

VLSI

Installation of TSMC 180 nm Technology Files in LT SPICE

&

NMOS & PMOS Characterization

Sanjay Vidhyadharan

Installing CMOS SPICE Model in LT SPICE

1. Download the following files from my webpage
<https://sanjayvidhyadharan.in/Downloads>

(a) tsmc018.lib

(b) cmosn.asy

(c) cmosp.asy

tsmc_180_nm.zip and extract it

2. Download LTSPICE and Install it

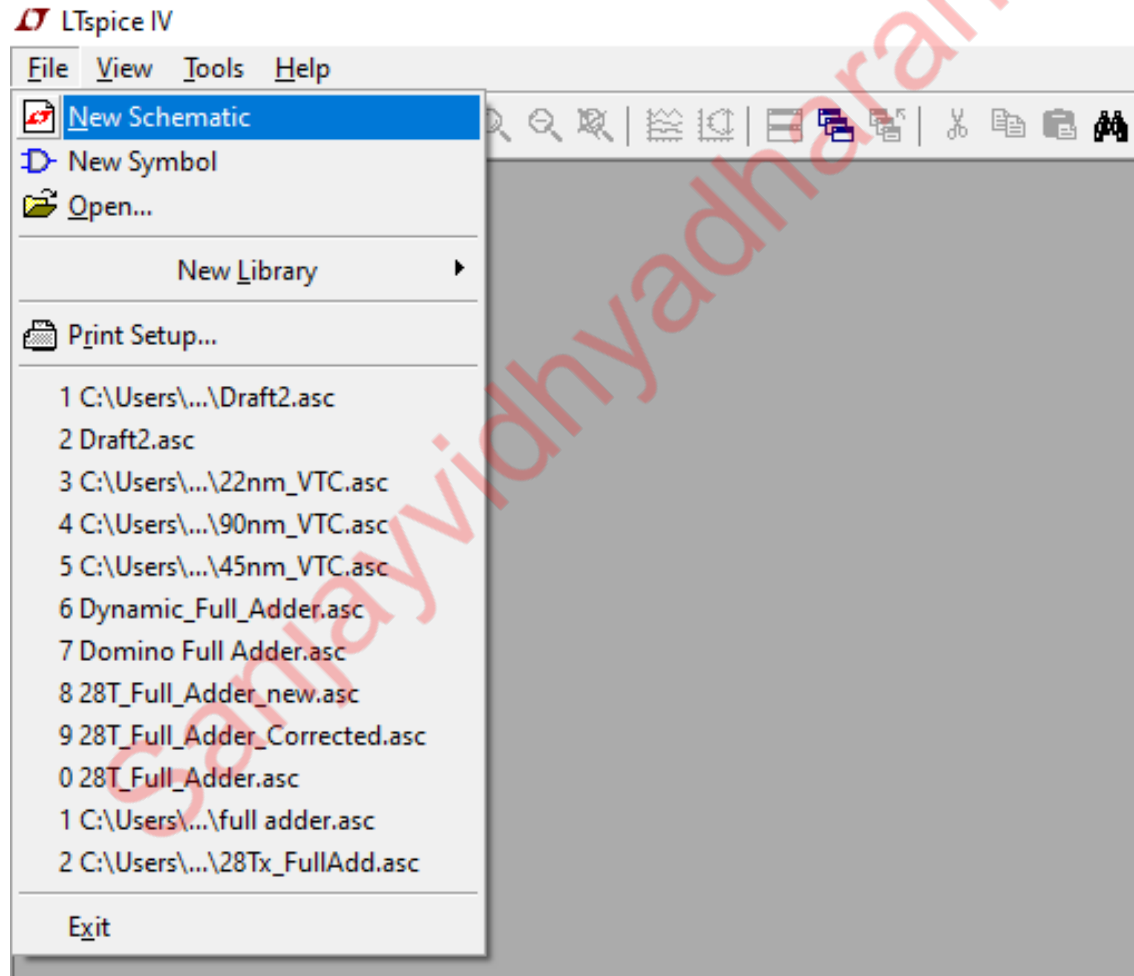
LT SPICE Webpage

<https://sanjayvidhyadharan.in/Downloads>

LT_SPICE_Instaltion_Files.rar and extract it

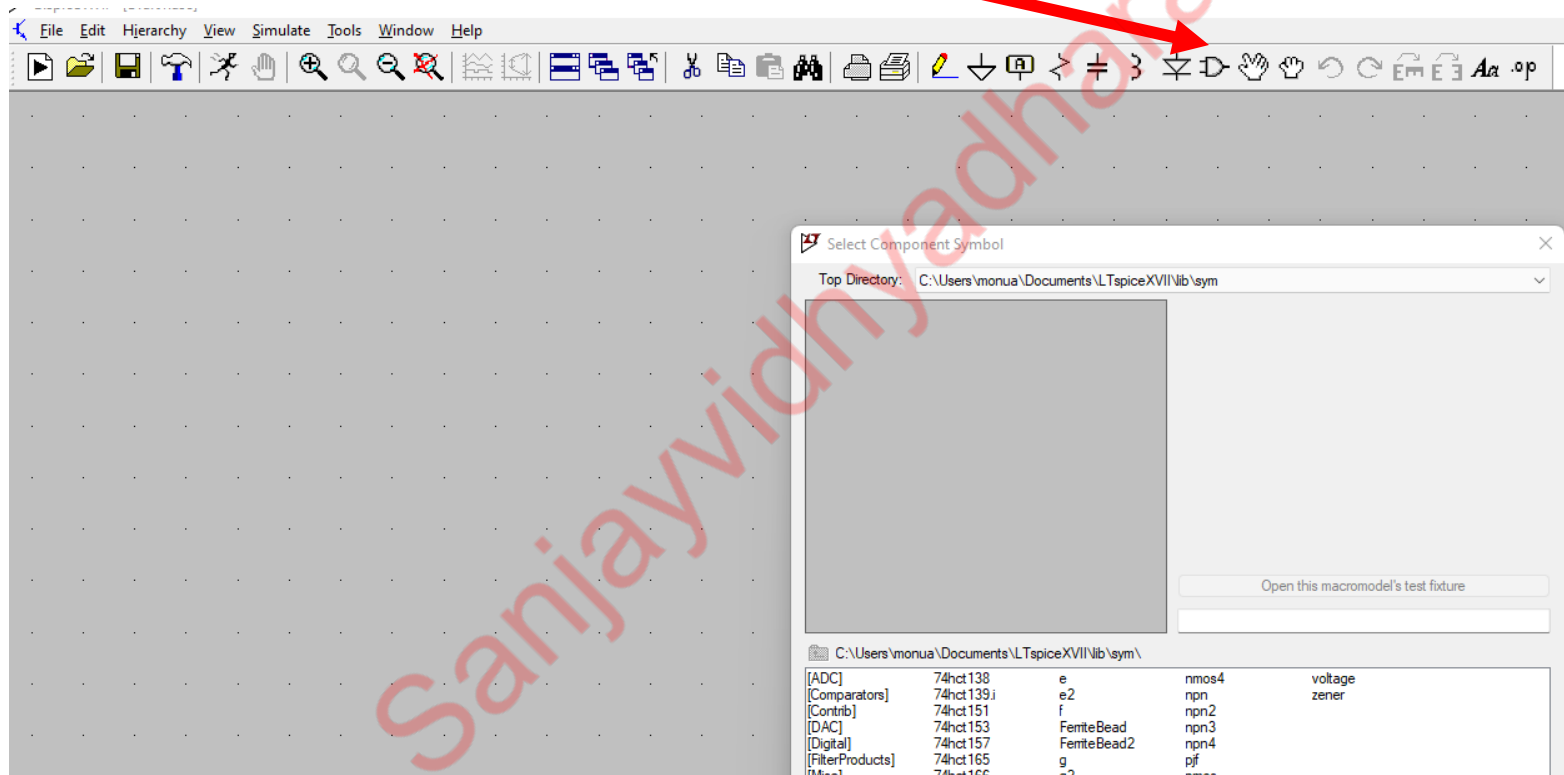
Installing CMOS SPICE Model in LT SPICE

3. Open LTSPICE. And open a New Schematic from File tab.



Installing CMOS SPICE Model in LT SPICE

