

MULTISIM DEMO 2.1: INTRODUCTION TO MULTISIM/THE THREE-WAY SWITCH

Let's pretend you are an electrical engineer interviewing at Bell Laboratories back in the 1950's. The manager interviewing you is very impressed with you thus far, but then you are given one last question...explain how a three-way light switch works.

We most likely have all encountered three-way light switches at some point. Three-way switches are the wiring setup which allow us turn on a light at the bottom of the stairs, and then turn the light off at the top of the stairs. Sometimes there are even extra switches, possibly a third switch at the middle of the stairs. We'll focus on systems that just have two switches, however.

To introduce Multisim, we'll analyze a straightforward three-way light switch setup. We'll do it in a direct current (DC) circuit for simplicity, but the same general concepts apply to the three-way light switches we find in domestic lighting which runs off on alternating current (AC).

Let's start by observing the schematic of the three-way light switch shown in Figure 2.1.1 below.

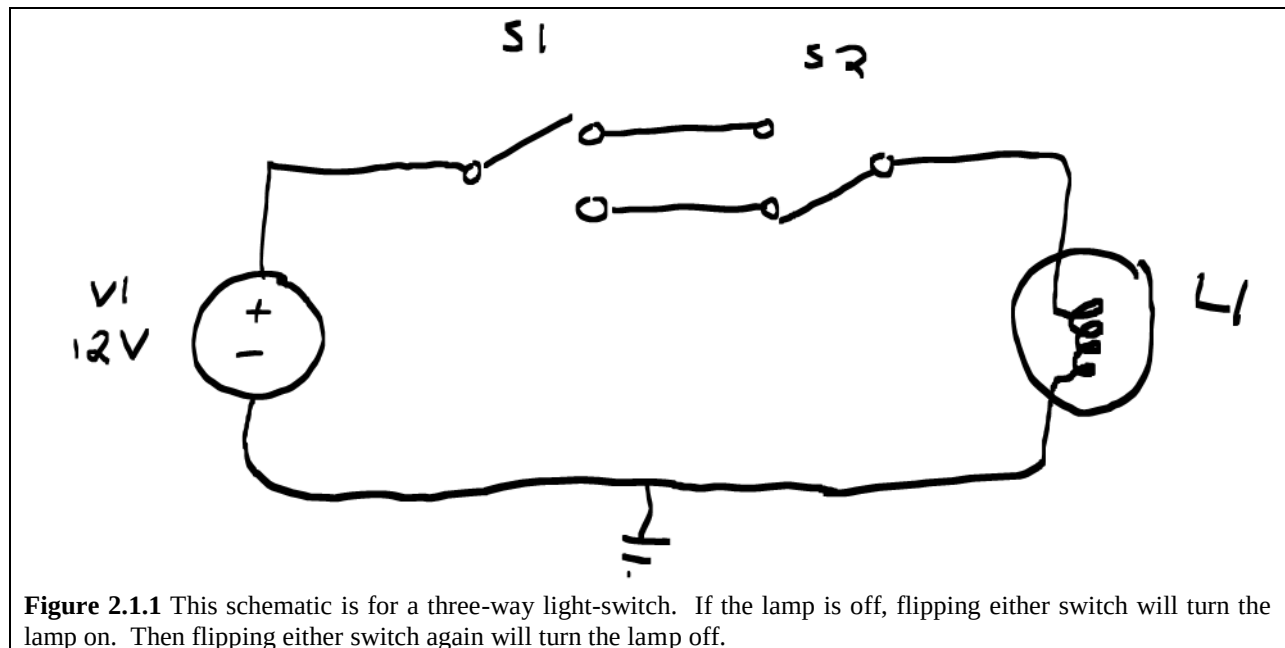


Figure 2.1.1 This schematic is for a three-way light-switch. If the lamp is off, flipping either switch will turn the lamp on. Then flipping either switch again will turn the lamp off.

Now we want to take the above schematic and construct it in Multisim. Go ahead and start up Multisim if you haven't already. Once it has loaded, you should find yourself looking at a screen similar to that shown in Figure 2.1.2 below.

