

## MULTISM TUTORIAL

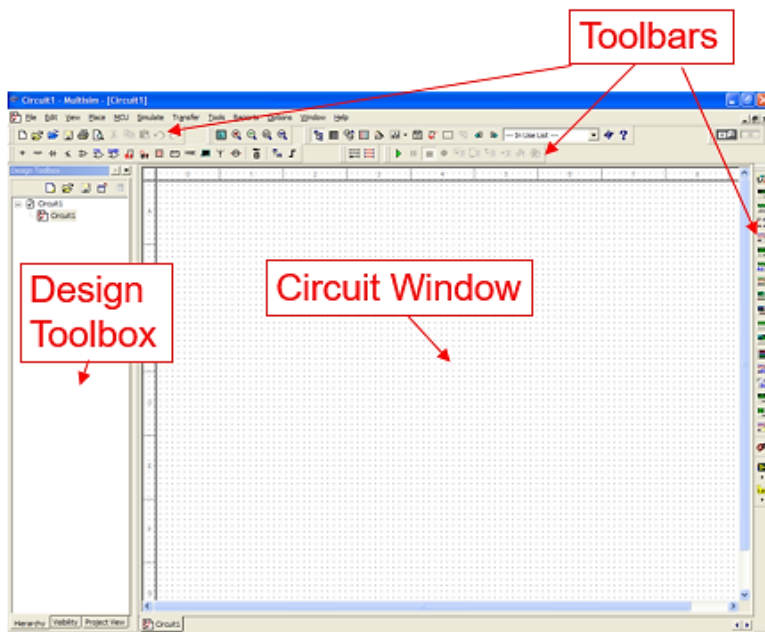
Multisim is an industry-standard, best-in-class SPICE simulation environment. It is the cornerstone of the NI circuits teaching solution to build expertise through practical application in designing, prototyping, and testing electrical circuits. The Multisim design approach helps you save prototype iterations and optimize printed circuit board (PCB) designs earlier in the process.

### Application

- Use a unified environment to teach analog, digital, and power electronics.
- Get to results faster with intuitive graphical simulation.
- Standardize on a powerful design tool used across multiple industries.