4. Open LT SPICE, Open New Schematic and Click on the Component tab (looks like a AND gate) on the new schematic window.



5. Note the path of the LT SPICE lib file

Installing CMOS SPICE Model in LT SPICE

6. In the folder C:\Users\Monua\Documents\LTSPICE\lib\sym

Paste

(a) cmosn.asy

(b) cmosp.asy

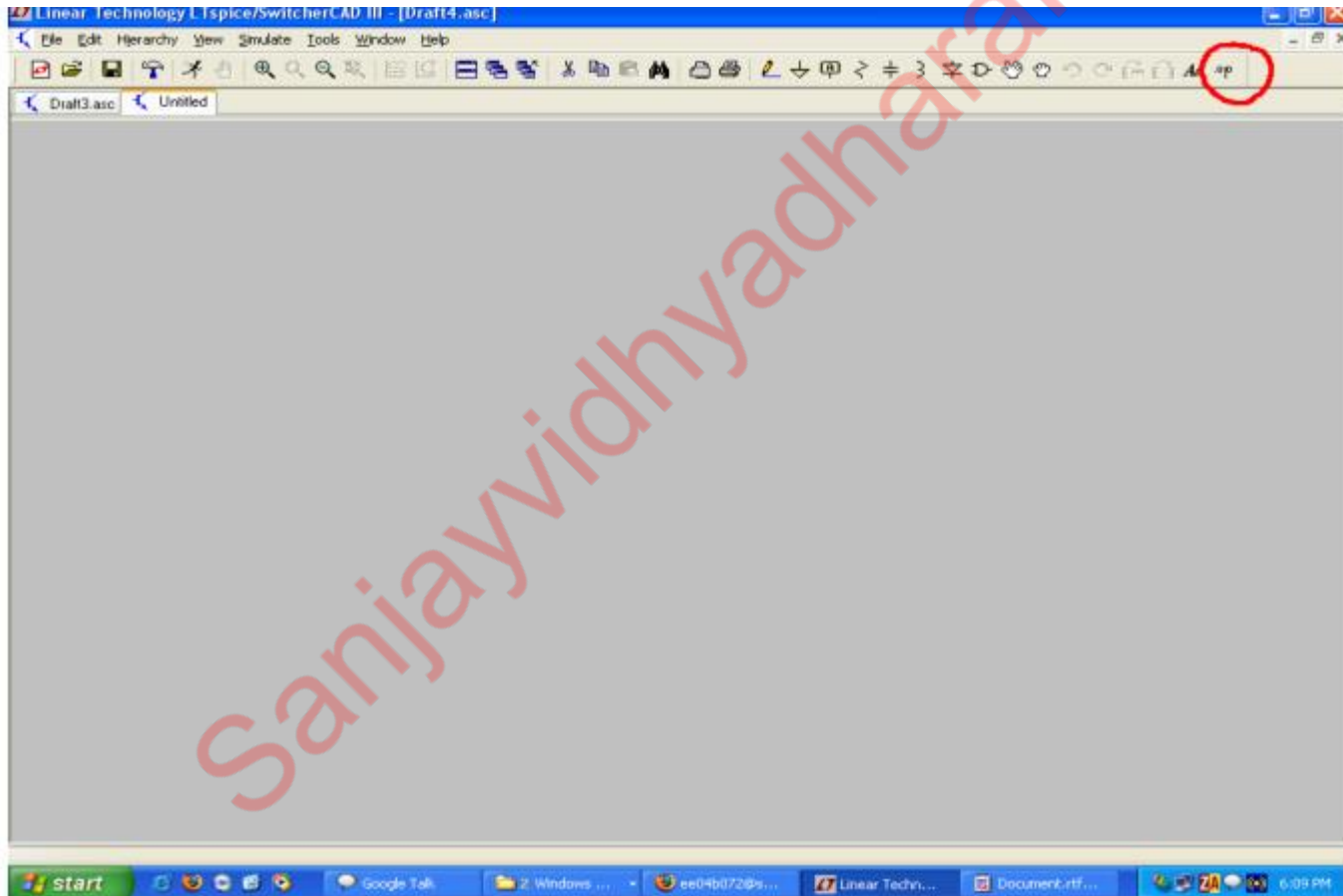
7. In the folder C:\Users\Monua\Documents\LTSPICE\lib\sub