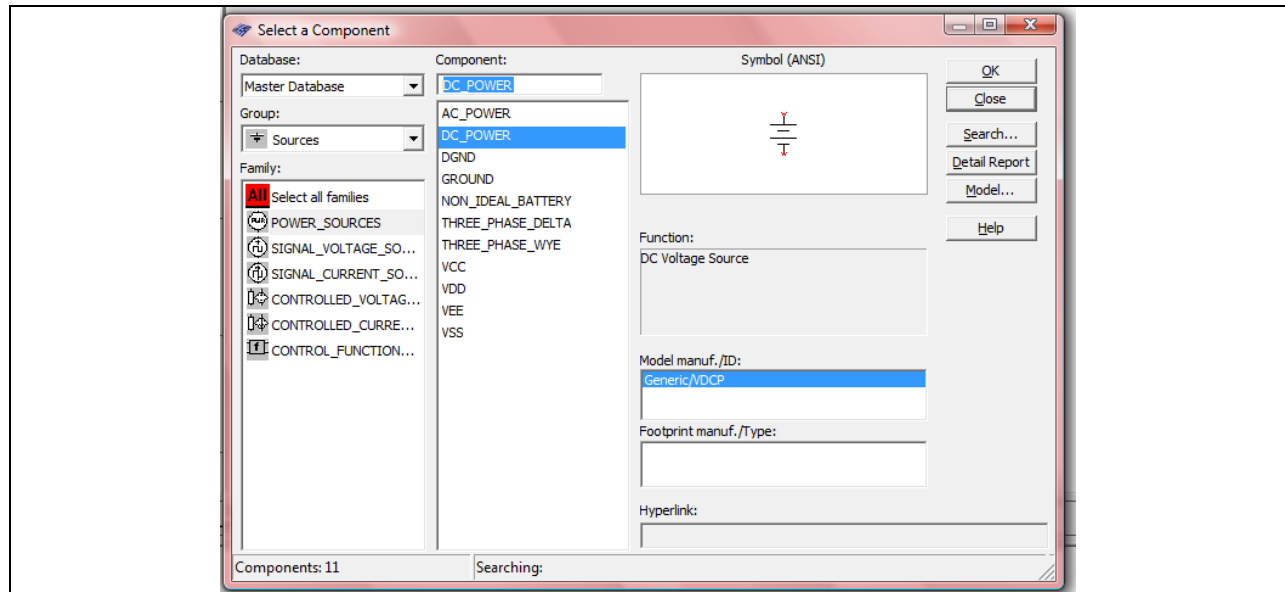


Figure 2.1.4 The Select a Component Window. This window will become extremely familiar to you as you use Multisim for all of your circuit simulating needs.

The way in which we just obtained the component can be thought of as “the long way.” There are two other ways to place a component. The first is through utilizing the shortcut key combination **Control – w**. This will bring up the Select a Component Window, and from there, everything is the same as what we’ve already gone over. Multisim has a plethora of shortcut key combinations, a brief list of which is located in this tutorial’s Appendix A.

Yet another way to get to the Select a Component Window (specifically in the Sources Group) is by pressing the “Place Source” button, which looks like , and should be found up towards the top in one of the toolbars. Pressing the  button will bring up the Select a Component Window with the Sources Group already selected. This means all you have to do is select the necessary Family and the Component.

No matter which way you do it, after placing the component, the Select a Component Window should pop back up by default. Press Close, and look at your Schematic Capture Window. It should resemble that shown in Fig. 2.1.5. Simple, yet satisfying...a 12 Volt DC source.

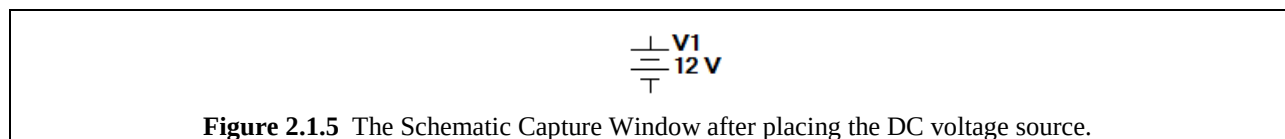


Figure 2.1.5 The Schematic Capture Window after placing the DC voltage source.

One component does not make a circuit (generally speaking), so we must now add the other into the schematic. Press **Control – w** to access the Select a Component Window and under Group select Indicators and under Family select LAMP. Place the 12V_10W component into your schematic. Next, in the Select a Component Window, under Group select Basic, and under Family select SWITCH. Scroll to the bottom and find the SPDT switch; place two of this component into the schematic.

Note: SPDT stands for Single-Pole-Double-Throw. Other common switch types are SPST (Single-Pole-Single-Throw) and DPDT (Double-Pole-Double-Throw).

The schematic should look like that shown in Fig. 2.1.6.

