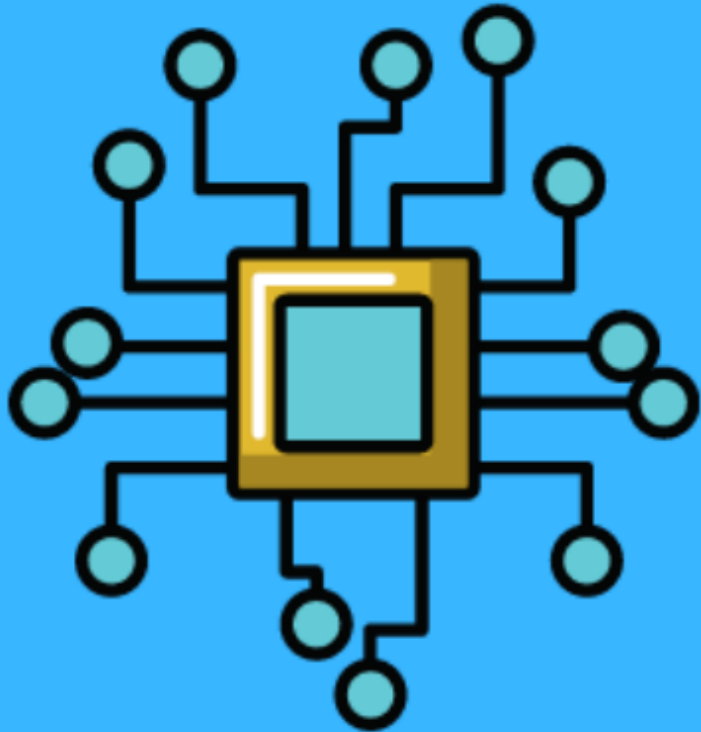


Electronics



Lab 2

Proteus- Installation guide

Eng. Mohamed Goda
Eng. Manar Ibrahim
Eng. Ismail Ahmed

What is ?

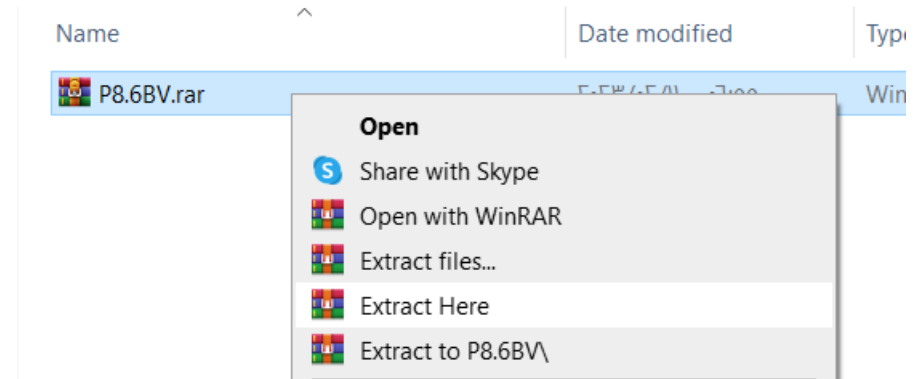
Proteus Design Suite : is a proprietary software tool suite used primarily for electronic design automation, to create schematics and electronic prints for manufacturing printed circuit boards.



Installation guide

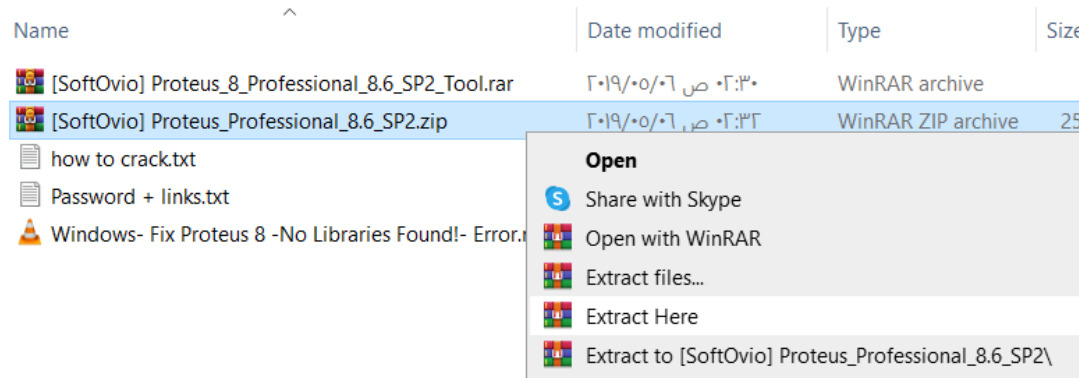
1. Download “P8.6BV.rar” file. you can find the download link on Electronics Microsoft Team

