To setup Electric and LTspice, please do the following:

1. Download the Electric tool in some folder (Preferably create a new folder)
   Click on this url https://www.staticfreesoft.com/productsFree.html
   Select following download: **GET THE GNU ELECTRIC BINARY RELEASE, version 9.07**

2. Install runtime java environment
   from https://www.oracle.com/technetwork/java/javase/downloads/jre8-downloads-2133155.html

3. Once you have installed JRE of step-2, you can invoke ELECTRIC by double clicking on **electricBinary-9.07** that you downloaded in step-1

4. Install Java 3D on your system. Please refer to
   https://download.java.net/media/java3d/builds/release/1.5.2/README-download.html

5. Install LTspice on your system. You can download LTspice from
   https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

6. To integrate LTspice with ELECTRIC, take help available
   at http://cmosedu.com/cmos1/ltspice/ltspice_electric.htm

Please get this setup ready so that we can start learning CMOS Schematic and Layout.