Experiment No. 1: Introduction Proteus

Objectives:

In this lab we will learn the basics of Proteus Software and, we will make and simulate some basic circuits.

Theory:

Proteus:

The Proteus Design Suite is a proprietary software tool suite used primarily for electronic design automation. The software is used mainly by electronic design engineers and technicians to create schematics and electronic prints for manufacturing printed circuit boards.

Main Window:

Main window when opened.

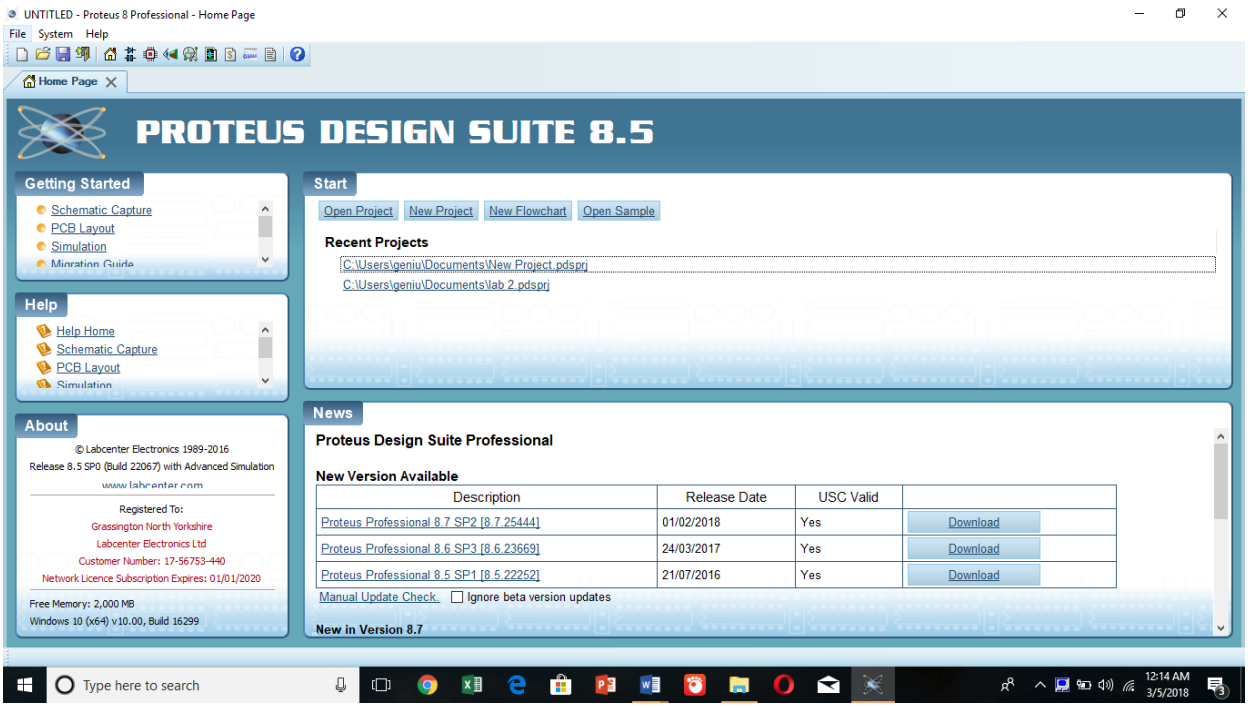


Figure 1 Main Window

To Open a New Project:

