

Introduction to the basics of Proteus

The purpose of this document is to provide a quick start guide when using Proteus 8 Professional schematic design. It is not meant to be an exhaustive document. Additional information can be found by searching the internet or the built-in help files.

Installing Proteus 8 Professional

If Proteus 8 Professional is not yet installed on the PC you are working on, follow the steps outlined in this document: <http://ictcms.tut.ac.za/moodle/mod/resource/view.php?id=40>

Please note:

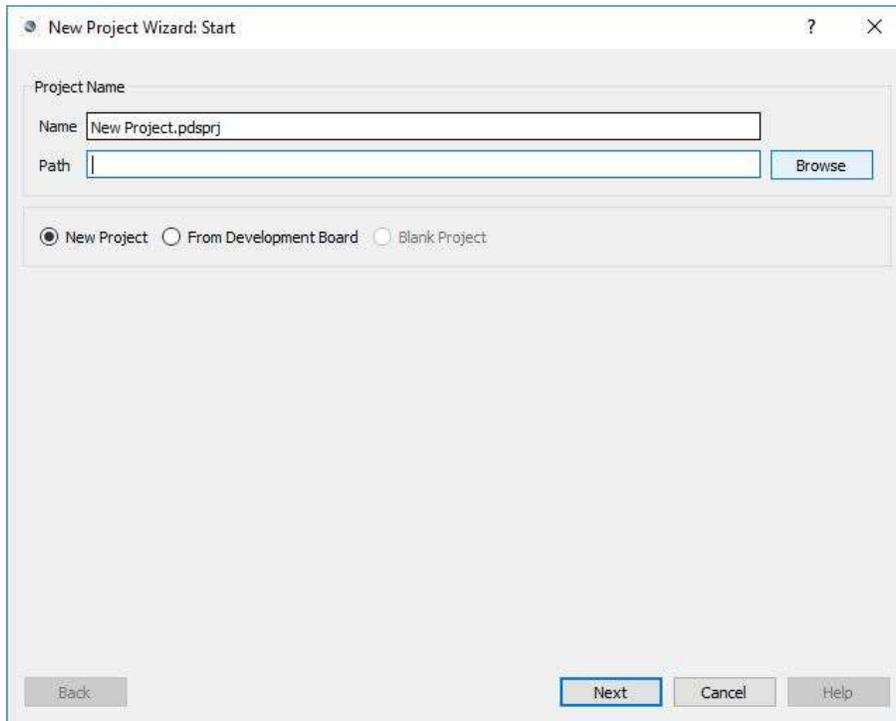
- The Department of Computer Systems Engineering, has a licensing server for Proteus 8 Professional, however you must be on the Soshanguve South campus of TUT, and you must be logged in to the TUT network.
- Proteus 8 Professional is currently not available off-campus (Soshanguve South Campus)

Create a new Proteus project

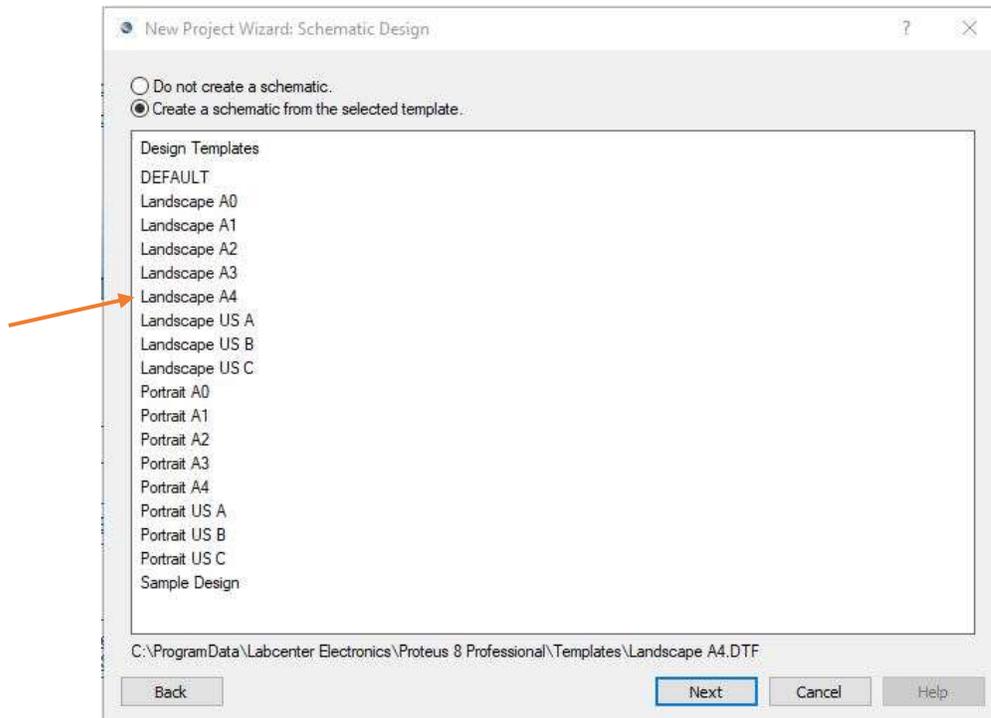
1. Open Proteus 8 by finding the icon on the Windows Desktop or by searching for the program in the Windows Menu.
2. Create a new Project by clicking on New Project



