The Multisim Highlights Tutorial

There are many things that Multisim just does better than most schematic capture and simulation packages. This tutorial shows you a few of them and is designed to be used with our Multisim Demo (CD or Web version) or by customers who have just purchased Multisim for the first time and want to get acquainted with the software. It’s not a complete User Guide or Help file, although these are available for your convenience on our product and demo CD’s and our Website. Furthermore, although this tutorial shows you some of the reasons we think you will appreciate Multisim, it’s not intended (nor is it long enough) to explain all of the power and flexibility of Multisim. It is meant as an introduction. Some functions are introduced in this tutorial, but explaining them fully is beyond its scope, and so in these cases, reference is made to the User Guide for more detail.

Good luck getting started and we hope you find Multisim to be the most intuitive, powerful, cost-effective design software available. Should you have any questions whatsoever, don’t hesitate to call your Technical Sales Representative at 1-800-263-5552 for assistance.

"It’s Easy to Use"

Ease of use - it’s one of our most important features. Together with exceptional value, this is why Electronics Workbench has more users of electronics design tools than any other EDA vendor. Here’s a quick tour of what makes our Multisim schematic and simulation product so popular: from the Design Bar which gives you shortcuts to the logical steps of the entire design flow, to our intuitive, modeless schematic capture and immediate simulation results.

The Design Bar

Like all the other toolbars in Multisim, the Design Bar can be resized and docked anywhere. By default, it’s found near the top-middle of the screen. The first button of the Design Bar shows and hides the Component Toolbar, which is initially shown and docked to the left, and is made up of a series of intuitively organized Parts Bins.
Placing a Component

Here’s where ease-of-use shows up next. To place a component on the circuit window, click or roll your mouse over a Parts Bin in the Component Toolbar. Clicking the second button down opens the Basic Parts Bin. Now you can click on the type of component you want to use. Choose the Switch Parts Bin, (eight button down on the left of the Basic Parts Bin). This brings up the list of switches. Double-click the Single Pole Single Throw (SPST) switch from the Component List in the middle portion of the Component Browser screen. A ghost of the switch automatically appears at the tip of your cursor. To position the switch, move the cursor to the desired location on the circuit window and click to place it. That’s all there is to it. You can use the figure below as a guide to where on your circuit window you might want to place the components for this simple example. Unlike other schematic capture products, we don’t give you a list of all 16,000 components in our database and force you to scroll to the one you want. After all, do you remember the part numbers of over 16,000 components?

Now, using the same method as you did with the switch, place (using the figure below again as a guide) a battery and a ground connection on the circuit window. These components are in the first parts bin (Sources). A Lamp can be found in the parts bin called Indicators (tenth bin down). Scroll down the indicators to find the lamp identified as (5V_1W). Place the lamp on the circuit window.

We also need a resistor for our simple example circuit. Pull a resistor from the Basic Components (second bin from the top) parts bin. You might notice there are different types of resistors available in Multisim. There are Real resistors (which come only in the “real” purchasable values and include actual footprints or package type for use in PCB layout programs, such as Ultiboard, also from Electronics Workbench) and there are Virtual resistors.

Using the Virtual Resistor provides you with an easy-to-change, purely simulation component that lets you experiment with all the resistor parameters in the early stages of your design. Place a Virtual Resistor on the circuit window. By double clicking on the resistor you will open its parameters dialog box. Change its resistance from 1k ohm to 10 ohms. It’s easy, just type 10 in the value box and change the units from kOhm to Ohms. You may note that a virtual resistor by default appears white where as a Real resistor appears yellow. In the same way, you can change the battery voltage to 5V (several other types of components also have Virtual options for experimenting). We have just placed all the components on the circuit window needed to let you wire together a simple schematic.
Wiring a Schematic

To wire components together, just click on a pin coming from one of the components, and then click on a pin of the component you want to wire to. In fact, using Multisim is easier if you don’t think too hard about wiring – it’s completely intuitive. Click a pin to start, roll the mouse to your destination pin or wire. Multisim shows you the path for the wire by providing a ghost of it. If you’re satisfied with this path, click on the destination pin; if not, click on the circuit window along the path (in as many places as you wish) you want the wire to take. Right-click to cancel the wire. Use the figure above as a guide to wire all the components together.

Unlike many schematic capture tools, Multisim is intelligent enough to know what you want to do based upon where you click. Without noticing, you have just observed “modeless” operation – you don’t need to tell Multisim if you want to be in wiring mode or part placement mode as is necessary with other capture tools. Component placement starts when you choose a component from the browser, wiring begins when you click on a pin, component moving starts when you click on a component body, etc.

If you need to reshape an already-routed wire, just select it by clicking on it. You can drag its segments or corners to any location you wish without the wire disconnecting from the component. Such “rubberbanding” is an important schematic feature. If you want to add drag points, click on the wire while holding the Control key and additional points are automatically created.