To open a new project, select circled icon in the given image…

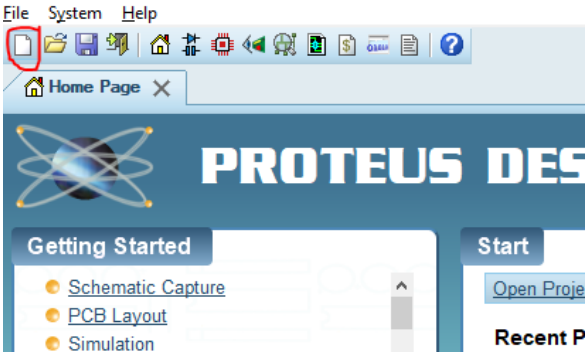


Figure 2 New Project

After that we will get this window…

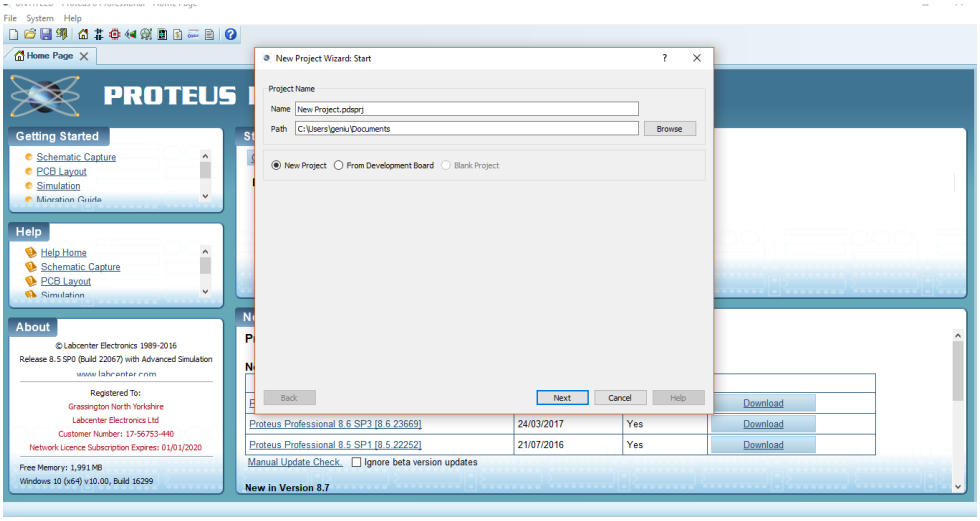


Figure 3 Option

After that select Potrait A4 from the options…

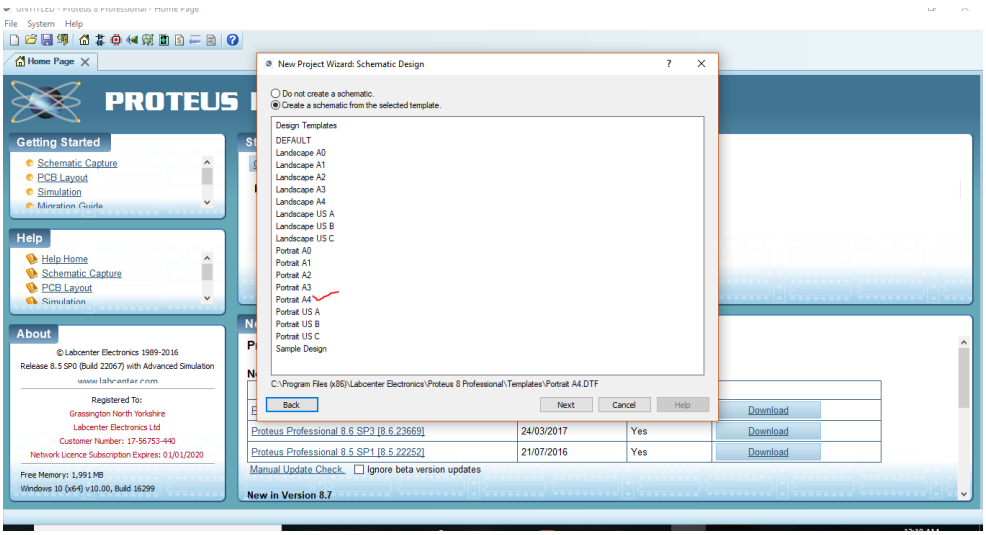


Figure 4 Options

After that select Single Eurocad (2 layer)…

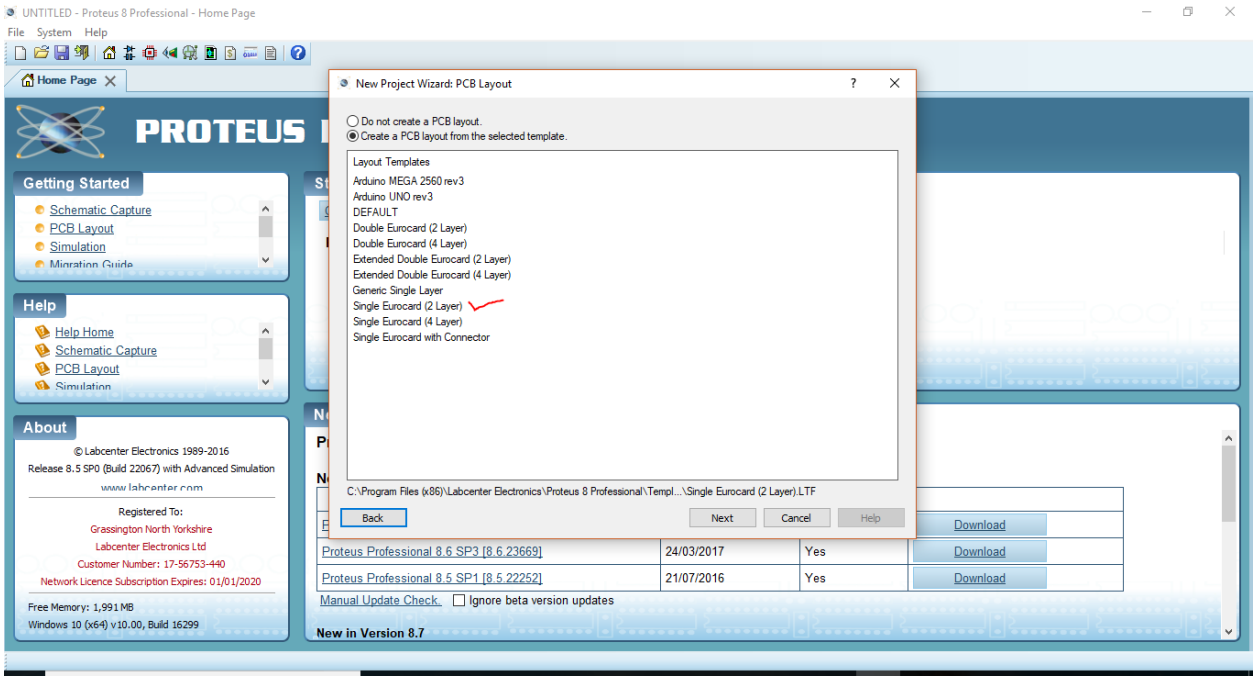


Figure 5 Options

So.. after finishing this we will get this screen …

If we want to Design a PCB layout we have to select this option (in the red)…

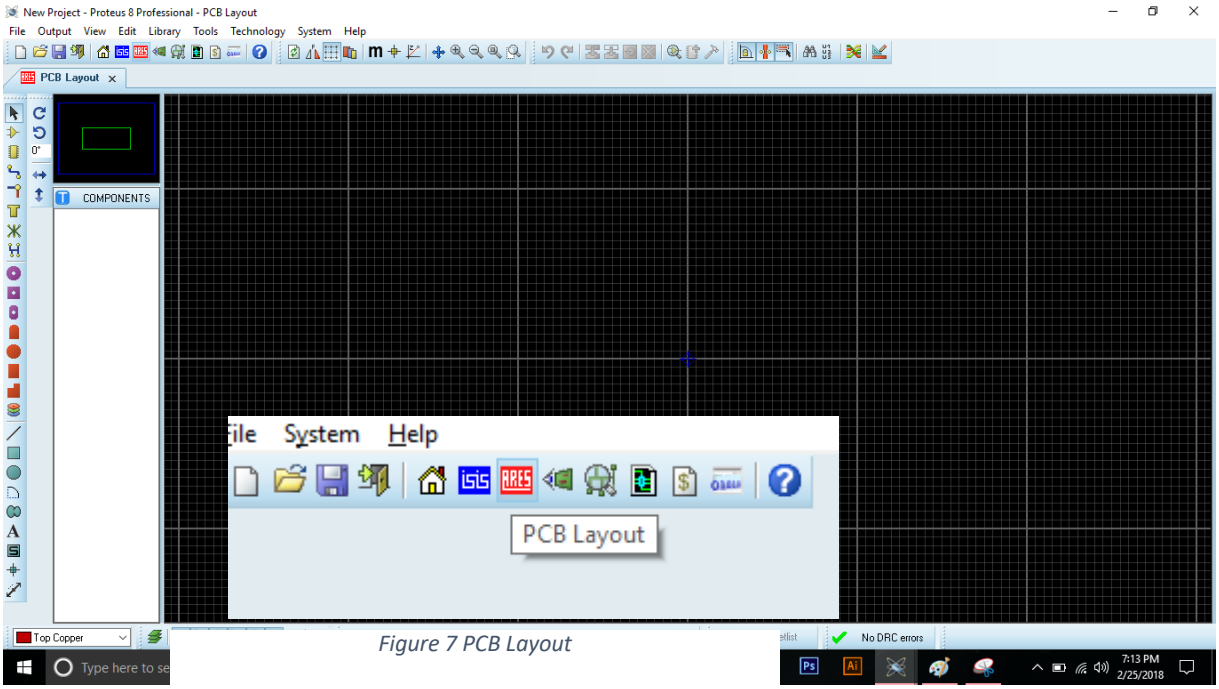


Figure 7 PCB Layout

Figure 6 Main Interface

If we want to Design Schematic we have to select this option (in the blue)…

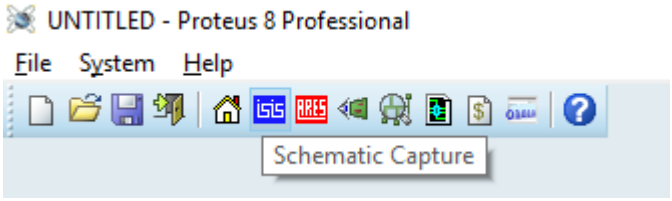


Figure 8 Schematic

Component Mode:

For designing a circuit we need components which can be selected from here below the arrow..

