

EE386 : INTRODUCTION TO PROTEUS

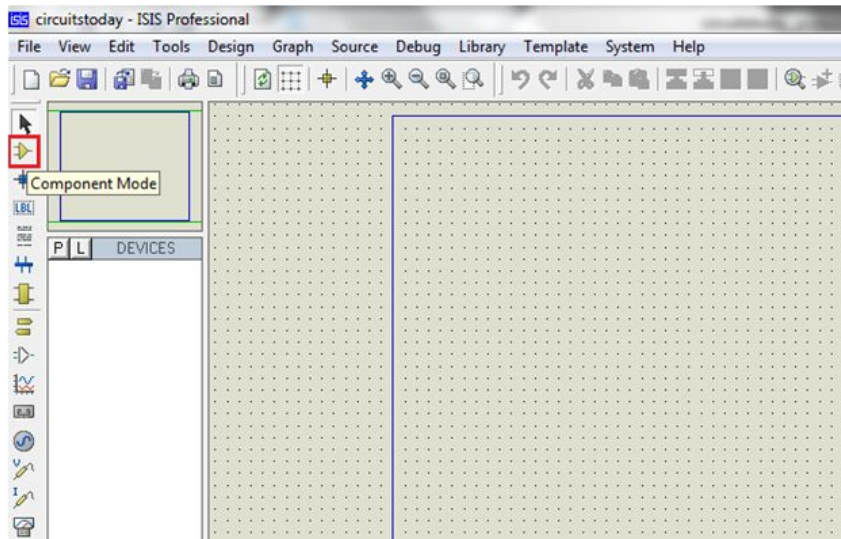
Starting New Design

Step 1: Open ISIS software and select New design in File menu.

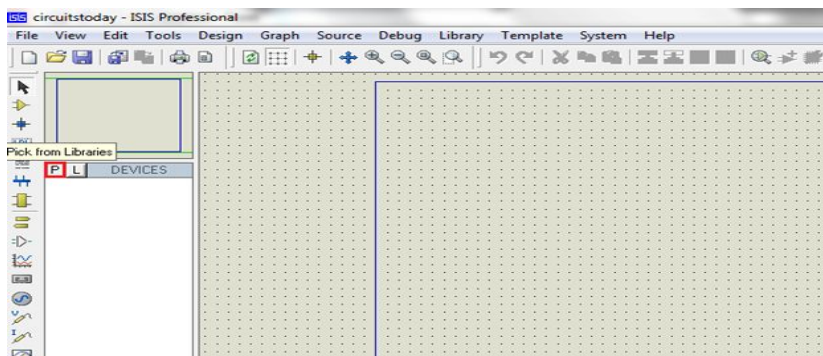
Or

1. Click new project on the start menu.
2. Rename and select the path.
3. Check on Create a Schematic from the selected template. select default.
4. Click do not create a pcb layout.
5. Check on No firmware project.
6. Click Finish

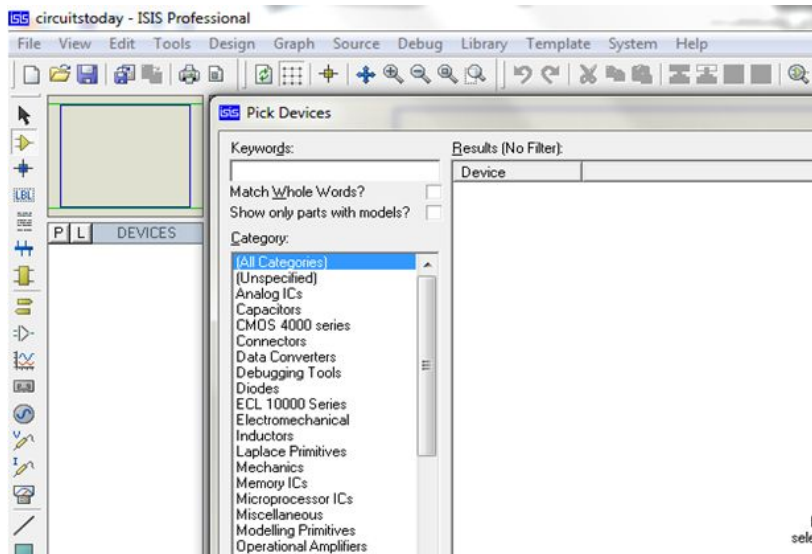
Step 2: To Select components, Click on the component mode button