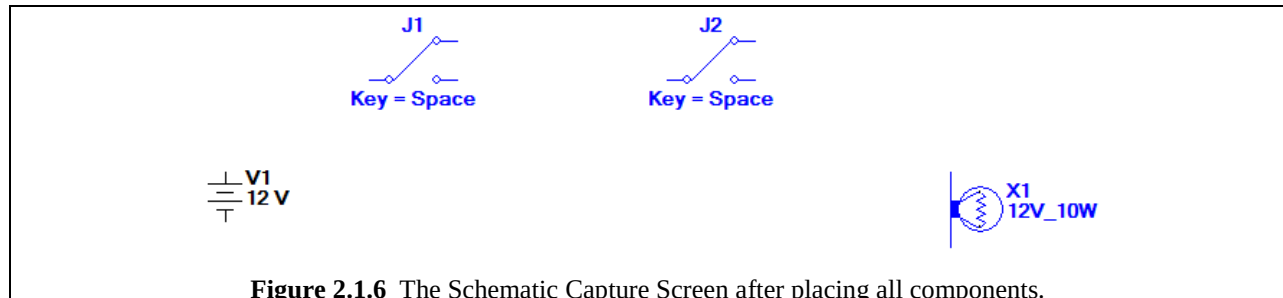


Figure 2.1.6 The Schematic Capture Screen after placing all components.

The next step is to wire the individual components together, but before we do that, we have to rotate switch J1 so that it will match more easily with the schematic in Fig. 2.1.1. To do this, click on switch J1. Be careful not to click on the letter or the “Key = Space” part. Additionally, clicking on the toggle part of the switch toggles the switch rather than selects it, so click on one of the white circles or another non-activating part. When the entire switch is selected, you will know it by there being a dashed box around it like that shown in Fig. 2.1.7.

To rotate the switch press **Ctrl – r**. Each rotation is set to 90° clockwise, so we need to rotate twice to adjust the switch to the right position.

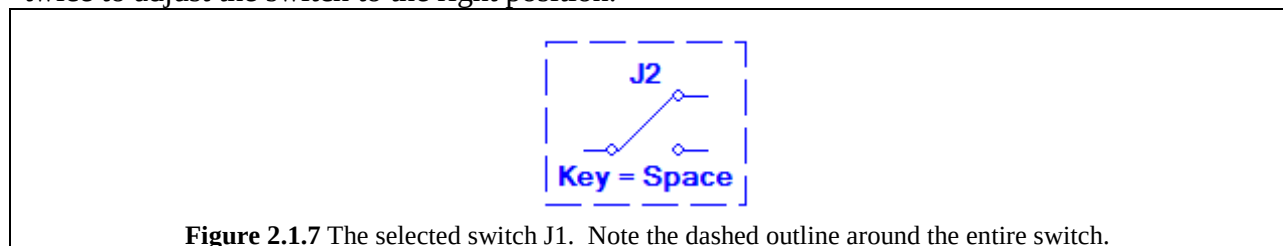
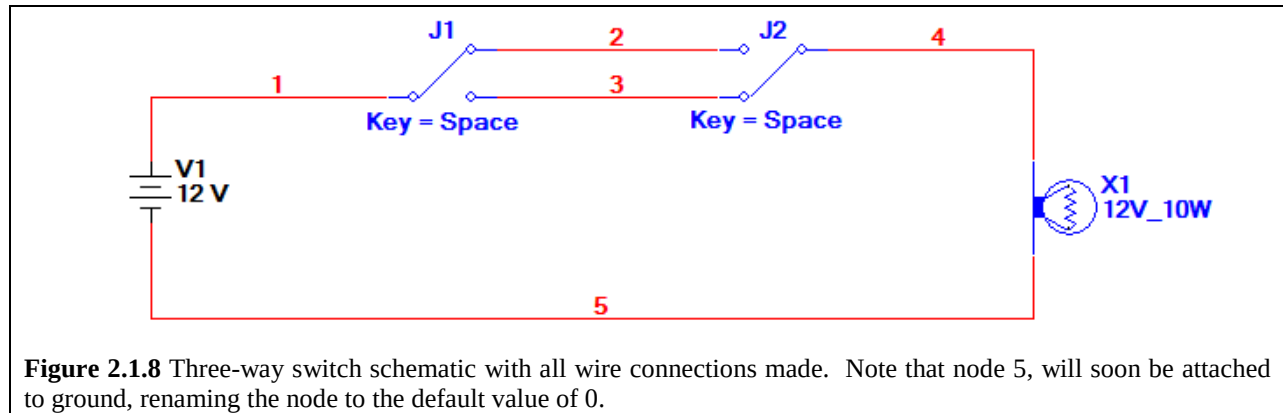


Figure 2.1.7 The selected switch J1. Note the dashed outline around the entire switch.