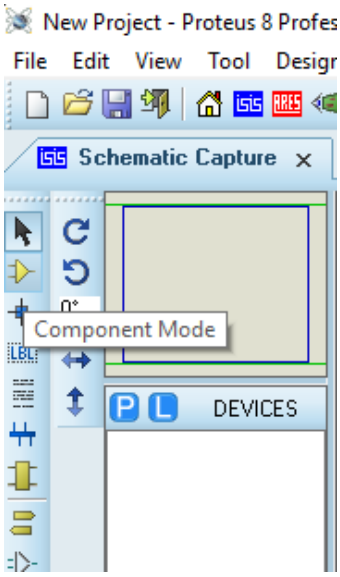


Figure 9 Component Mode

We will get this window to select or search for a specific component type the keyword of that component. We can select it form the category list or can search it directly by typing the keyword of element we want to search. E.g we have

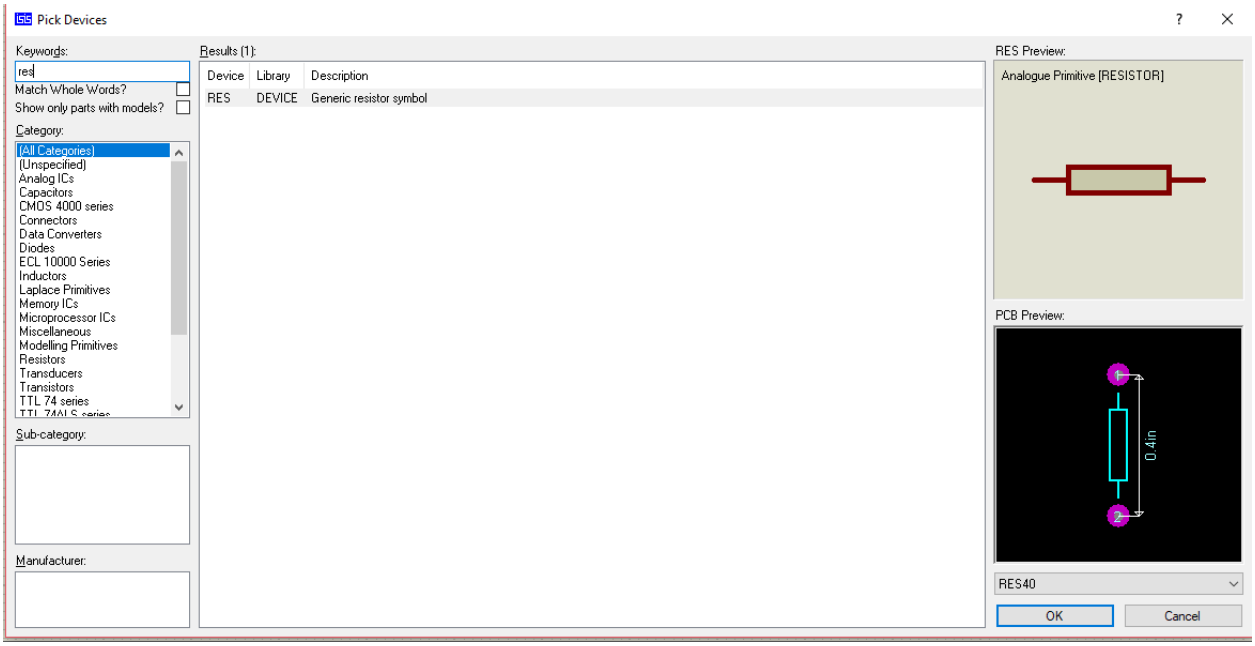
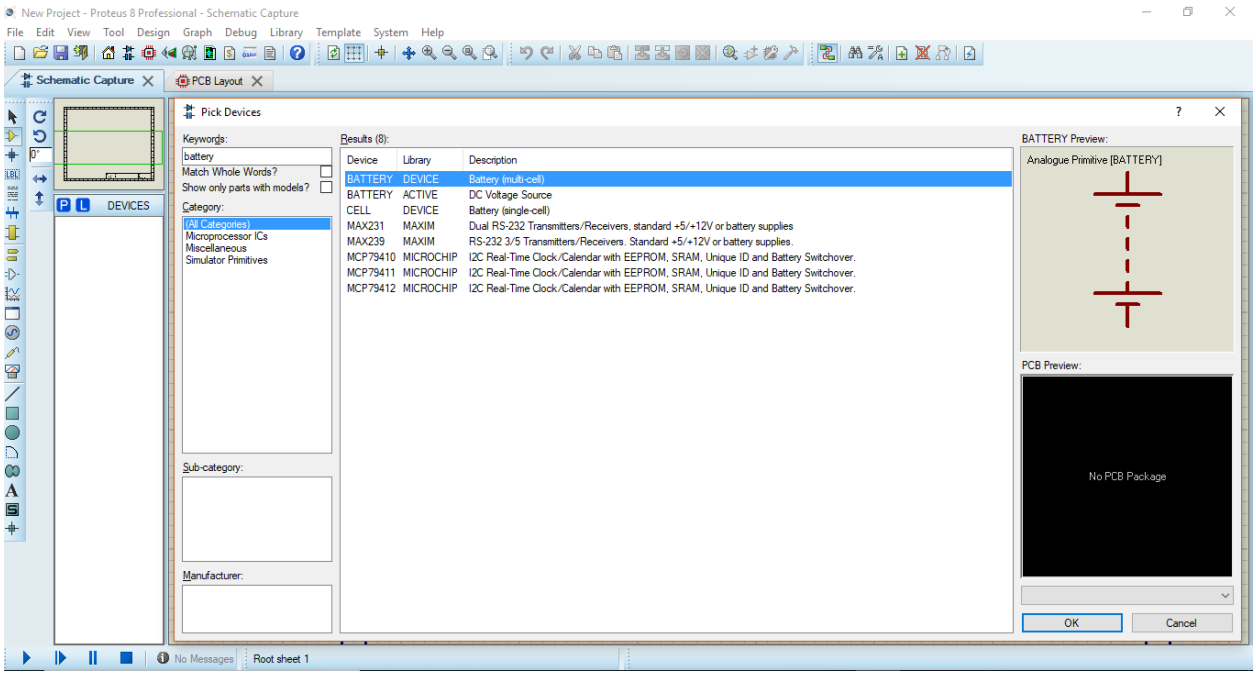


