If you plan to use PSpice, perform the following steps in the sequence indicated (this sequence is important):

Installing PSpice:
On the website click on the Install_PSpice_9.1 and extract all of its content in a separate folder. Double-click on Setup.exe to start PSpice installation. Make sure to choose Schematics (NOT Capture) in the screen titled “Select Schematic Editors” and default settings on the following screens. PSpice should be installed in the C: Drive (not in any other Drive).

Installing Simulation Files and the Library:

Follow the steps below:

1) On the Website, Click on “PSpice Power Electronics Lab Schematic Files” and extract all its content to “C:\FirstCourse_PE_Book09”. Note: C: Drive (not in any other Drive).

2) Launch PSPICE Student version Schematics program.

3) Go to Options menu and select Editor Configuration.
   - Click on Library settings.
   - Click on Browse and type 'C:\FirstCourse_PE_Book09\Library' and select 'First_Course_PED' library.
   - Click on 'ADD*' button. The library you are adding is a symbol library so make sure that the Symbol check box is ‘Checked’.
   - Click OK.
   - In the Editor Configuration dialog box, click OK.

4) Go to Analysis menu and select Library and Include Files.
   - Click on Browse and type 'C:\FirstCourse_PE_Book09\Library' and select 'First_Course_PED' library.
   - Click on 'Add Library*', then on 'Add Include*'.
   - In the Editor Configuration dialog box, click OK.

5) Go to Analysis menu and select Library and Include Files.
   - Click on Browse and type (or Browse for) ' C:\Program Files\OrCAD_Demo\Capture\Library\Pspice\eval.lib '. Click on 'Add Library*', then on 'Add Include*'.
   - In the Editor Configuration dialog box, click OK.

6) YOU ARE DONE!