2. Extract .rar file. you need to Enter the Password “12345”

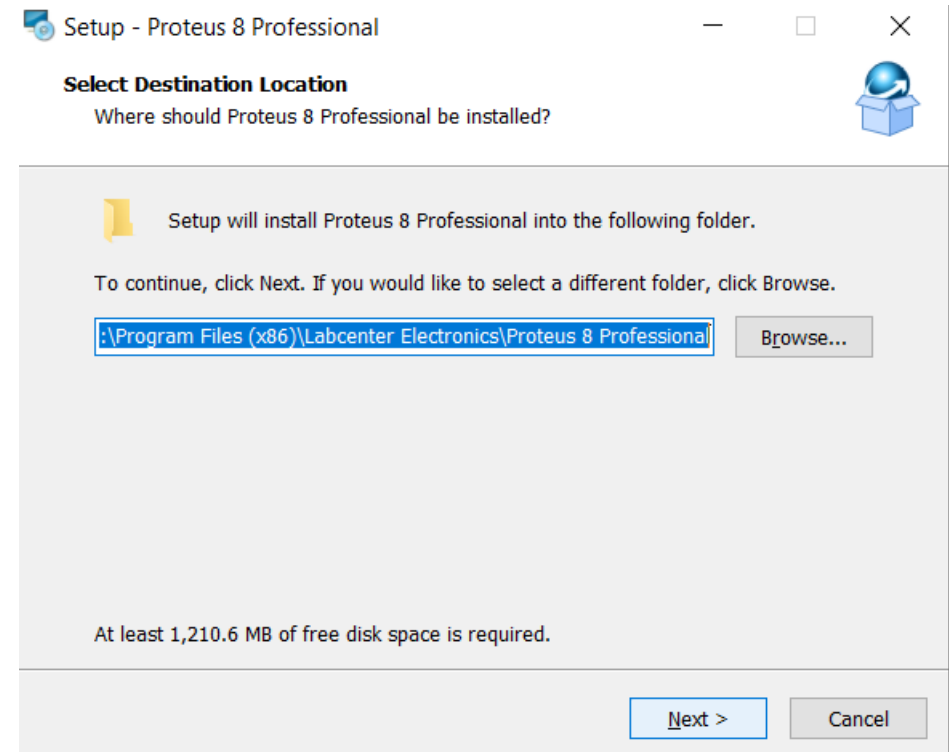


Installation guide

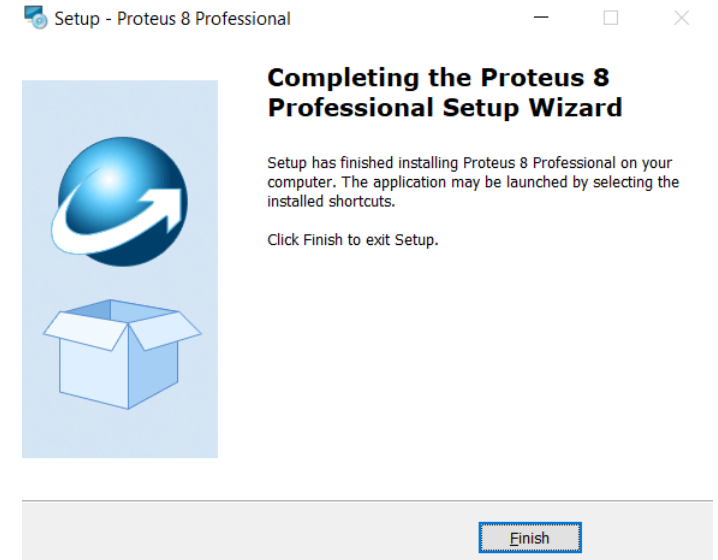
