

Guidelines on working with Electric VLSI design tool

EE619A (2022)
(ruchiv21@iitk.ac.in)

1. Link to Install runtime Java Environment : https://download.cnet.com/Java-Runtime-Environment-JRE/3000-2213_4-10009607.html
2. Link to install java 3D for windows: <https://www.softpedia.com/dyn-postdownload.php/510785460aeeb80767d64faf58222758/635fb775/27c79/0/1>
3. Link to download Electric Vlsi design tool: <https://www.staticfreesoft.com/productsFree.html> . Click on this link and download -Electric Binary version 9.07.
4. If you have LTspice then okay but if you do not have then you can download it from this link: <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
5. After this download pdk file which you want to use (45nm or 180nm) in text document format(.txt).
6. Once you install all these, place them in the same drive (D or E) and in the same folder(for example EE619_layout_project).
7. Electric is a tool where we can only design schematic and layout and to run the simulation electric uses LTspice. So we need to integrate LTspice with Electric and to do so you can follow this documentation: http://cmosedu.com/cmos1/ltspice/ltspice_electric.htm or the steps are mentioned bellow:
 - Click on electricBinary-9.07 → go to file → preferences → Tools → Spice/CDL → choose
Spice engine: Spice3
Spice Level: 3
 - Then in Running Spice:
After Writing Deck: choose Run, Ignore output
Run Program: write the location of LTspice(where it is intall).
With args: -i \${FILENAME} -r \${FILENAME_NO_EXT}.raw -o \${FILENAME_NO_EXT}.out
 - After this click on run probe. Then Apply and OK, now the LTspice has been integrated with Electric.
- 7.a. Again go to file → preferences → technology → technology → choose
Startup Technology: mocmos
Layout Technology to use for Schematic: mocmos, then click on submicron rules, analog then apply
And OK.
- 7.b. Again go to file → preferences → technology → Scale → choose mocmos and for 45nm pdk write 22.5nm in
The technology scale box. Then Apply and OK.

8. To design any schematic, click on file → new library (type library name, for Example: Rollnumber_Project) → select library and click on cell and select schematic. Now you are ready to design schematic using different components.
9. For design purpose you can prefer this youtube video link: <https://youtu.be/PzOe6Bl895A> for schematic design process and <https://youtu.be/1y5zWQYwo1E> this link for layout design.
10. After schematic design, you need to write spice code (same as you were using for LTspice simulation) to run the simulation which you can write by clicking on **Misc** (in component), → spice code.
11. Now to use 45nm pdk you will need to include the location of pdk file where is saved (for example :D/EE619_Layout_project/45nm.txt)
12. For simulation: go to Tool → Simulation(Spice) → write spice deck , LTspice window will open, then right click → add traces.