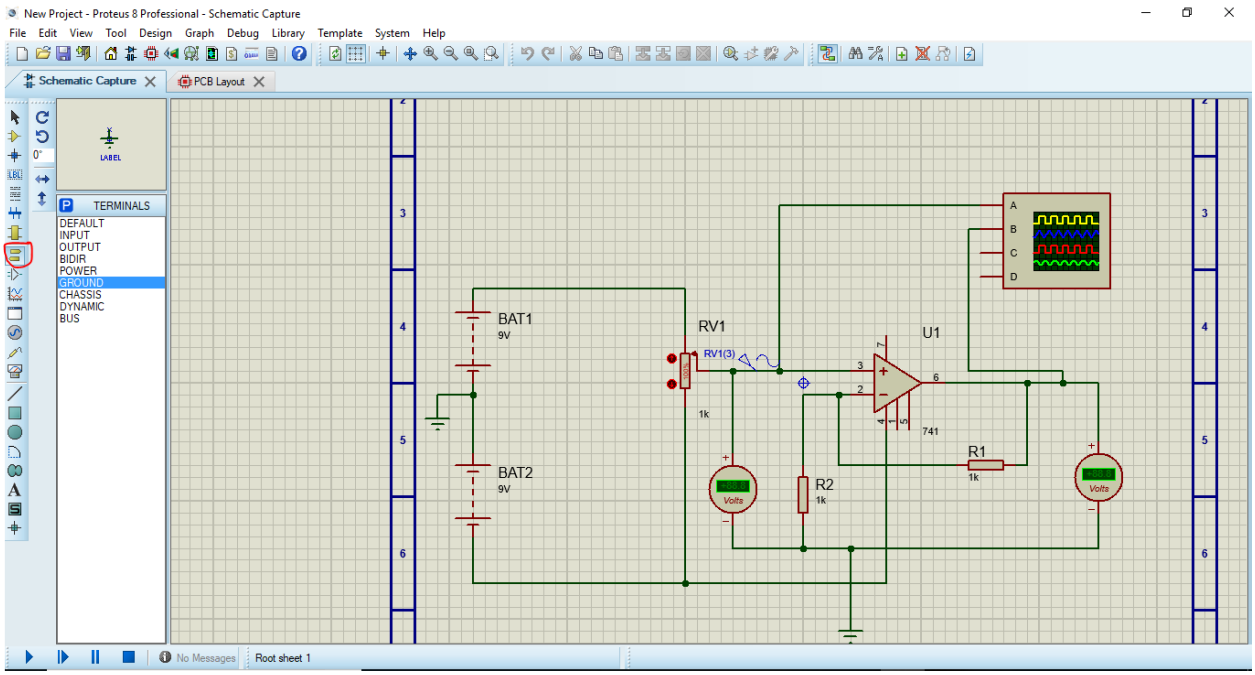
Figure 10 Pick Devices

searched for resistor. Keyword is res.

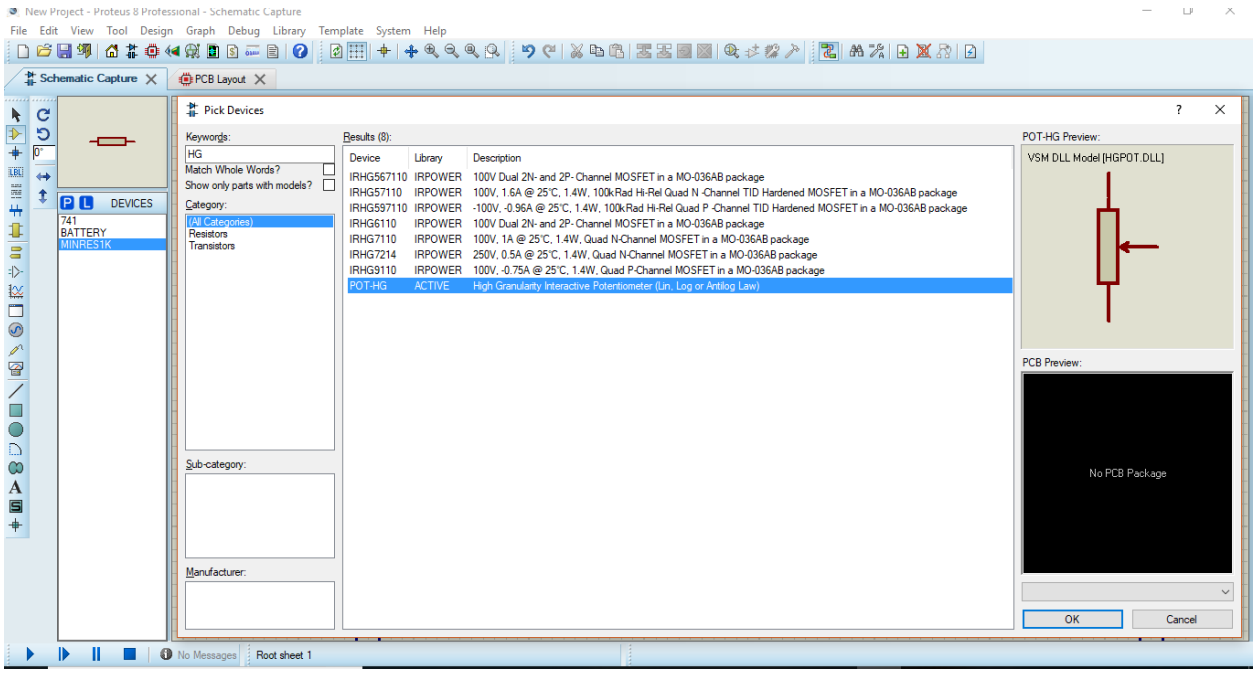
For selecting battery



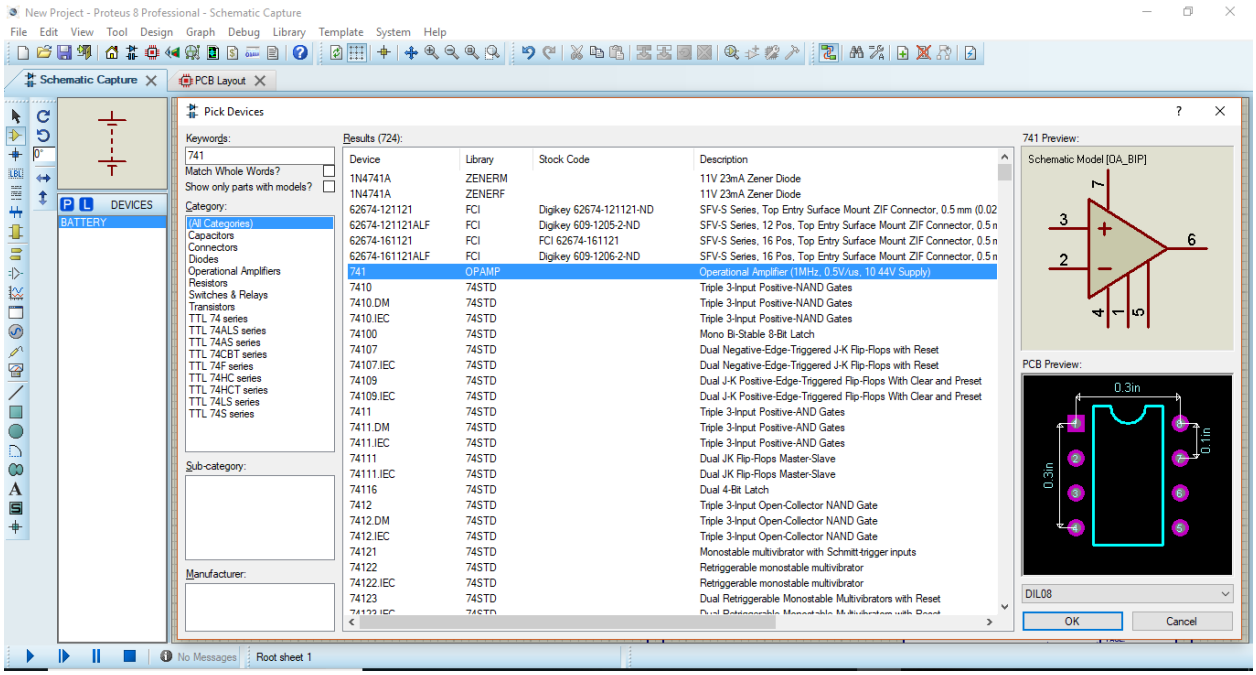
For selecting ground go to the Terminal Mode as shown below and Select GROUND