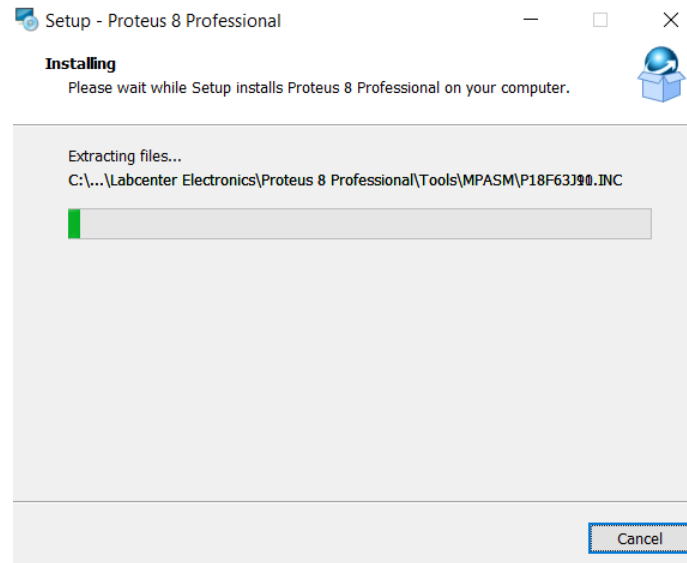
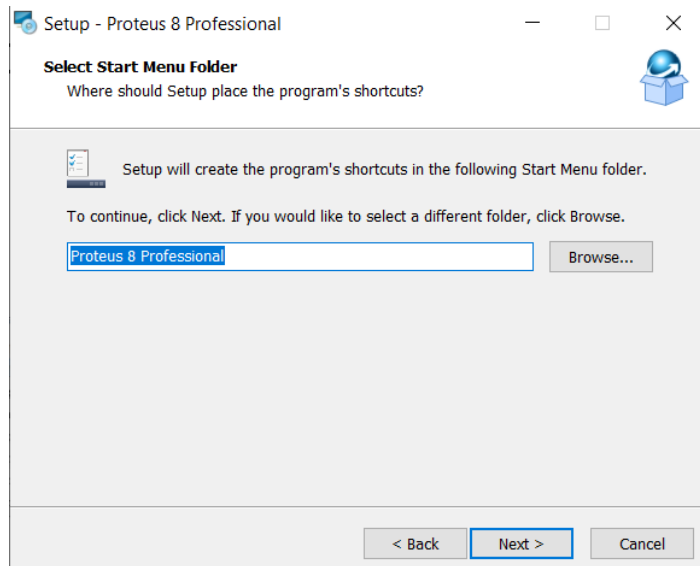
3. Extract the .zip file. then, Double Click on “Proteus_8.6_SP2_Pro.exe”



