**2.4: Installing PSpice and simulating a circuit**

**PSpice Installation Files:**

Click the link to download PSpice 9.1 student version:

[PSpice\_student\_version\_9.1](https://drive.google.com/file/d/1VOtUYsaFzPAUfJTuCCqaetKoTI-eOOYB/view?usp=sharing)

Here is an alternative version of PSpice (9.2):

[ORCAD PSpice 9.2 family release](https://drive.google.com/file/d/1oXTMyhgSKoiPOlfoFOoZflTsFXX97GA1/view?usp=sharing)

**FAQs:**

**1. Why are there two versions of PSpice? Do I need to install the both?**

--> You just need to install any one of the two. PSpice student version (9.1) has some compatibility issues. It may not be installed in some PCs. But student version is very light-weight and uses less resource. So, I am suggesting you to try installing the student version first.

**2. I opened the installer file for PSpice student version (9.1). But nothing shows up (as if nothing happened).**

--> If that's the case, you have to install the alternative version (9.2). But make sure you watch the first video because you will get an introduction on how to simulate a circuit with PSpice in that video.

**3. Can I install PSpice in any drive of my PC?**

--> You may do that. But I am not suggesting that. Please install the software in the default directory (C:\ drive).

**4. I have installed the software properly. But I cannot find any component when I press the "Get New Part" button (or it says some library files are missing).**

--> If that's the case, you have to run the software with Administrator permission. When opening "PSpice Design Manager" right click on it and select "Run as administrator". Hope it will solve the issue.

**5. None of the installers runs on my PC. What should I do now?**

--> Then you have to do a workaround. Actually the installer runs in the background. You can find that from "Task Manager". The installer waits for some processes to be completed or closed. Do the following steps:

Open "Task Manager" --> Go to "Details" tab --> Find out "SETUP.exe" --> Right click on it and select "Analyze wait chain" --> Select the checkbox of the drop-down menu and end the process (you may have to do this step multiple times)

If you still face any problem, please let us know in the discord server.

CSE250\_Lab2\_4: Installing PSpice and simulating a circuit

<https://www.youtube.com/watch?v=VlLbpttMBYY>

**Installing Alternative version of PSpice (9.2)**

<https://www.youtube.com/watch?v=4e3IqOK6JKE>

**2.5 Alternatively Simulating on Proteus**

Lab2: Series Parallel Circuit Simulation on Proteus

<https://www.youtube.com/watch?v=qX_tgowYHts>