Instructions for working with PSpice and schematics files

1. Copy the files in the "PE_PSpice_Files" folder TBE-B311 desktop in your USB drive and carry with you to TBE A311
2. Log into one of the computers in A311, copy the above folder from your USB drive to the desktop.
3. Click on "Programs", select "Cadence", then select "Design Entry CIS".
4. In the pop-up window that lists Cadence Product Choices, select "Allegro PCB Design CIS L"
5. Now, you can open on of the schematic files by simply following the standards steps, i.e., click on file, open, project, location of project etc