### HOW TO START

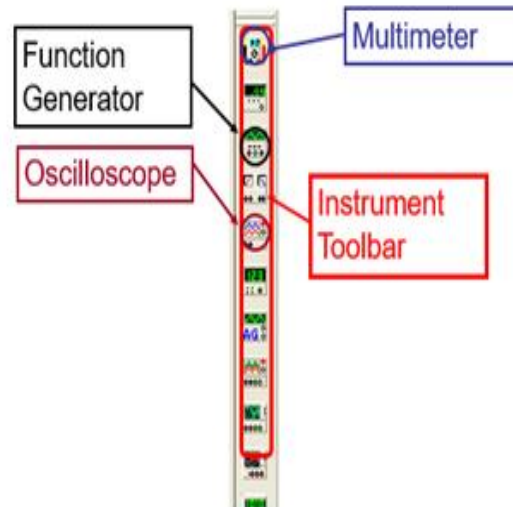
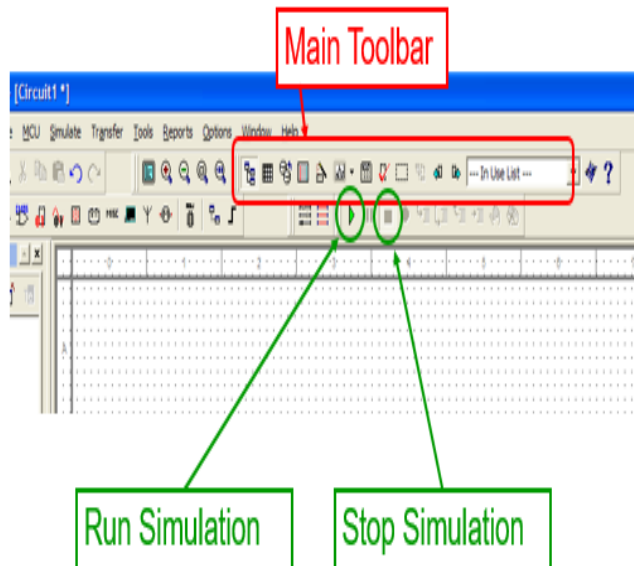
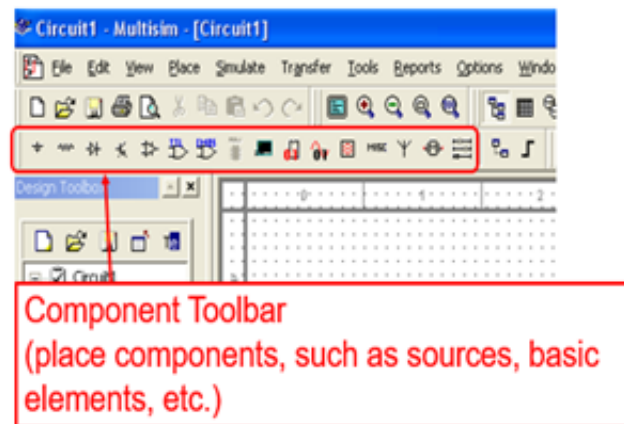
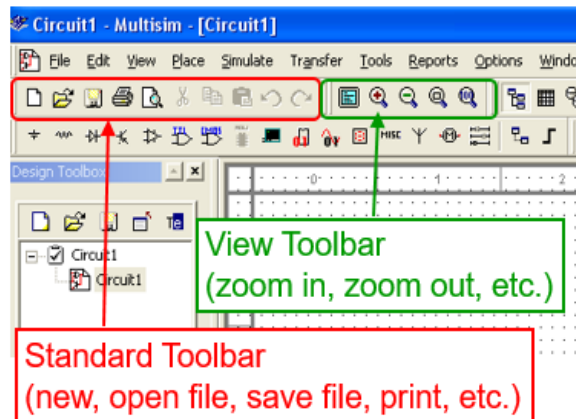
- Install Multism
- Click on start
- Select All Programs
- Choose National Instruments
- Open Circuit Design Suite 12.0
- Double click Multism



## TOOLBARS

Multism has moveable toolbars located at the top and side of the screen.

The following images identify the most commonly used toolbars.



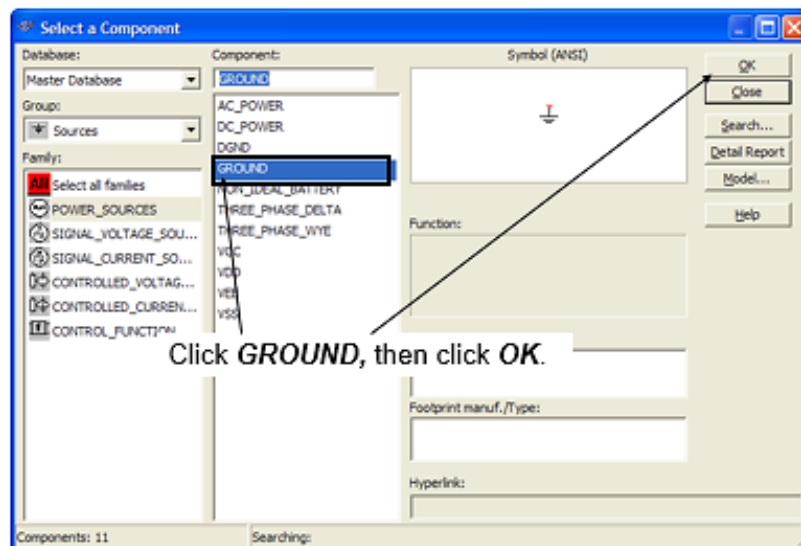
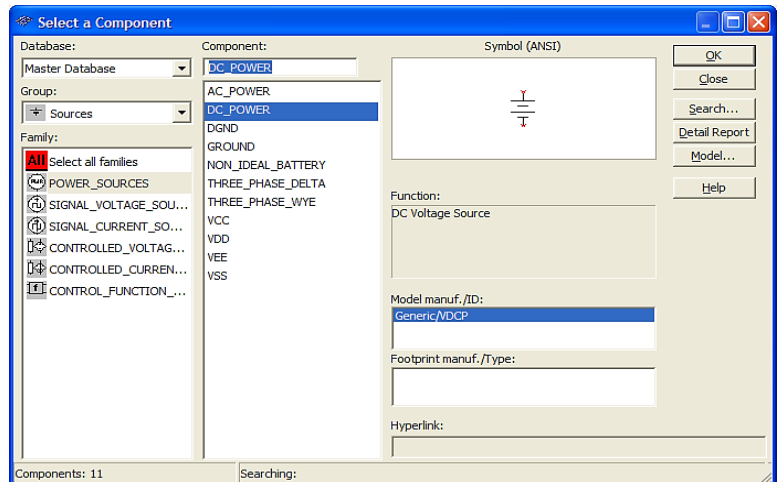
## DRAW A SIMPLE CIRCUIT?