Once you create a connection, Multisim will not disconnect it unless you delete the wire or component. Moving or rotating a component maintains the proper connectivity by re-wiring the connections to the moved pins. You can even auto-connect a component by just dropping it onto an existing wire – Multisim opens and and reconnects automatically!
Simulation

Though there are other ways to simulate, you can start simulation right away by clicking on the Toggle Switch. Starting simulation for the simple circuit lights the Lamp when the switch is closed and turns it off when the switch is opened. Notice the switch has a key assigned to it? Hitting that key (for the switch, the key by default is "SPACE") closes the switch. During simulation, hit the "spacebar" as if you were typing a space. The Lamp turns On or OFF (notice the lamp illuminates and goes out, indicating it is being turned on and off) as the switch is closed and opened. Just like you'd expect from a Lamp. This is one of the many animated parts included in Multisim. Having the proper connectivity (behind the scenes Multisim updates the netlist in real time) means it's trivial to simulate without having to debug a SPICE netlist. Turn off simulation now using the toggle switch.

Virtual Instruments

One of the most common ways of examining simulation results is with Multisim's virtual instruments, which look and behave just like their real-world counterparts. Clicking the third button of the Design Bar opens the Instruments toolbar. Like the other toolbars, rolling your mouse over each button shows a tool tip label after a brief delay. Multisim's eleven instruments are in alphabetical order, with the oscilloscope as the fourth-last. The scope can be placed on the circuit window by clicking it to select, then dragging the ghost of the instrument that automatically appears to the desired location, and clicking again. Attaching the oscilloscope "A" input to the wire between the Lamp and the switch, double-clicking on the instrument (to open the instrument's face) and restarting simulation, will show you the voltage waveform. Of course, if you change the switch position during simulation, you will see the corresponding change in the waveform, another feature unique to Multisim.

It's all part of that ease of use idea. You get your first waveform without worrying about any syntax and without worrying about generating a netlist. The instruments actually use the results of various complex SPICE commands (the oscilloscope uses SPICE's Transient Analysis), employed behind the scenes, transparent to you. Of course, for those who want complete flexibility, Multisim makes all such commands available.
Multisim offers many types of simulation, including of course SPICE-based simulation so it gives you the power of SPICE, including the ability to use thousands of simulation models created by component manufacturers. In addition, Multisim makes SPICE easy to use. All you do is choose the level of detail you want and Multisim handles the rest. Multisim also offers thirteen different "modelmakers" to translate data book parameters into SPICE models. Using these utilities is described in the User Guide.

You might notice Multisim has several other simulation features not available in most simulators. Although Multisim has full digital SPICE simulation, it also has VHDL and Verilog HDL, which are explained in detail in the documentation. VHDL and Verilog HDL simulation allow you to simulate digital designs that are larger (often too large for modeling in SPICE at the transistor or gate level) and more portable. In addition to supporting the use of SPICE, VHDL, Verilog alone (and others such as C-code and RF Simulation), Multisim also offers co-simulation of all these together. This capability (unique to Multisim) is particularly useful when interfacing analog or small/medium scale digital chips (which would typically be modeled in SPICE) with large-scale, complex digital components, such as microprocessors or FPGAs (which might be modeled in VHDL or Verilog). For the very first time, you can now simulate everything on your board all together. Co-simulation is completely intuitive with Multisim and is addressed in the User Guide.

Analyses

As well as its virtual instruments, Multisim also includes a number of analyses for examining your circuit’s behavior in ways just not possible in the real world.

To see some of the advantages of the analyses, load the amplifier circuit 25db_amp.msm from the sample folder on your Demo (CD or Web version).
The fifth button of the Design Bar has a menu showing the eighteen available analyses: more than any other product of its kind. You can see the effects of changing temperature on this circuit by choosing Temperature Sweep. In the Temperature Sweep dialog box, you can enter a comma-separated list of temperatures in the Values under Points to Sweep. Also, on the Output Variables dialog tab, you can tell Multisim which of the circuit's voltages and currents you want plotted and click the dialog's Simulate button. The steps needed are explained in more detail in the Multisim User Guide.

The Design Bar's sixth button starts the Postprocessor allowing you to edit and manipulate the simulation results. For example, you could compute power by multiplying the voltage by the current waveform. Using the post Processor is beyond the scope of this tutorial, but is explained fully in the Multisim User Guide.

**Macros and Hierarchy Blocks**

Multisim provides hierarchy to help you organize your design. You can create a macro by selecting a portion of your circuit, cutting or copying and pasting them as a macro. Double clicking on the resulting macro box allows you to probe into the macro. A macro is stored in the circuit file that uses it but, for flexibility, Multisim also provides hierarchy blocks, which can be stored in a remote file. So if another team member changes a block your design uses, you get the changes immediately. Using hierarchy is again beyond this introductory tutorial, but is of course covered in the Multisim User Guide.

**Buying into the Benefits**

Some of Multisim's capabilities have caught your attention? Why not call our Sales team at 1-800-263-5552 for detailed information on which edition of Multisim is right for you? Or visit our Website, at http://www.electronicsworkbench.com.