

## How to get LT spice working with tsmc018.lib in 5 steps

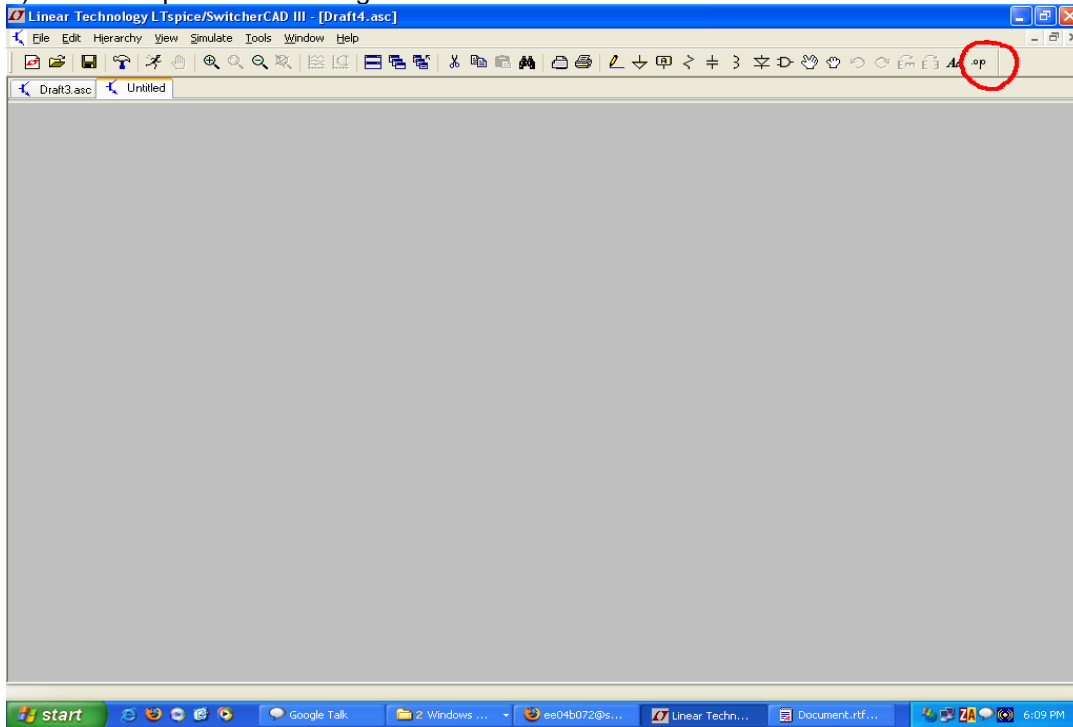
---

1) Copy the file tsmc018.lib to the directory Installationpath\LTC\SwCADIII\lib\sub (Usually it is C:\Program Files\LTC\SwCADIII\lib\sub) and

the files cmosn.asy and cmosp.asy to the directory Installationpath\LTC\SwCADIII\lib\sym (C:\Program Files\LTC\SwCADIII\lib\sym).

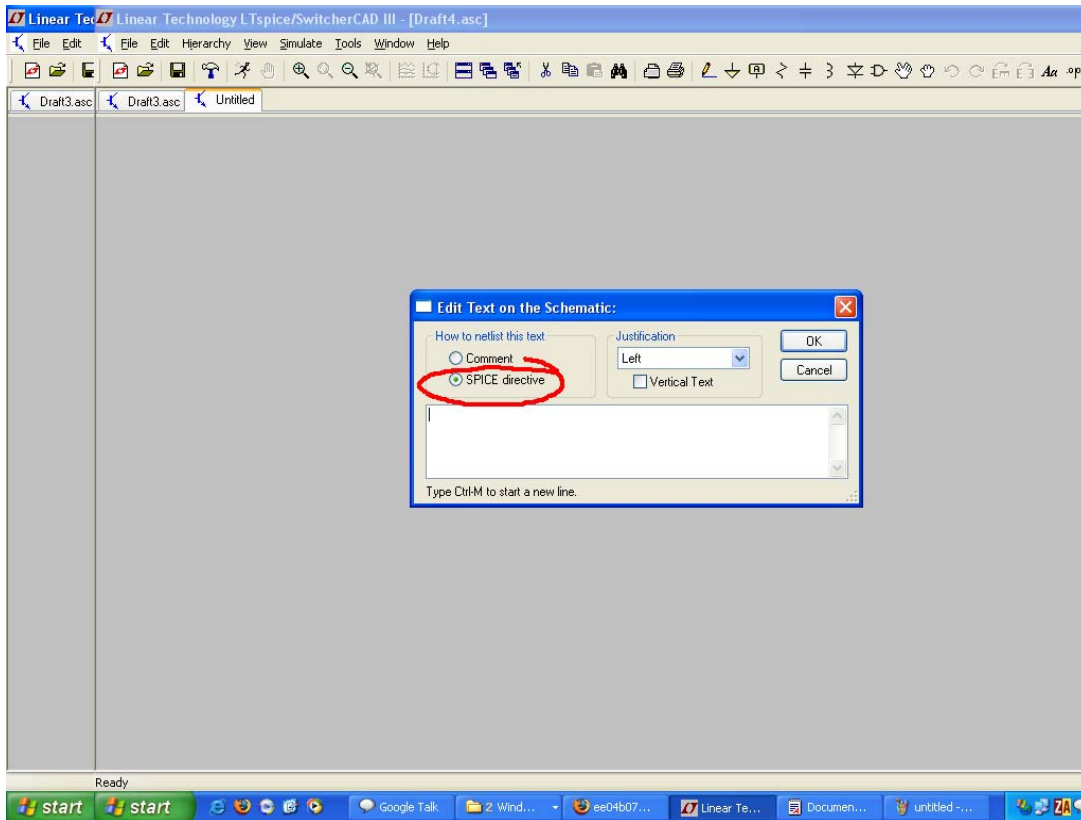
2) Load LT Spice : File-> New schematic

3) Click on .op as shown in Figure



4) Type : `.INCLUDE tsmc018.lib`

The SPICE directive must be chosen.



5) Select Component and Select cmosn or cmosp and u r done.