4. Follow the installation setps



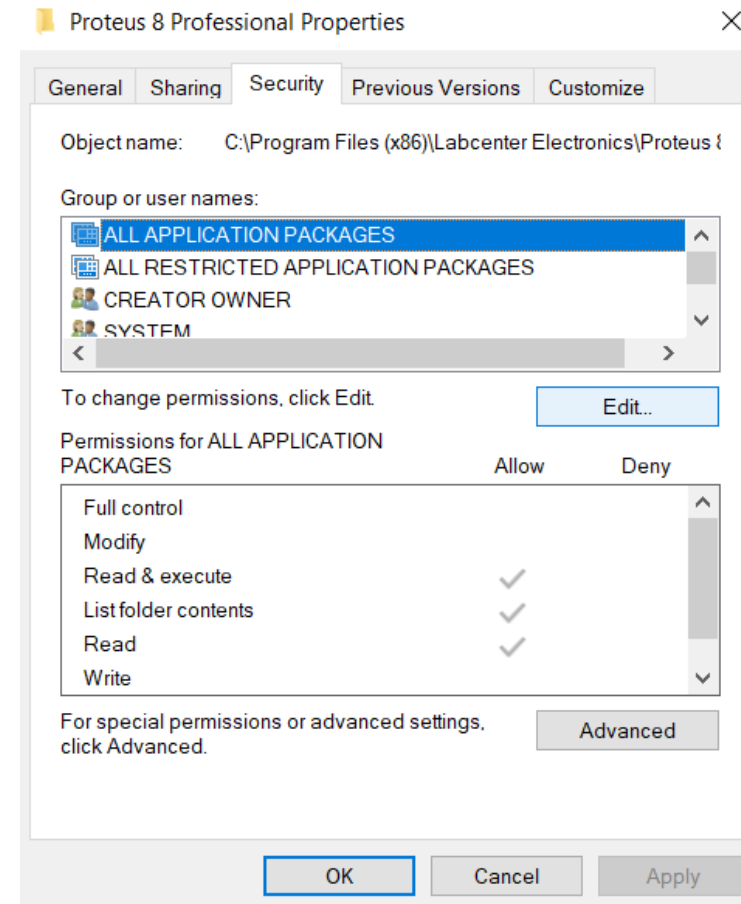
Installation guide



Installation guide

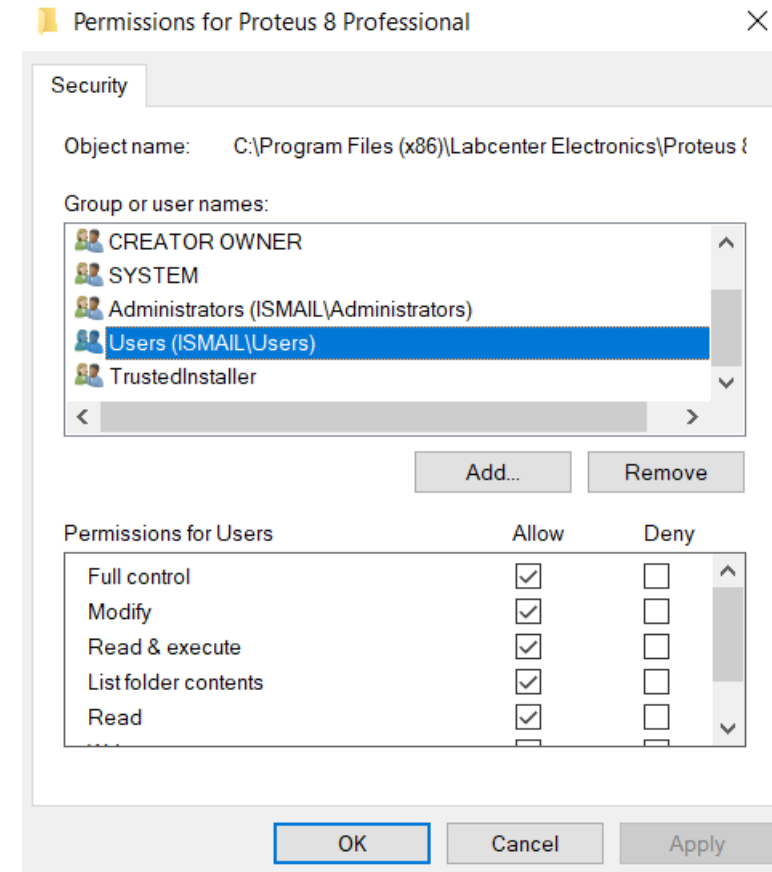
You have to Fix Proteus 8 - No Libraries Found Error

5. on your Computer, go to this path
“C:\Program Files (x86)\Labcenter Electronics”
6. Open the Properties of “Proteus 8 Professional”
folder. from the Security tab, Click Edit Button