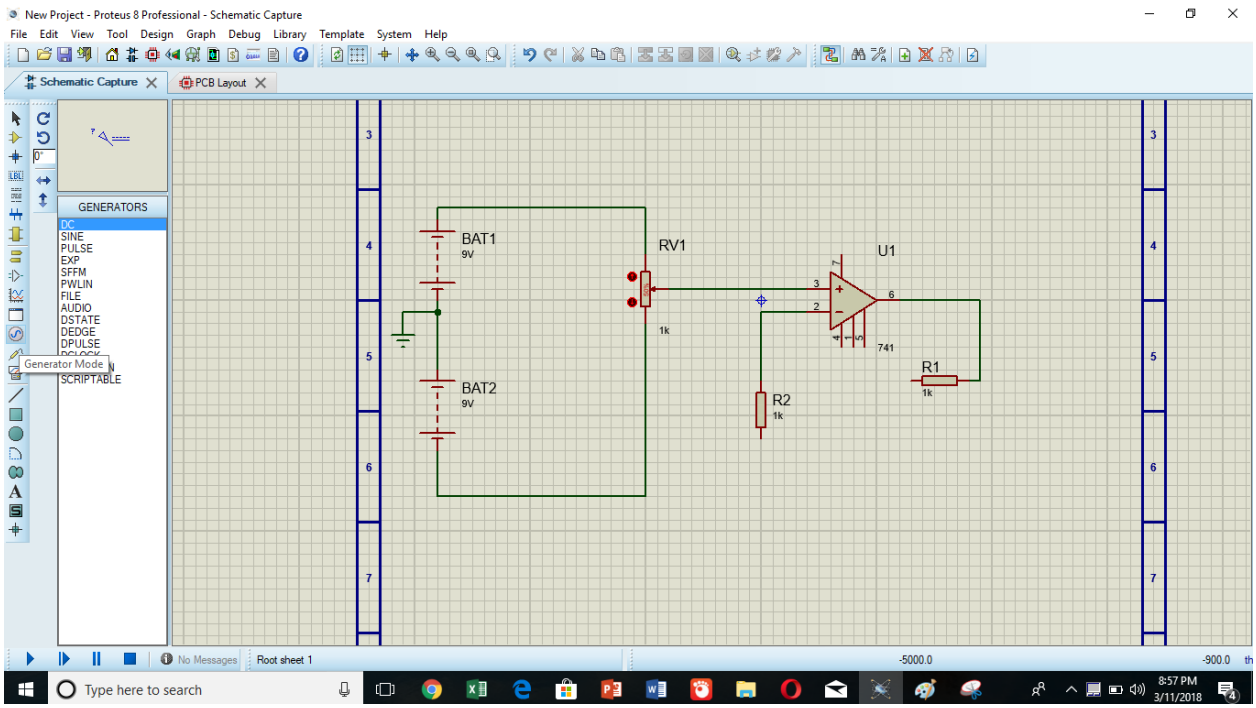
For potentiometer go to component mode and type HG