To place a component on the window, you can:

- Click on the component toolbar, then browse for the part as shown below;

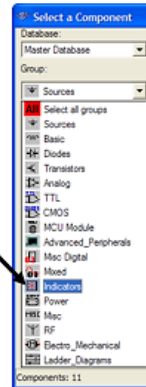


- OR, click on the **Place** on the menu bar ➔ Select **Component** ➔ browse for the part as shown below;

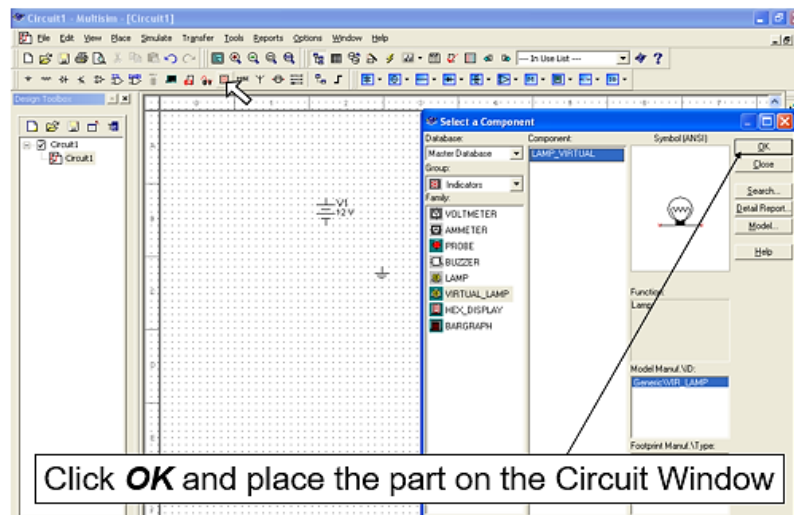
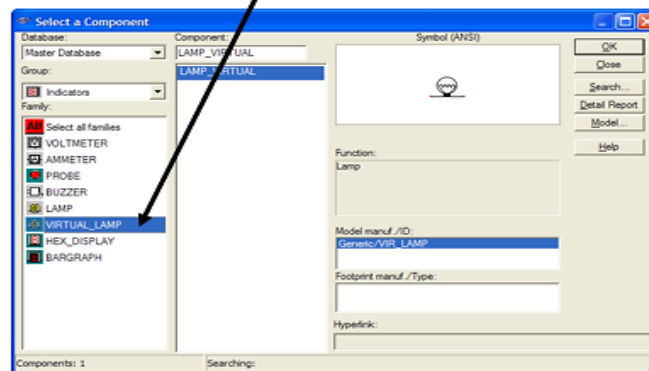


Next, we will place a **Virtual Lamp** from the **Indicators** menu.

