NI Multisim and Ultiboard

BME Senior Design Fall 2011

About Multisim

- A schematic capture and simulation tool for electrical circuit design.
- Application allows for placement of circuit parts, simulation of circuits, and transfer to other programs for PCB layout
- How to Access Multisim
 - Start > All Programs > National Instruments > Circuit Design Suite 11.0



1 Menu Bar 2 Design Toolbox 3 Component Toolbar 4 Standard Toolbar 5 View Toolbar 6 Simulation Toolbar 7 Main Toolbar 8 In Use List 9 Instruments Toolbar 10 Scroll Left/Right 11 Circuit Window 12 Spreadsheet View 13 Active Tab

Multisim – Placing a Component

Database: Compo	onent:		Symbol (ANSI)	011
Master Database 👻 270		Ω		OK
Group: 270m				Close
100° Basic				Search
amily: 270k				Detail report
All Select all families				View model
V BASIC_VIRTUAL		E	Save unique component on placement	Help
BD 3D VIRTUAL		C	omponent type:	
m RPACK			<no type=""></no>	•
+ SWITCH		Т	blerance(%):	
3E TRANSFORMER				•
SE NON LINEAR TRANSF				
-121-7 LOAD =		M	odel manufacturer/ID:	
- RELAY			Schene / VINTOAL_NESISTAINCE	
SOCKETS		F	potprint manufacturer/type:	
SCH CAP SYMS			<no footprint=""></no>	
		I	PC-7351 / Chip-R0201 PC-7351 / Chip-R0402	-
		I	PC-7351 / Chip-R0603	-
			PC-7351 / Chip-R0805 PC-7351 / Chip-R1206	
		I	PC-7351 / Chip-R1210	
		I	PC-7351 / Chip-R2010	-
		H	pc_7351 / Chin_D 2512 yperlink:	
VARIABLE_INDUCTOR +		-		

- On top of screen select Place > Component to bring up the Select Component box.
- Components are sorted by Database Group Family

 Click OK to select the component then left-click on the workspace to place the component

Multisim – Wiring the Circuit

- Each component has pins that can be used to wire them to other components or instruments
- Multisim will recognize the pins and automatically change the cursor to a cross-hair for wiring
- Left-click on the pin of the first component, then left-click on the pin of the component to wire to.
 - Multisim will automatically place the wire between the two components that were selected
- New components can be directly placed onto an already wired connection and Multisim will automatically connect the new part

Multisim – Virtual Instruments



- Multisim offers several virtual instruments which can be used to simulate and analyze your circuit
 - It is important to perform thorough analysis of your circuit in simulation to save you time and money when building your PCB
- Found in two places on the user interface.
 - Tool bar on the right hand side
 - Click Simulate > Instruments

Virtual Instruments - Oscilloscope

VCC

- The oscilloscope instrument can be wired directly onto your circuit design.
- Double click on the oscilloscope graphic to show the dialogue box and output.
- To run the simulation click
 Simulate > Run or the green Run arrow



Virtual Instruments - Grapher

- This tool is used to display all of the Multisim analysis in graph form.
- > This graph can be edited, saved, and exported
- To access the grapher first run the simulation
- Click view > grapher



Multisim – Bill of Materials

- A summary of components within the design
- Only lists "real" components (excludes virtual instruments etc.)
- Click Reports > Bill of Materials

🖬 😂 🖻 🍌 🔶 👫 🤫						
	Quantity	Description	RefDes	Package		
1	1	SEVEN_SEG_DECIMAL_COM_A_BLUE	U1	Generic\75EG8		
2	1	74LS, 74LS47N	U3	IPC-2221A/222		
3	1	CAP_ELECTROLIT, 1uF	⊂1	IPC-7351\Chip		
4	1	CAP_ELECTROLIT, 10nF	C2	IPC-7351\Chip		
5	1	CAP_ELECTROLIT, 100uF	СЗ	IPC-7351\Chip		
6	1	CONNECTORS, HDR1X4	J3	Generic\HDR1>		
7	2	SWITCH, SPDT	J1, J2	Generic\SPDT		
8	1	LED_blue	LED1	Ultiboard\LED9		
9	1	RESISTOR, 1kΩ 5%	R3	IPC-7351\Chip		
10	1	POTENTIOMETER, 50k	R2	Generic\LIN_P(
11	1	RESISTOR, 200Ω 5%	R1	IPC-7351\Chip		
12	1	RPACK_VARIABLE_2X7, 180 Ohm	R4	Generic\DIP-14		
13	1	OPAMP, OP37AJ	U4	Generic\TO-99		

About Ultiboard

- PCB application used to lay out and route wiring to prepare for manufacturing
- How to Access Ultiboard:
 - Start > All Programs> National Instruments > Circuit Design Suite
 I I.0
- Files can be transferred from Multisim directly to Ultiboard
 - In Multisim select Transfer > Transfer to Ultiboard > Transfer to Ultiboard 11.0
- This will create a NetList of all components (not virtual)
 - Shows the nets, layers, and components
- Click OK to import the NetList to Ultiboard



- 1 Menu Bar
- 2 Standard Toolbar
- 3 Select Toolbar
- 4 Draw Settings
- Toolbar
- 5 View Toolbar
- 6 Main Toolbar
- 7 Autoroute Toolbar
- 8 Status Bar
- 9 Workspace
- 10 Spreadsheet View
- 11 Design Toolbox
- 12 3D Preview
- 13 Birds Eye View

Ultiboard – Design Toolbox: Layers Tab

- Double click on a layer work on it
- Some important layers
 - Board outline controls the shape of the board. Your board should be large enough to fit all of your components and proper wiring
 - Copper top select this layer to place the components on the top of the board

Ultiboard – Editing Board Shape

- Double click on Board Outline in the layers tab
- Click on the board (in yellow) and hit <Delete> on the keyboard to delete the current board
- Two ways to edit the board
 - Board Wizard
 - Tools > Board Wizard
 - Manual
 - To select and draw a shape by hand click Place > Shape
 - Then draw the shape directly on the board

Ultiboard – Placing Parts

- In the design toolbox under the layers tab select the Copper Top layer.
- Click and drag the part to place it on the grid in the Spreadsheet view.
- Several of the lines that are shown may help to place the part correctly
 - Yellow Ratsnest lines: theoretical connections between pins as specified in your Netlist. It is best to try to arrange them so that they are organized and not running through other parts



Ultiboard – Part Placement



Ultiboard - Traces

- Be sure you are working on the Copper Top when making any traces
- Several options for placing traces:
- Manual
 - Place > Line
 - Draw a line in segments to connect one pin to another

Follow-me Trace:

- Place > Follow-Me
- Ultiboard will determine the best trace based on the Ratsnest lines. The user has a little flexibility in drawing.

Autoroute

- Autoroute > Start/Resume Autorouter
- Ultiboard will draw all of the traces automatically

Ultiboard - Autoroute



Ultiboard – Cleaning up



- Before sending to a manufacturer the board should be finalized and cleaned up
- Clear any open-ended traces
 - Edit > Copper Delete > Open Trace Ends
- Delete unused vias
 - Design > Remove unused vias

Ultiboard – 3D imaging

View your circuit board in 3D

Tools > View 3D



References

National Instruments Multisim:

http://www.ni.com/academic/multisimse.htm