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

🖉 Linear Technology LTspice/SwitcherCAD III - [Draft4.asc]								
🔨 Elle Edit Hjerarchy Yiew Simulate Iools Window Help 🛛 📃 🖉 🗶								
🖻 📽   🖬	🗣 🛪 🕘	<b>. ि</b>	� �   ≌ ! □   Ε	1 🖷 📽   🐰 🛍 🖻	M 🗇 🍜 🖊 🤜	▶ @ < + 3 字	: <b>₽∛00</b> 0€	∺É∃ <b>44</b> .∾P
🔨 Draft3.asc	🔨 Untitled							
🐉 start	😂 🕹 😒	🕑 🧿	Google Talk	🗀 2 Windows 👻	🕹 ee04b072@s	🚺 Linear Techn	Document.rtf	🌯 🛒 📶 📯 💽 6:09 PM

4) Type : .INCLUDE tsmc018.lib

The SPICE directive must be chosen.



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

