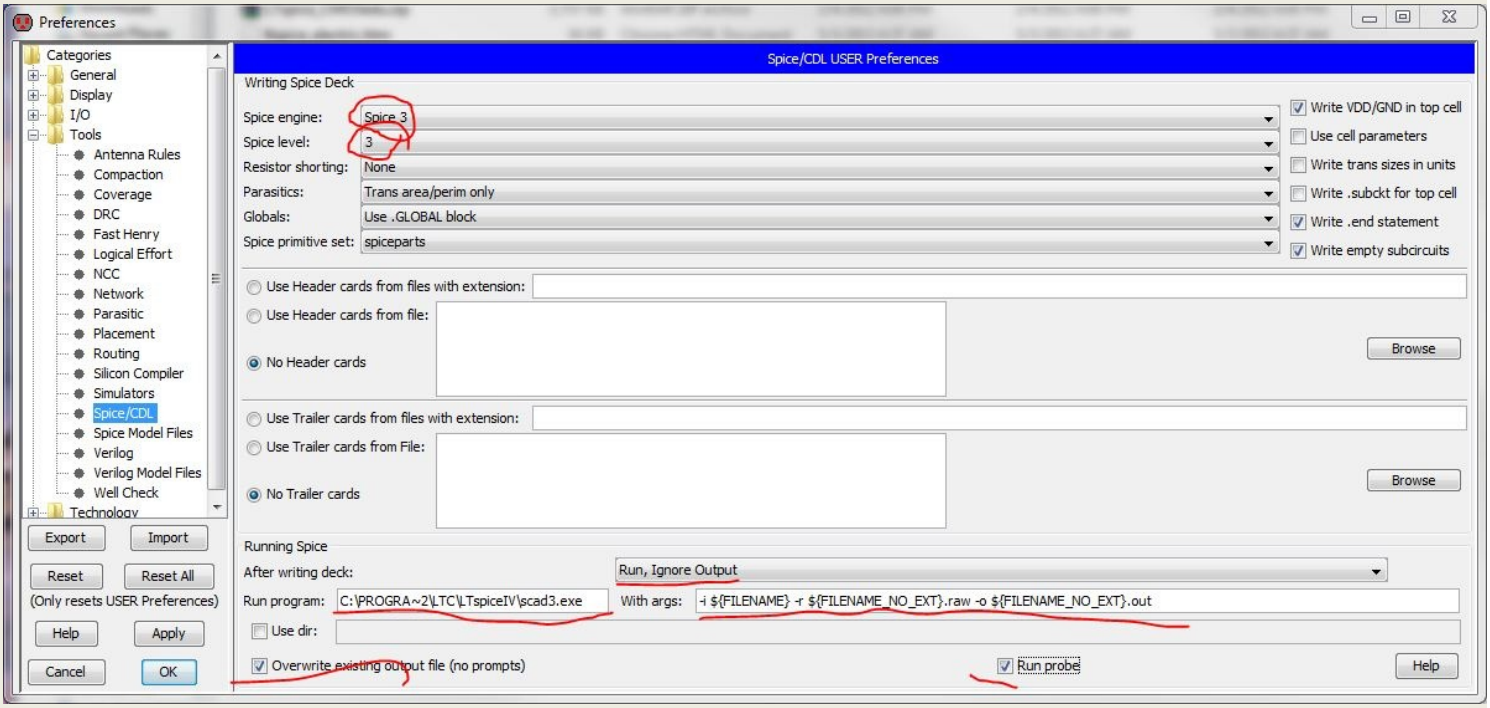
Setting LTspice up for use with Electric

Return to the LTspice page at CMOSedu.com or Return to the Electric VLSI page at CMOSedu.com

- Ensure LTspice is installed on your computer.
- Start Electric and go to File -> Preferences (see image below). The Run Program path is either:
  C:\PROGRA~1\LTC\LTspiceIV\scad3.exe or
  C:\PROGRA~2\LTC\LTspiceIV\scad3.exe or, if using LTspice XVII,
  C:\PROGRA~1\LTC\LTspiceXVII\XVIIx64.exe
  C:\PROGRA~2\LTC\LTspiceXVII\XVIIx64.exe
  where, using the 8.3 filename, C:\PROGRA~1\ is the same as C:\Program Files\ and C:\PROGRA~2\ is the same as C:\Program Files (x86)\.

- Note that while the Run Program field is not case sensitive the "with args:" field is case sensitive (so *use the uppercase names* as seen)
  For copying into the "with args:" field: **-i ${FILENAME} -r ${FILENAME_NO_EXT}.raw -o ${FILENAME_NO_EXT}.out**

- Cross-probing between layout or schematic using Electric's Probe window (instead of LTspice's probe), can be done with these setups if the –i (interactive) is changed to –b (batch)
  Ensure that in LTspice, Tools -> Control Panel -> Operation, raw files (simulation results) are not automatically deleted when LTspice is closed else you won't see the results!!!
  If you get an Exception Caught!!! when Electric loads simulation data it likely means that you have run out of memory. See number 15 here for how to increase the memory allocated to the JVM



Return to:
The LTspice page at CMOSedu.com
CMOS Circuit Design, Layout, and Simulation
CMOS Mixed-Signal Circuit Design