Paste

(c) tsmc018.lib

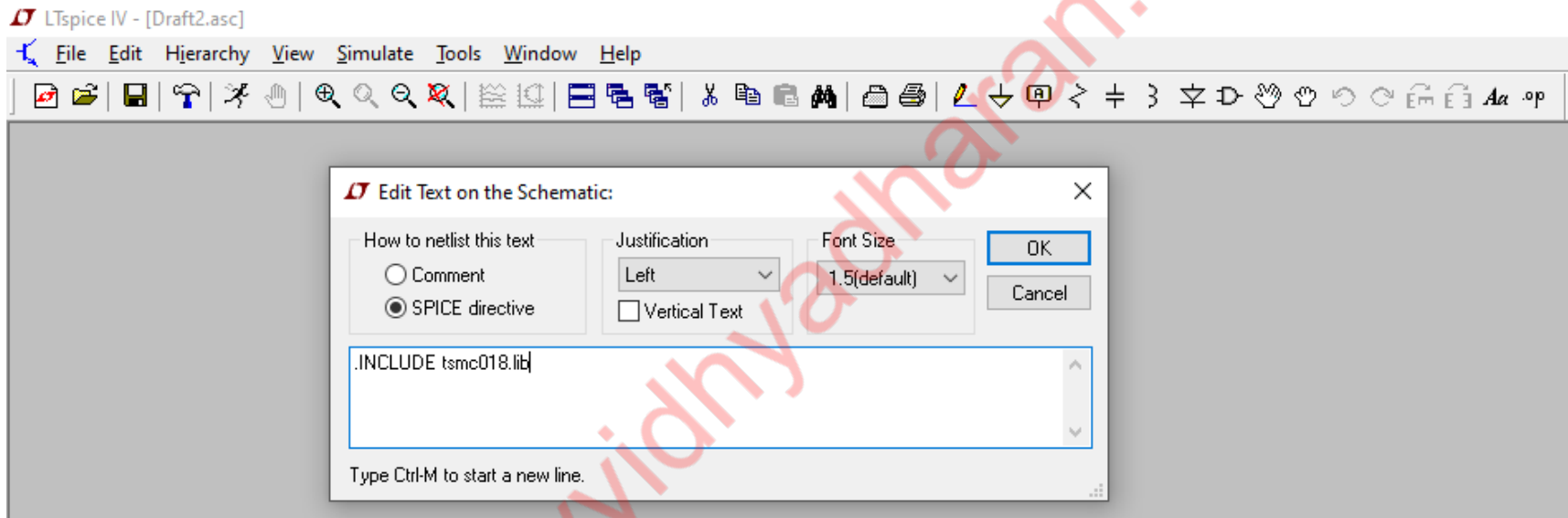
CMOS INVERTER

5. Click on the SPICE Directive tab as shown below:-



CMOS INVERTER

5. Select SPICE Directive and Type .INCLUDE tsmc018.lib



6. You can select **cmosn** or **cmosp** from component menu and set the required **W** and **L**.

Subscribe to <https://sanjayvidhyadharan.in/youtube>

Sanjayvidhyadharan.in