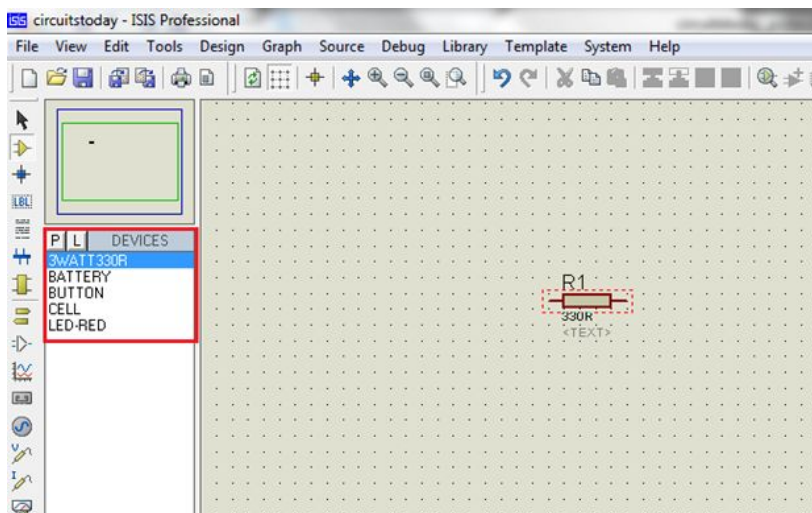
Step 3: Click On Pick from Libraries. It shows the categories of components available and a search option to enter the part name.



Step 4: Select the components from categories or type the part name in Keywords text box.

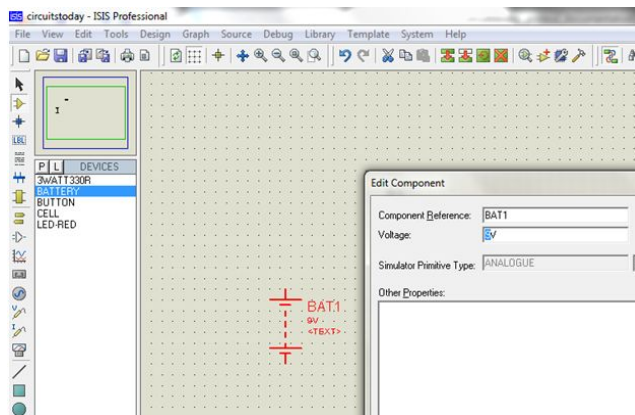


Step 5: The selected components will appear in the devices list. Select the component and place it in the design sheet by left-click.