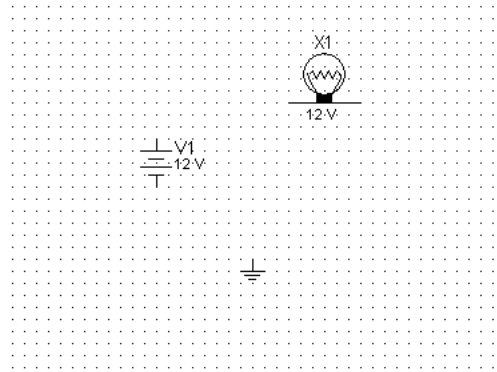
Under **Group**, click the drop-down arrow and click **Indicators**.



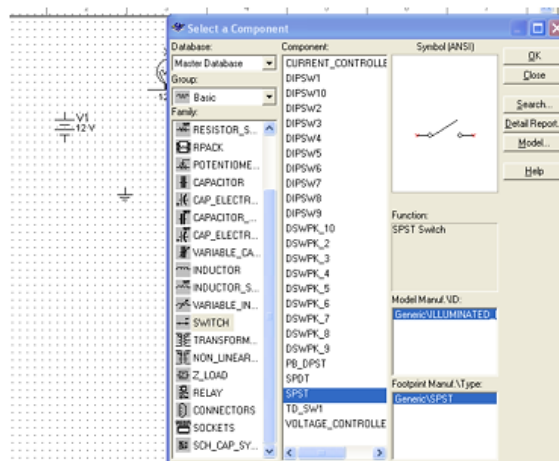
Choose **VIRTUAL\_LAMP** under **Family**.



Click **OK** and place the part on the Circuit Window

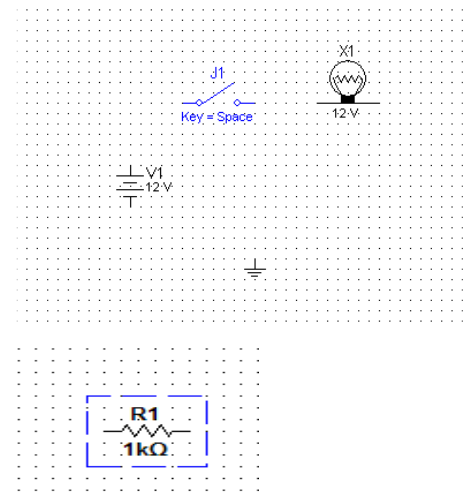
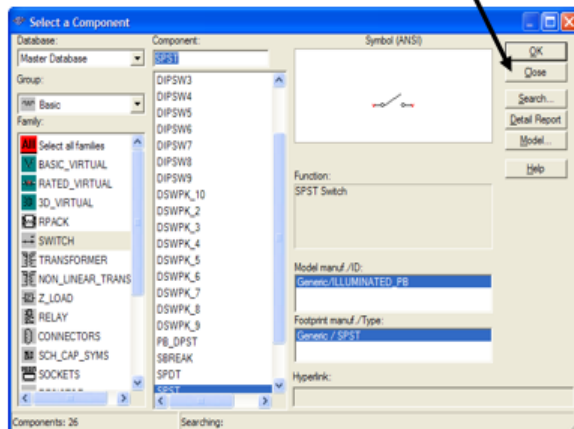


Now we can repeat the part placement to place a switch. See if you can find the switches in the **Basic Group**.



Place a switch. Choose the switch labeled **SPST** for single pole, single throw.

We have placed all of the components that we need for now. Click **Close**.



## WIRING

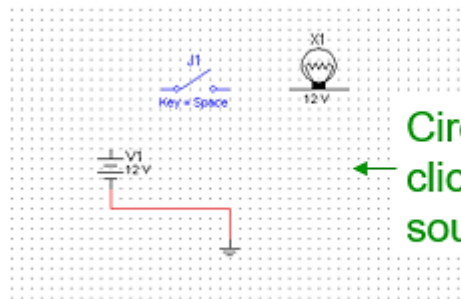
- To begin wiring, hover the cursor over a part terminal (end pin). Notice that the cursor changes shape to indicate that you are starting a wire.