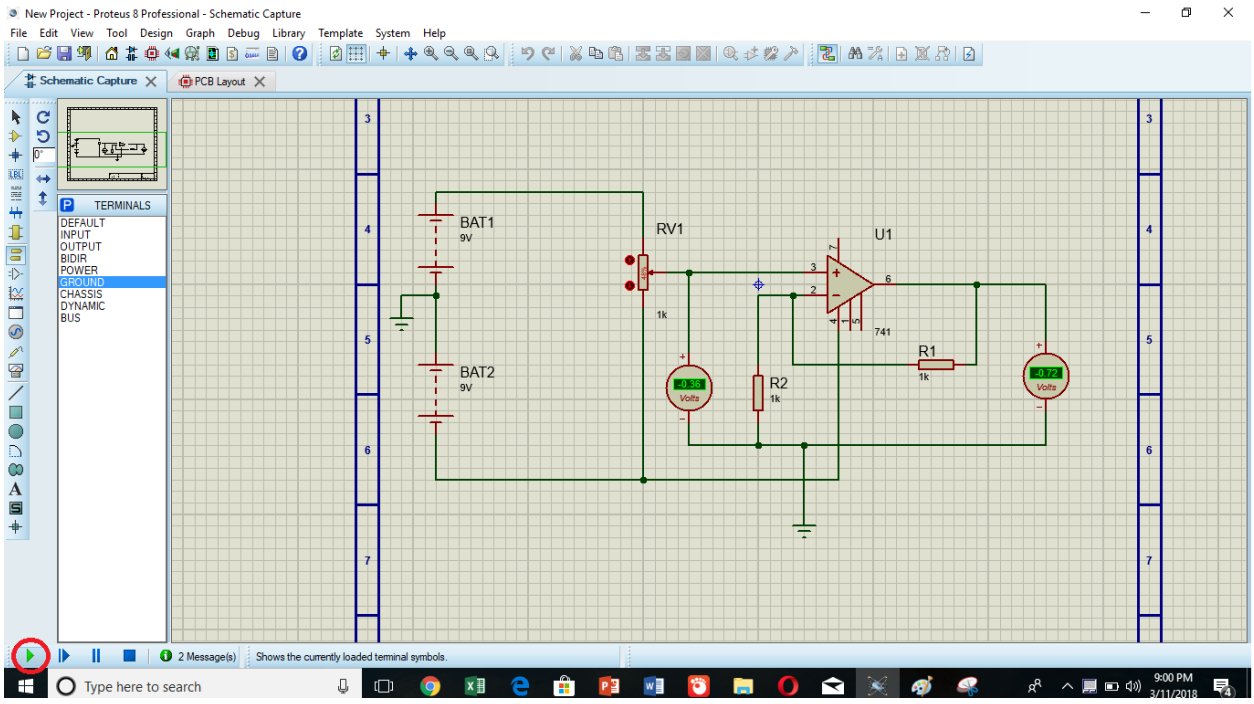
For OpAmp type 741 or any model number



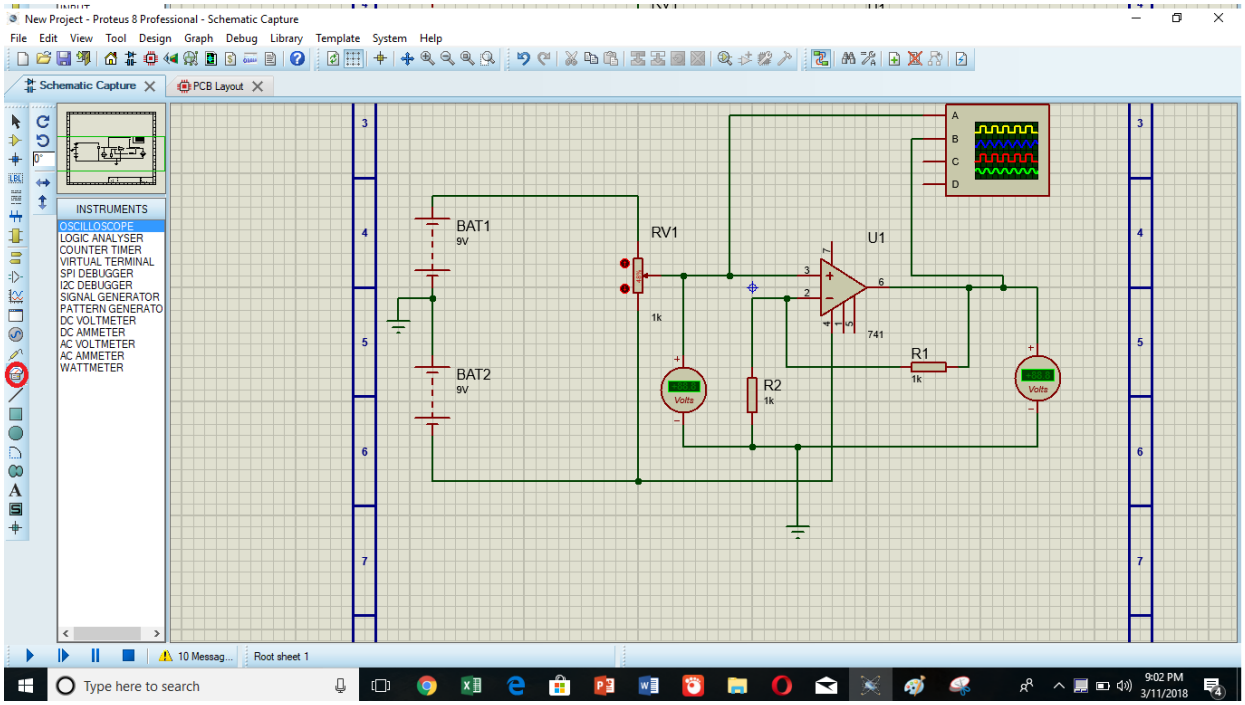
You can select DC or Sine in Generator mode as shown below



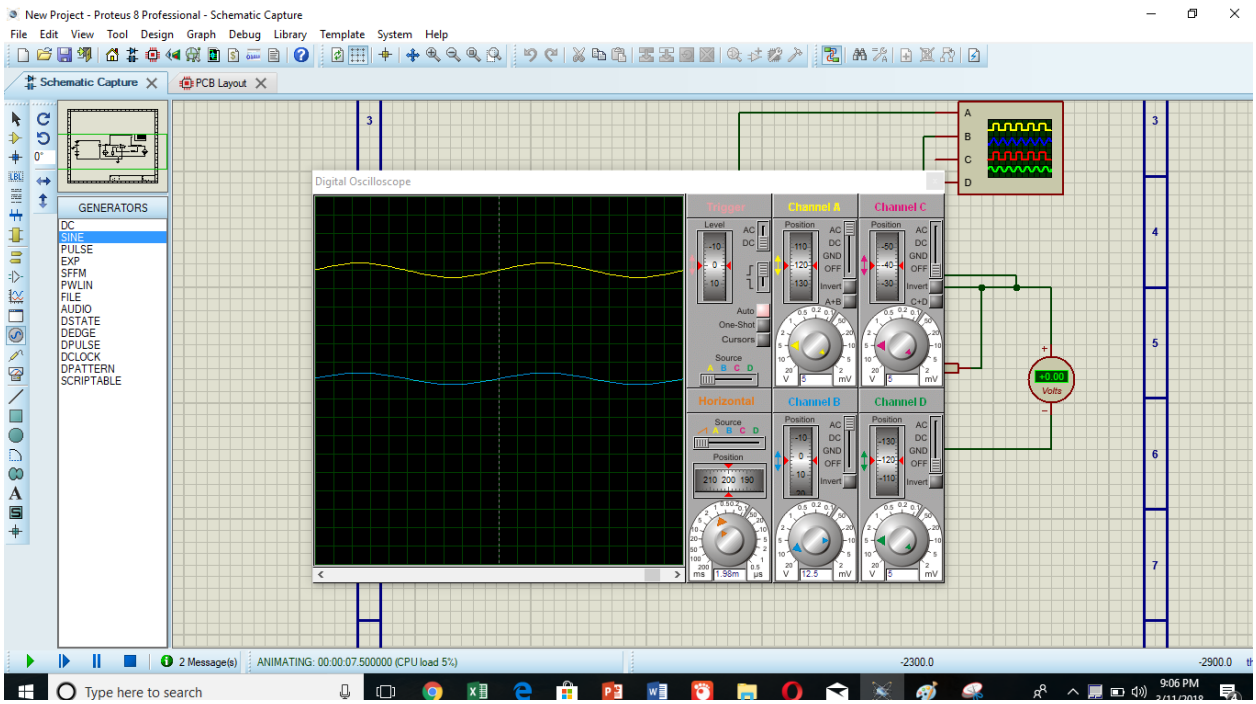
Click on the arrow in the bottom left for simulation