3. Enter the name of the project and select an appropriate location to save the project, when finished, click NEXT.



4. Select the following template: Landscape A4 and click NEXT

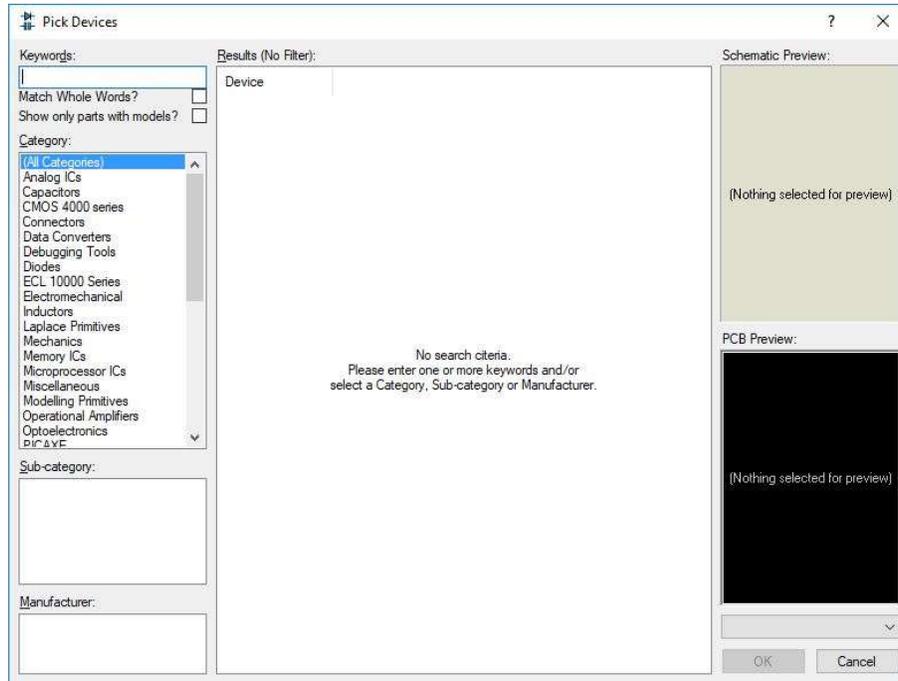


5. Ensure the 'No Firmware Project' is selected and click NEXT. (If not going to simulate the PIC or 8051 controllers)
6. Confirm all the settings are correct and click on FINISH.

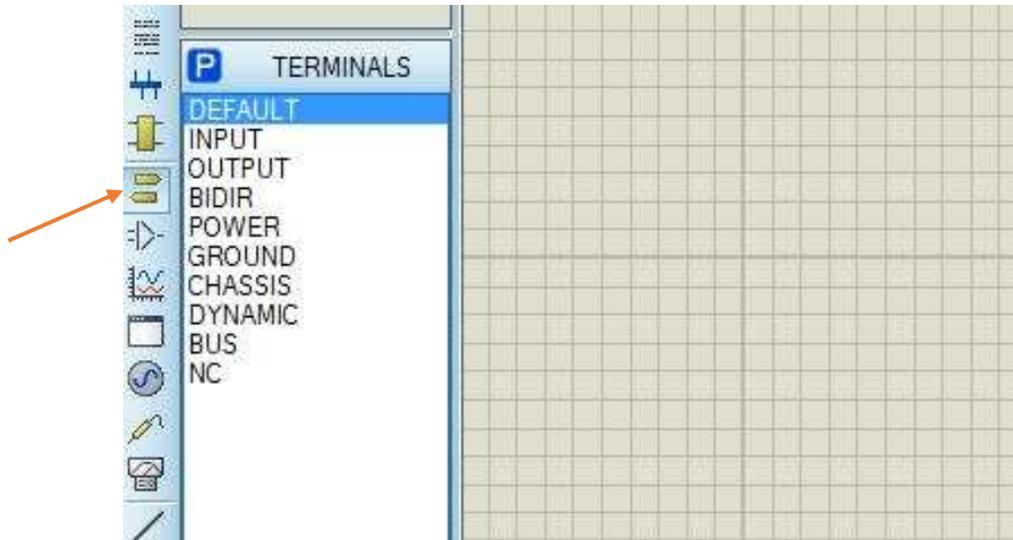
7. Your project is now created, and a schematic sheet will open. Use this sheet to draw, design and simulate your circuit.

