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.

Open Symbol: G:\lavoro\DIDATTIC\corsi\Mixed_Signal\appunt Attribute Value SpiceModel CMDSP025 Value2		x
Attribute Value SpiceModel Value CMOSP025 Value2 SpiceLine L = 0.9u W = 114u m = 1 SpiceLine2	1\2013\L	ez_s
Attribute Value SpiceModel		
Attribute Value SpiceModel Value Value CMOSP025 Value2 Value2 SpiceLine L = 0.9u W = 114u m = 1 SpiceLine2 Value3		
Attribute Value SpiceModel Value Value CM0SP025 Value2 SpiceLine SpiceLine L = 0.9u W = 114u m = 1 SpiceLine2 SpiceLine2		
SpiceModel Value CMOSP025 Value2	Vis.	*
Value CMOSP025 Value2 SpiceLine L = 0.9u W = 114u m = 1 SpiceLine2		-
Value2 SpiceLine L = 0.9u W = 114u m = 1 SpiceLine2	Х	
SpiceLine L = 0.9u W = 114u m = 1 SpiceLine2		Ξ
SpiceLine2		
		-
Cancel		