Cursor shape changes, showing that a wire is beginning on the ground symbol.

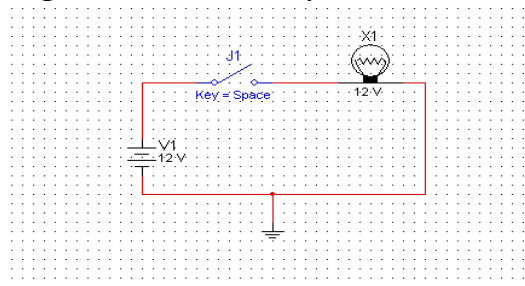


- Click once. Now you can start the wire. To end the wire, click on another terminal.

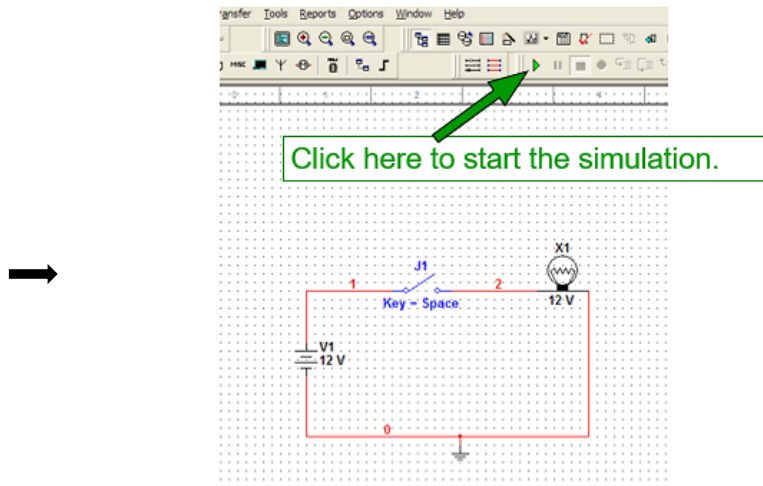


Circuit window after clicking on the source terminal

- Click on the top of the source. Move the cursor to the left switch terminal and click again. This should draw a wire connecting the parts.
- Continue wiring the circuit until you have a complete circuit.

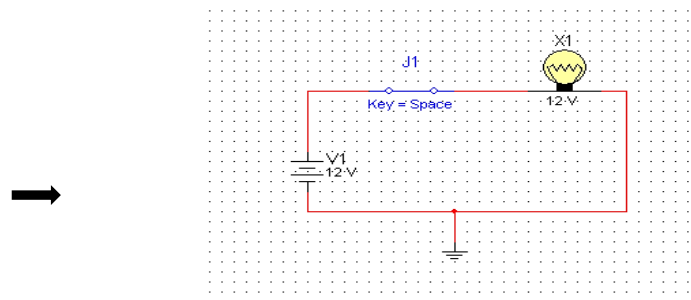


Now the circuit is complete. We will start the simulation by using the **Play** button.

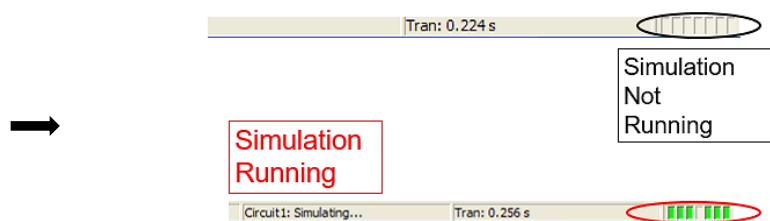


## Running the Simulation

Once your simulation is running, click on the **Circuit Window**, then press the **Space** bar on the keyboard. This will operate the switch.



Press the space bar on the keyboard again. What happens?

