1. START UP Multisim7:

Click start -> programs -> Multisim7.

The central part of the screen is the drawing plane for our circuit. Elements are on the left bar, such as power source, resistors, and diodes. The right bar contains function generator, oscilloscope, distortion analyzer and other useful equipment.

2. GET COMPONENTS:

a. Go to “Place”, select “Component”, the window “Select a Component” will pop up. In “Database” region, select “Multisim Master”.
b. We may need a certain kind of power source. Select “Sources” in the “Group” region, then the component family will come up in the window below. Click “Power_Source” in “Family” region, and then select “DC_POWER” in component region. Click “OK”, then left click the drawing plane, the power supply will appear in the drawing plane. We can move the power source, and place it anywhere in the plane. Moreover, we may change the parameter for the power source by double clicking its default parameter.

c. We need to choose resistors with proper parameters. Go to “Select a component” window, we can select “Basic” →”Resistor” →”1Kohm_1%”. Then we can put the resistor onto the drawing plane.
In the same sequence, we can GROUND.

d. We need to choose GROUND. Go to “Select a component” window, we can select “Source” □ “GROUND” Then put GROUND onto the drawing plane.
We may rotate the elements to construct our circuit. Just right click the device or equipment, we can find rotation demand in the menu popping out. The online help is also in the menu. It is helpful if we want to know the details of the elements.

3. CONNECTION WIRES:

It is easy to place wires between the devices. Just click at one port and drag the mouse to another port and click again.
4. SIMULATION:

Now we are ready to simulate the result. The circuit should be run before multimeter shows results. We need turn on the switch at the upper-right corner. After turning off the switch, double clicking the analyzer, multimeter, we can get the results.