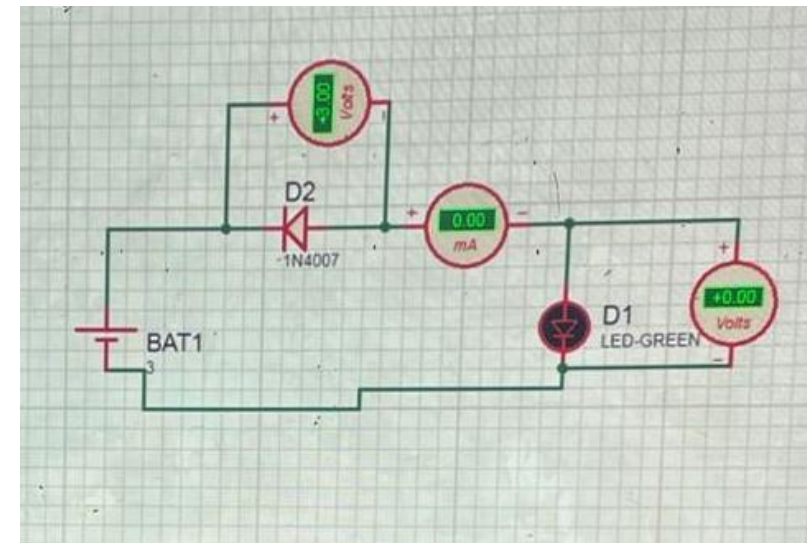
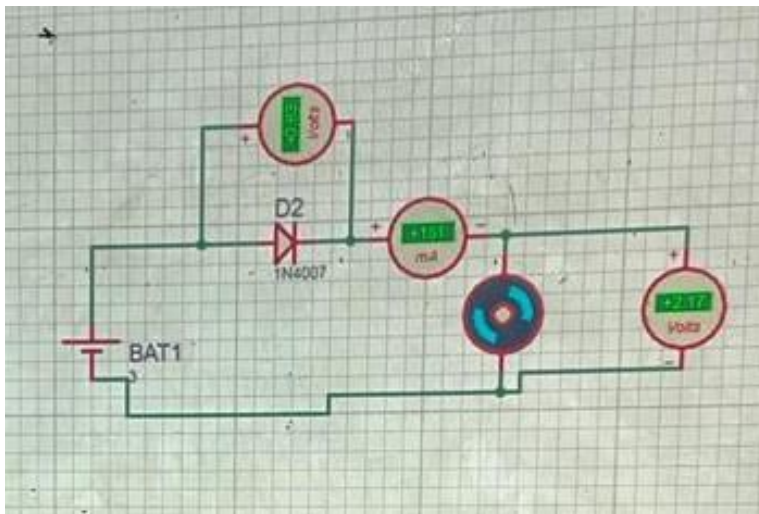
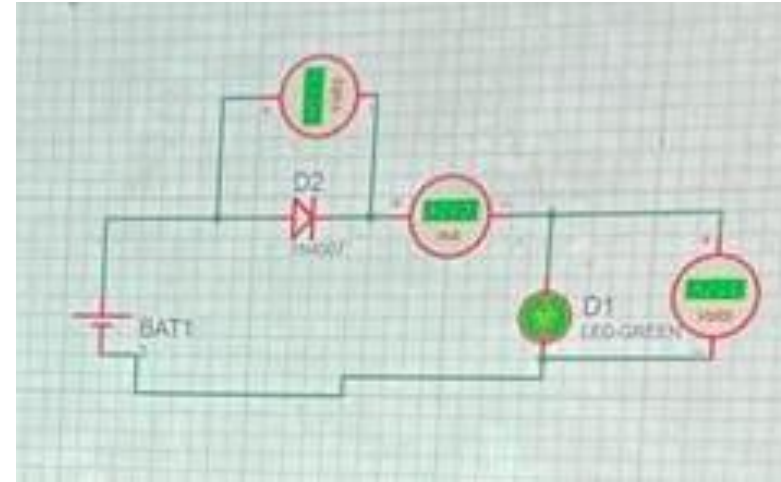
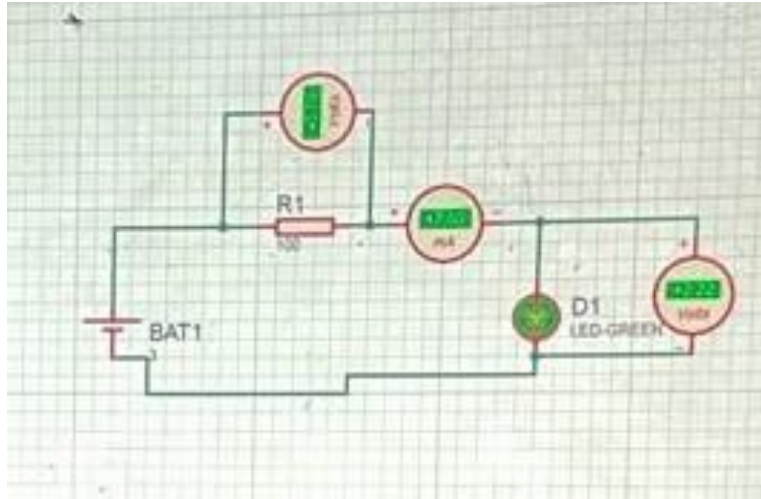


Installation guide

7. Choose “Users(DesktopName\Users) and give it Full control. then, Click Apply, OK



Circuits that we design in this Lab



Enjoy
with
your
Design

**Thank
You**