The next step is to add wires between the different pins of each component. To do this, simply place the cursor over one of the pins of a given component; the cursor will change from the traditional arrow to something which looks like a crosshair with a big black dot in the middle. Click once, and move your mouse to the pin on another component. When the cursor is over that pin, the crosshair-black-dot cursor will appear telling you that you are on a pin. Click once more. This attaches the wire to the second component. Following the schematic in Fig. 2.1.1, make all the necessary connections to build the circuit. When finished, your schematic should look like that shown in Fig. 2.1.8 found on the next page.



All circuits in Multisim (and pretty much all SPICE software) must have a ground somewhere in the circuit. If you ever find yourself getting a simulation error, first thing you should check is that you have a ground in your circuit and/or that your ground is properly hooked up.

In Multisim, the node number attached to ground is automatically set to “0”. So even if you have a node numbered “7”, as soon as it is attached to ground it will be renamed “0”.

To place a ground, click on the  button on the toolbar or go to Place>Component, which will bring up the Select a Component window. Select Sources under the Group pull-menu, then the POWER_SOURCES family, and finally the GROUND component. Click OK and place the ground component anywhere. Attach the ground to the bottom node of the circuit.

You are not limited to only one ground symbol in any given schematic. For example, the following circuits shown in Fig. 2.1.9 are equivalent.

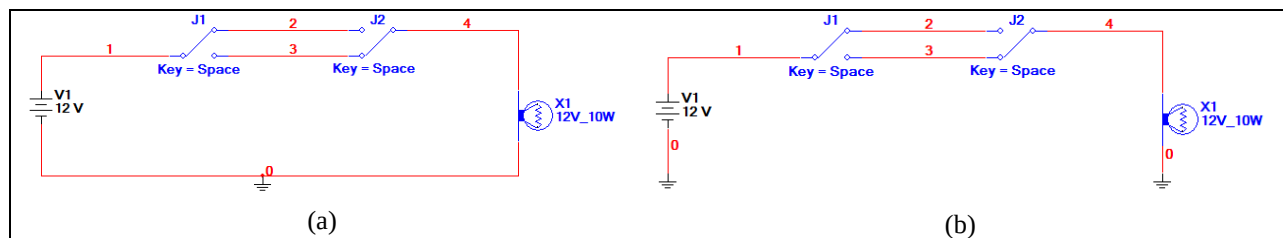


Figure 2.1.9 The circuit in (a) has only one GROUND symbol, however it is electrically identical to the circuit drawn in (b) using two separate ground symbols. Multisim and SPICE software in general often allow for many different ways to draw or model the same circuit. The circuit in (b) can often make the drawn schematic cleaner looking because there are fewer wires, however, the circuit in (a) can sometimes drive the point of a common ground home more to the user.

Now in the three-way switch schematic, each switch, in addition to having its name visible (J1 or J2), also has the phrase “Key = Space” visible. What this means is that you can flip the switch by pressing the space bar. Try it (make sure your active window is the Schematic Capture Screen). Now notice that since both switches J1 and J2 have their Key assigned to Space, pressing the space bar flips both switches simultaneously.

We can flip each switch individually by going and clicking on the toggle part of the switch. To do this, first place your cursor over the toggle section of the switch (the part that moves). The

cursor should change from the traditional arrow to a finger-pointing-hand, and the toggle section of the switch should become bold. Click the left mouse button, and the switch should flip.

By default, all SPDT switches are assigned the space bar as their key, but since we are simulating a three-way light switch, we need each switch to be controlled by different keys. To do this, double click on one of the switches. This should bring up the Properties Window for the switch, which is shown in Fig. 2.1.10. Make sure the Value tab is brought up, and click on the pull-down menu for the Key for Switch field. Set the one switch to “B” (for bottom of stairs), and click OK. Then set the other switch to “T” (for top of stairs).



Figure 2.1.10 The Properties window for one of the SPDT switches in the schematic. By default the Value tab should be up front when the window opens. Clicking on the different tabs will allow access to different properties and characteristics of the switch.