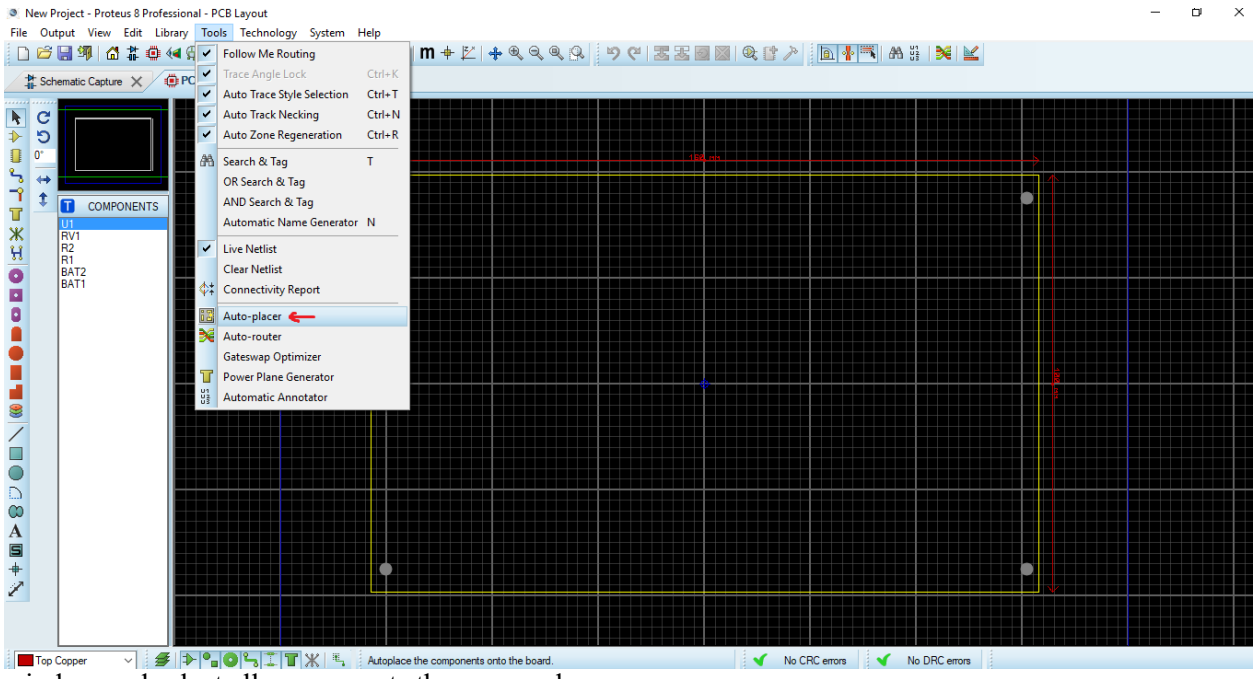
Now if you want to see the waves on oscilloscope you can select it from Instruments where you can also get voltmeter, ammeter etc



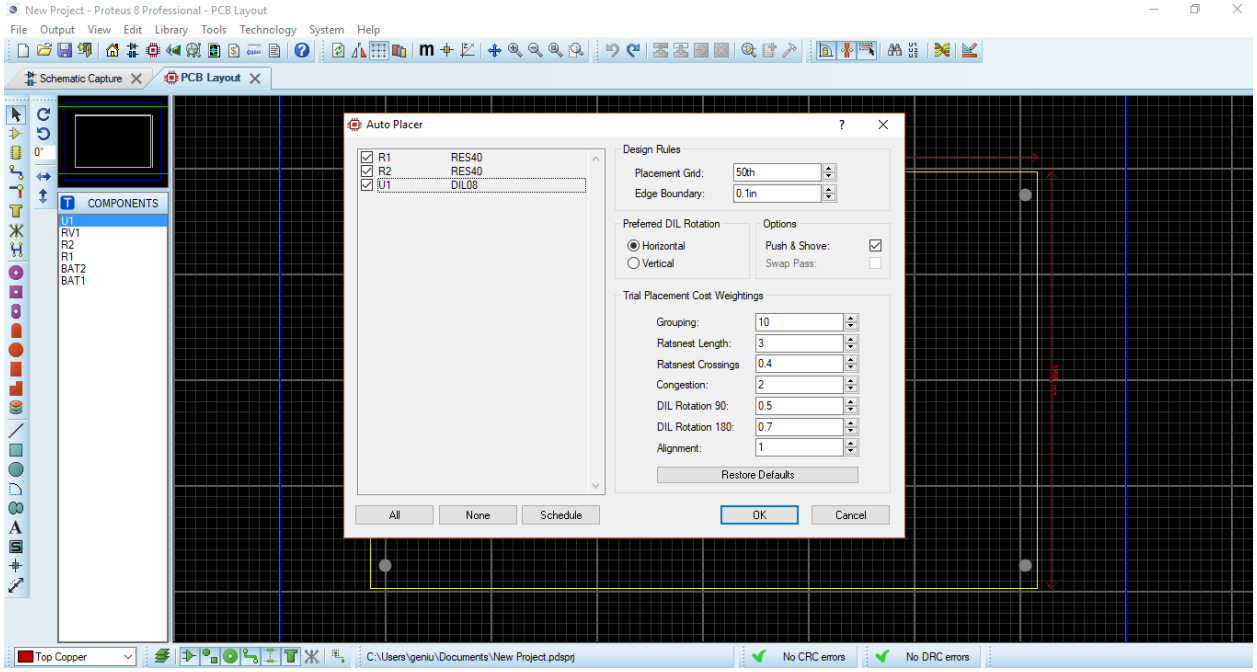
Here is the oscilloscope simulation



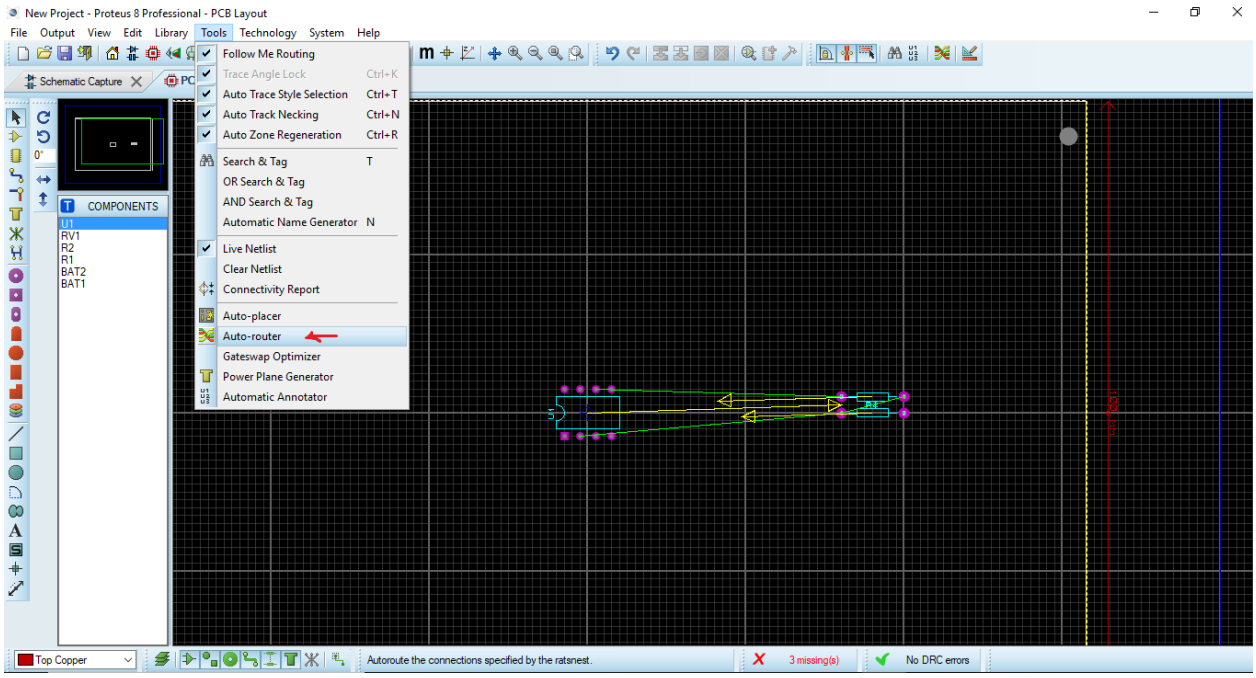
For PCB layout go to tools and click on auto placer