Create the circuit diagram

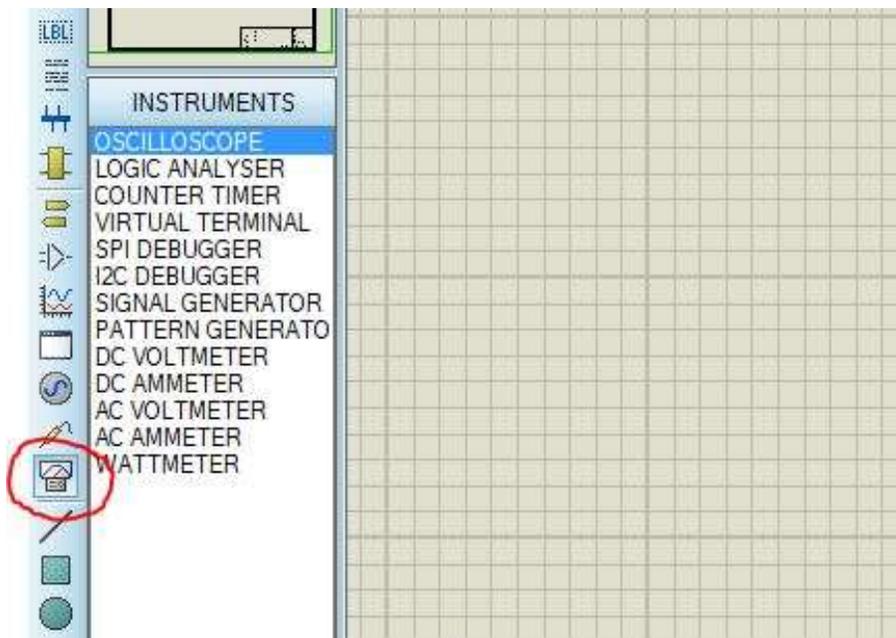
1. Open the 'Pick Devices' window by pressing the 'P' key or by using the right-click context menu.



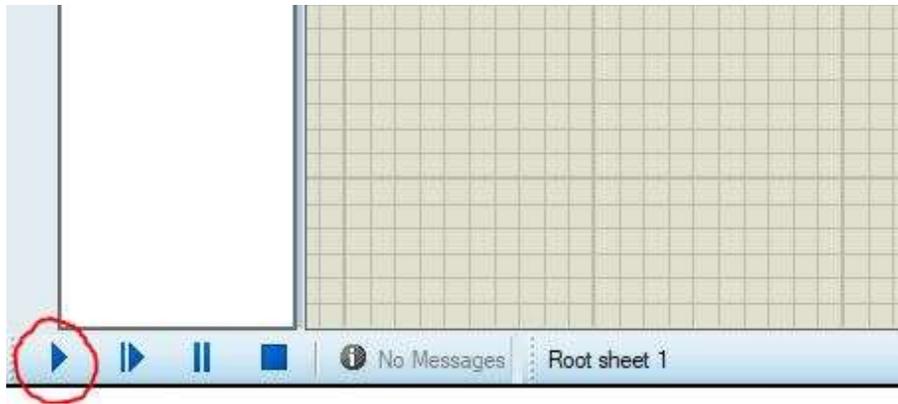
2. In the 'Keywords' text box enter the name/number of the component you want to search for.
3. Select the desired component from the search results and click the 'OK' button.
4. Click on the schematic sheet where you want to place the component.
5. Continue the above process until all the components have been placed.
6. Connect component pins together by clicking on the starting pin and then clicking on the pin of the component you want to connect to.
7. Add Power and Ground terminals to your design by using the Terminals sub menu.



8. Add some virtual instruments to test your design by clicking on the Virtual Instrument icon in the left-hand toolbar and selecting the desired instrument.



9. After the schematic is complete, click on the run arrow at the bottom left of the project window.
10. Use the virtual instruments and probes to verify your design is working according to specifications.



11. Click on the STOP button when you are done with your simulation.