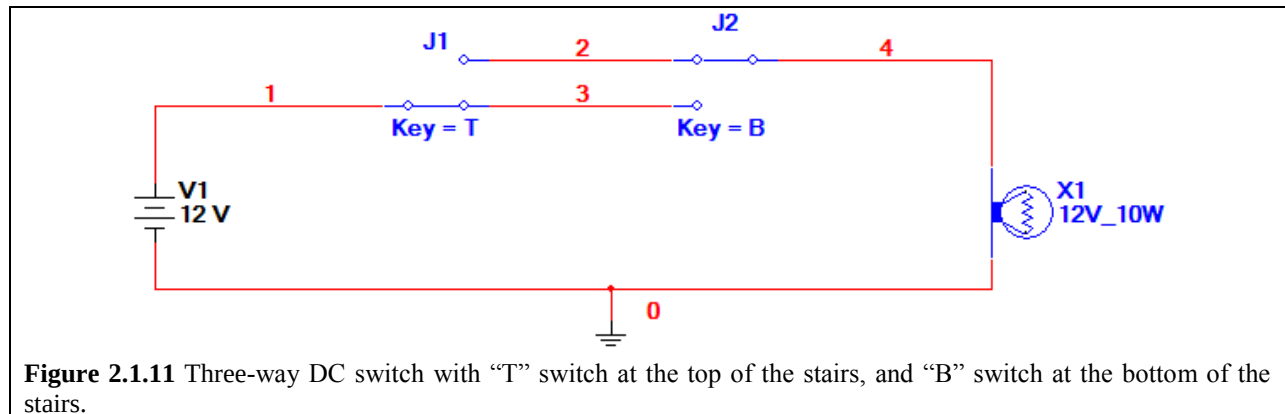


Figure 2.1.11 Three-way DC switch with “T” switch at the top of the stairs, and “B” switch at the bottom of the stairs.

Now the circuit is ready to simulate. There are numerous ways to simulate circuits in Multisim, and we’ll get into a lot of the different kinds later in this manual, but for right now we’ll just do the simple Interactive Simulation. This type of simulation allows us to flip the switches and see the result of that action while the circuit is simulated...hence the “Interactive” in its name.

To run this simulation, there are several approaches: One way is to go to the command bar at the top of the screen, and click on Simulate>Run. Another way is to go to the toolbar on the upper half of the screen and click on the Run/Resume Simulation button, . Additionally, the default keyboard shortcut for the Run/Resume Simulation command is F5.

If your circuit starts out with the light bulb X1 off, as the circuit in Fig. 2.1.11 on the previous page does, pressing either the key “T” or the key “B” will flip the appropriate switch and cause the light bulb to turn “on” indicated by yellow coloring, as shown in Fig. 2.1.12 below. Pressing either of the keys again will turn the bulb off. This is the beauty of the three-way switch. No matter what “state” the circuit is in, flipping either switch will change the state.

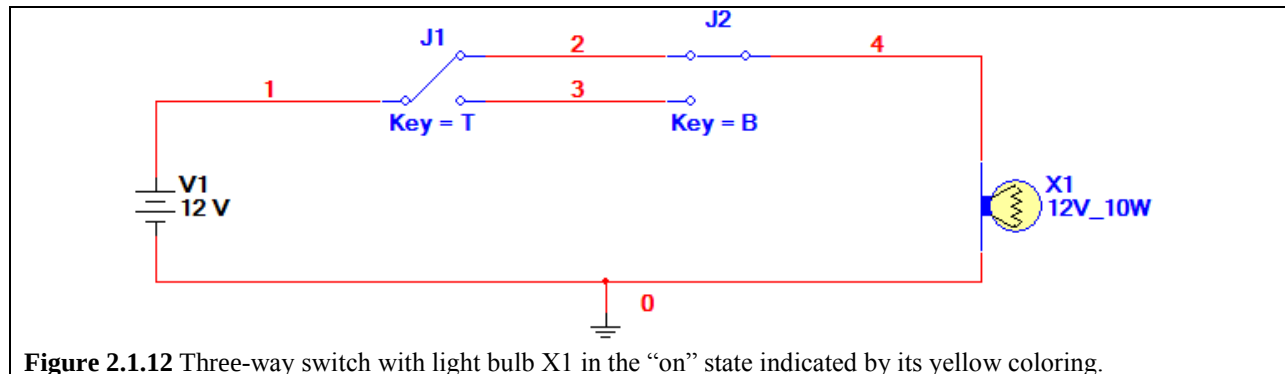


Figure 2.1.12 Three-way switch with light bulb X1 in the “on” state indicated by its yellow coloring.

So let’s pretend you just got home from work, and the light is off. You come in from the garage and are at the bottom of the stairs, you flip the bottom light switch, “B”, and the light comes on. You walk up the stairs, and once you reach the top, you flip the top light switch, “T”. Now when your roommate comes home 30 minutes later, s/he can do exactly what you did. There’s no need to reset anything. It is a beautifully simple circuit.