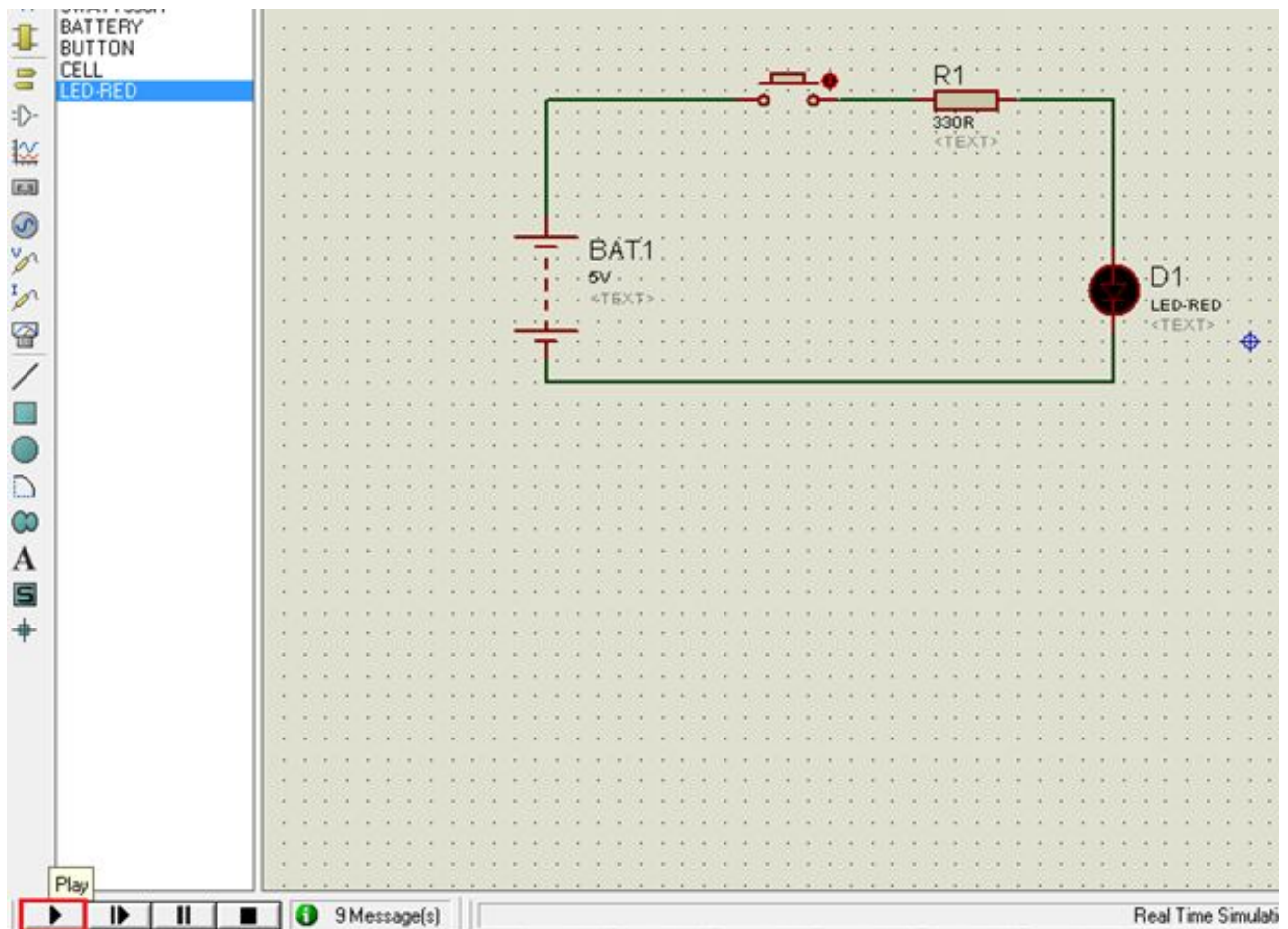


Step 6: Place all the required components and route the wires i.e, make connections. Either selection mode above the component mode or component mode allows to connect through wires. Left click from one terminal to other to make connection. Double right-click on the connected wire or the component to remove connection or the component respectively.

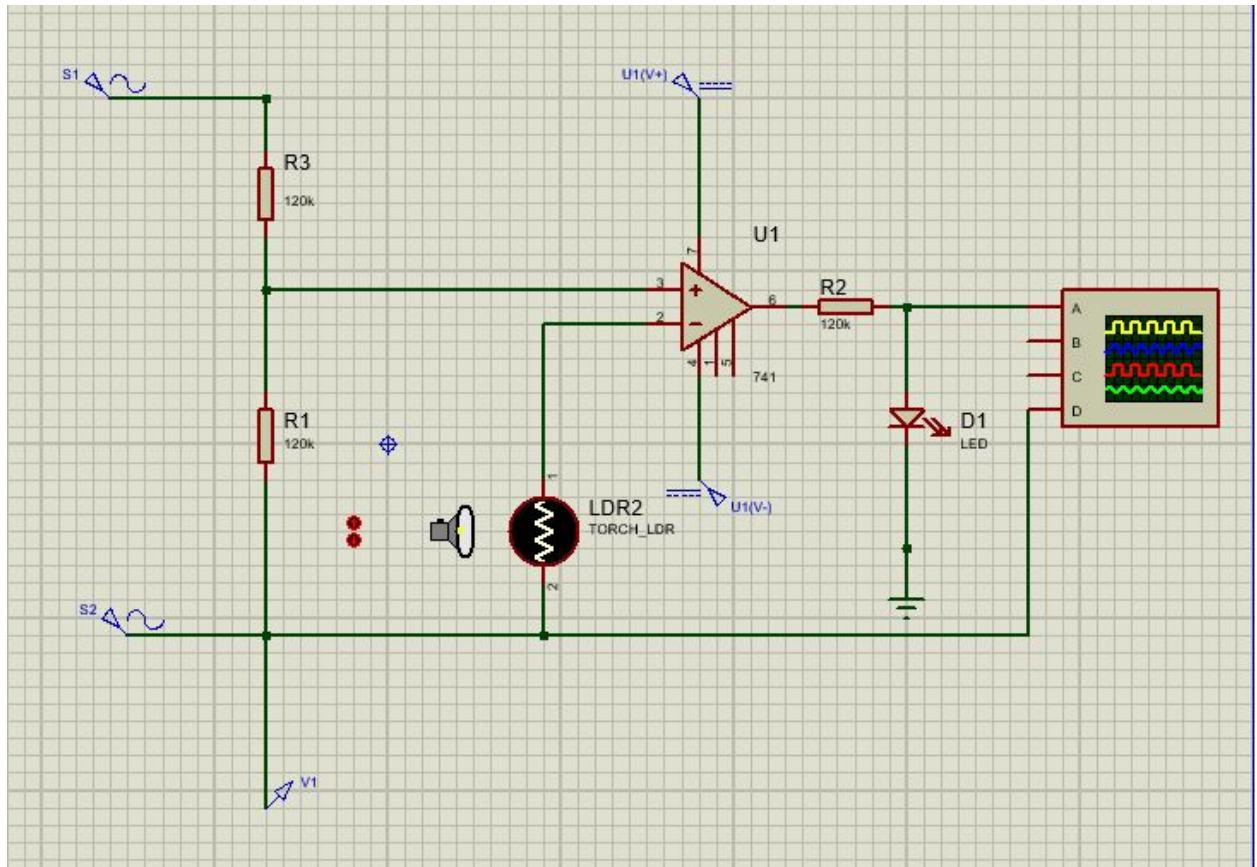
Step 7: Double click on the component to edit the properties of the components and click on Ok.



Step 8: After connecting the circuit, click on the play button to run the simulation.



Task1



S1 - 8V ,50 Hz sinusoidal
S2 - 4V ,50 Hz Sinusoidal
V1 - voltage probe
741 OPAMP