all families Stanua_voltrace_source Stanua_voltrace_source Stanua_voltrace_source CommonLeb_voltrace_source CommonLeb_voltrace_source CommonLeb_voltrace_source CommonLeb_voltrace_source Vec Vec Vec Ves	Function: AC Power Source Model manuf./ID: [Generic/JACP	Coce Search Vew Model Help	
Uservela Visibility Project View Bb caruet # 100 pp				-
Connectivity Error: For Net 1, Pin 1 of component R1 is		Hyperlink:		*
Connectivity Error: For Net 1, Pin 1 of component XSC1				
Connectivity Error: For Net 0, Pin 2 of component XSC1	Components: 11 Searching:		11	
		m		- · -
Results Nets Components PCB Layers Simulation				
				FEFFE
💿 🖻 🖉 🖉 😵 🐨 🕼 Circuit2 - Multisi	m 📗 🕹 Page Load Error - M 📗 🖨 Inbox for wm	ohit@ 🛛 🔁 Getting Started with 🖉 🔯 Docum	enti - Micro 🧌 🙀	4:08 PM Vednesday 1/21/2009

4. Move the cursor to the bottom-right of the workspace and left-click to place the component. Similarly, find the other components and place them. When a part is on the workspace and you want to place the same part again, highlight it and select Edit»Copy, then Edit» Paste. You can also select it from the In Use List and click to place it on the workspace. Press Ctrl+R if you want to rotate the component.The oscilloscope is obtained (by dragging it) from the component window on the right hand side of main window.



Multisim



- All components have pins that you use to wire them to other components or instruments. As soon as your cursor is over a pin, Multisim knows you want to wire and the pointer changes to a crosshair. You can also use ctrl + q for wiring the circuit. Don't forget to add Ground to the circuit (available in Sources option in Place>>Component).
- Choose the type of analysis you want to perform by clicking Simulate>> Analyses>>. Here, we have chosen transient analysis.



7. Select **Simulate**» **Analyses**»**Transient Analysis** and click on the **Output** tab. Add I(v1) to the right column by first clicking on I(v1) in left column and then pressing **Add** tab.

Circuit1 - Multisim - [Circuit1 *]							- 7 - 3
Eile Edit View Place MCU	J <u>S</u> imulate Tr <u>a</u> nsfer <u>T</u> ools	<u>Reports</u> Options <u>Window</u>	<u>t</u> elp				_ 8 ×
D 📂 🛎 🖫 🖨 🖪 🐇 🐚		. @, <mark>``</mark> ≣ 📽 🔲 A 🛛	- 🛗 🕼 🗔 🖏 🛤	In Use List 🔻	* ?		
	🔐 🗵 🖽 HISK 💻 Y 🕀	T 2 1 2 2 1 2 2 1 €	II ■ ● 5≣ Ç≣ *3	- A B			
Design Toolbox	• • • • • • • • • • • • • • • • • • • •	1	- 3 4 4	· · · · · 5 · · · · · · · · ·	• · · · · · · · · · · · · · · · · · · ·	8	18
		Transient Analysis					1000 A
Circuit1	A	Analysis Parameters Output	Analysis Options Summary			×	00 **
Circuit1-Breadboard		Variables in circuit		Selected variables for anal	ysis		
101		. All variables	•	All variables	■		
		V(2)		I(V1)			1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1
				V(1)			
						9	
			> Add :	>			
			< Remove	<			=
	C			-		с.	
			Edit Expression				
		Cites Hands at a Mariables	Add Expression				281
		Filler Unselected variables.			Bere		
		Harr Oakara					1
	· · · · · · · · · · · · · · · · · · ·	. More Options		Show all device parameters a	tend		23
		Add device/model param	eter	of simulation in the audit trail			<u></u>
		Delete selected varia	ble	Select variables to save			50
	E					€	
		:	šimulate OK	Cancel	Help		• •
	H						\$
							a
	F						-
							-
Hierarchy Visibility Project View	Circuit1 * 3D View -	Circuit1-Breadboard					• •
				-		Tran: 10.000 ms	
👝 🖸 🖬 🌽 😻 🌾	🙂 Gmail - Inbox (1)	Multisim_software	Manual for multisi	🔁 Getting Started with	4 Circuit1 - Multisim		11:43 PM
	S Inbox for wmchit@						Wednesday
							1/21/2009

- 8. Also choose the time (from transient Frequency parameters).
- 9. Click on Simulate tab. The output window appears which consists of tab for oscillator ouput as well as transient waveforms. Different colored waves can be viewed by choosing the color of respective wire of the electrical quantity (voltage and current). Right click on wire in the circuit and then click on Color segment... to choose the color of wire and thus the waveform color (after simulation).

Principles of Electrical Engineering Lab 1





10. With multisim, you can create 3D breadboard with components placed. You can explore yourself this option by clicking on show breadboard tab on main toolbar. Figure below shows example of 3D breadboard created in multisim.