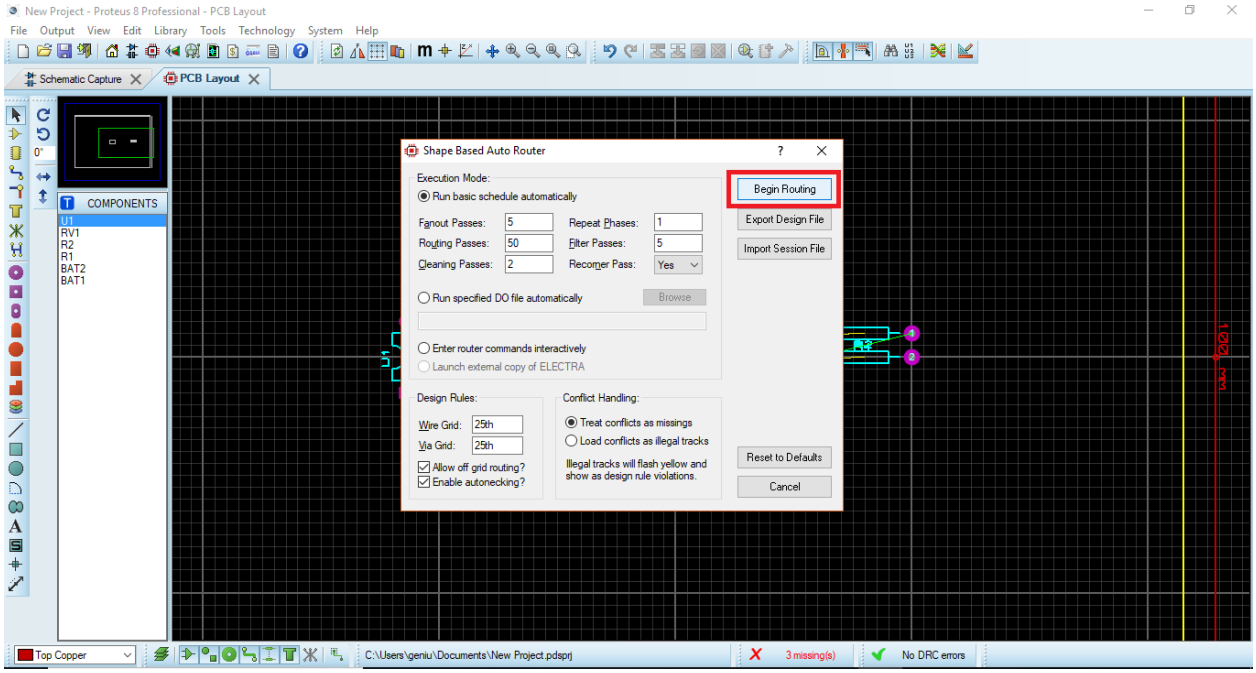
Then you will get this window and select all components then press ok



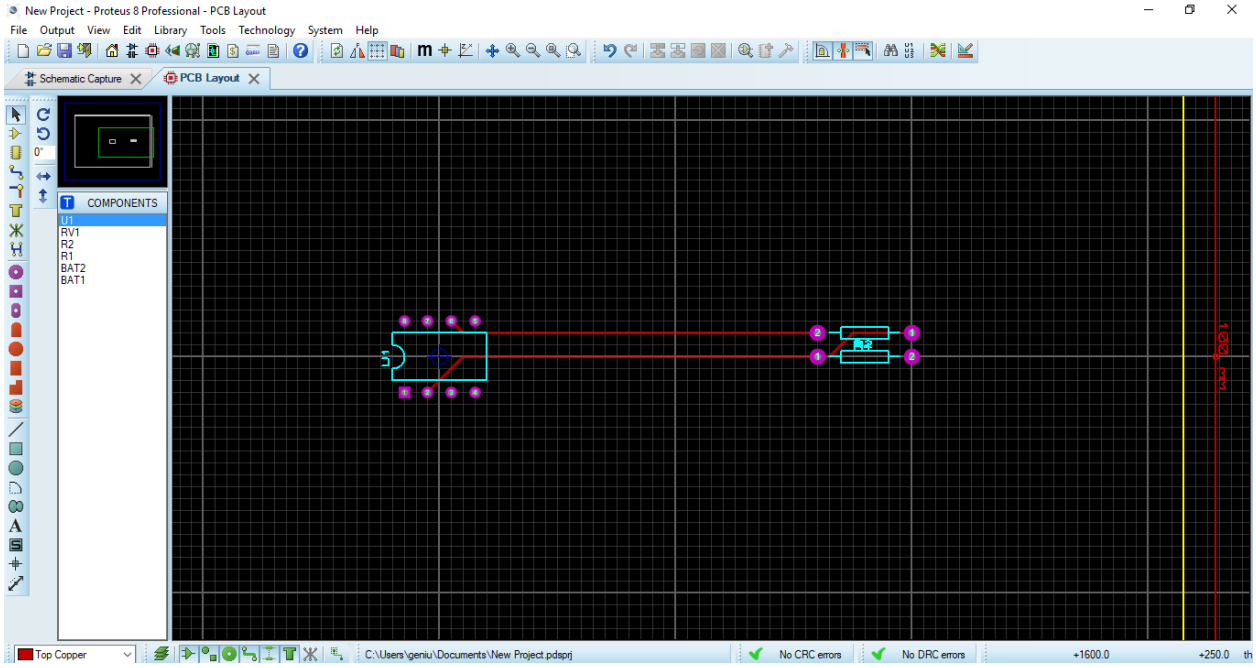
Now for routing purpose go to tools and select auto router



Click on begin routing



You will get the output result


New Project - Proteus 8 Professional - PCB Layout
File Output View Edit Library Tools Technology System Help
Schematic Capture
PCB Layout
COMPONENTS
U1
RV1
R2
R1
BAT2
BAT1
Top Copper
C:\Users\geniu\Documents\New Project.pdsprj
No CRC errors
No DRC errors
+1600.0
+250.0