Instruction for the LTC-SPICE models to be used in the Mixed Signal Design course

1) Create a work dir for the exercise

2) Copy the following files into the work dir:

CMOSN025.asy
CMOSP025.asy
CMOSN025.lib
CMOSP025.lib
TMSC025.mos

These files are included into the model.zip file. Note that also an "inv.asc" sample schematic is included for testing purposes.

To create a new schematic:

3) Open LTSPICE and create a new schematic (File->New Schematic)

4) Save the schematic into the created work directory

5) Insert components (F2) and choose the work directory between the two options proposed in the dialog window.

6) Edit each component through the dialog window (right click on the component), modifying the field indicated as "Spice Line", changing the values of L, W and m. Take care not to delete the postfix "u" (microns) after the L and W values.