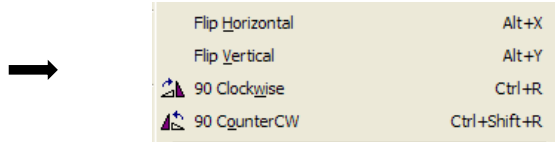


## Wiring Hints

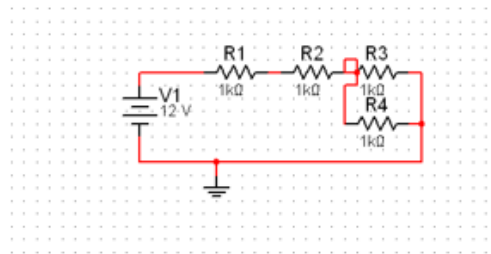
To start a wire with no component terminal nearby:

- Double-click anywhere in the **Circuit window** to start the wiring action.
- Double-click anywhere to end the wiring action.
- From the **menu bar**, choose **Place**, then **Junction**, to place a junction. Start the wire at this point.

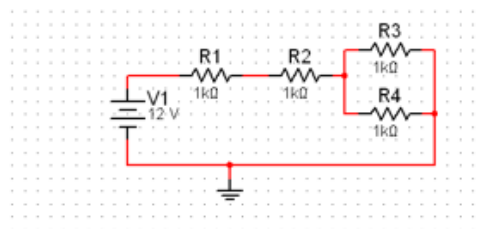
- Single-click while wiring to place a **corner**.
- Right-click while wiring to delete the wire.
- Multisim uses an automatic wire router. This can cause trouble if two terminals are very close to each other.
- **Ctrl R** rotates a component clockwise. You can also right-click and choose a rotation method.



**NB:**



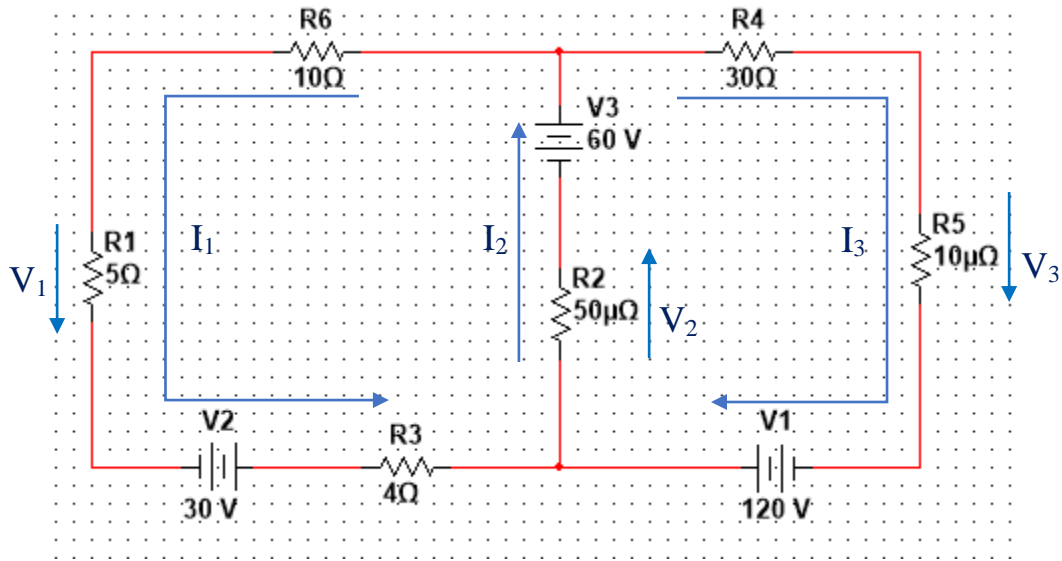
Resistor R2 and R3 terminals are too close.



Circuit layout is easier to follow with components separated.



**WEEK TWO**  
**PRACTISE TEST 1**



- Find the currents  $I_1$ ,  $I_2$  and  $I_3$  respectively.
- Find the voltages  $V_1$ ,  $V_2$  and  $V_3$  respectively.