SPICE MANUAL

1. Open AMS simulator in your mosaic XP.
2. file->new->text file, input the following spice file
3. file->save as input file name ***test.cir*** as follow.

1. Click run botton on your right
2. Trace->add trace, choose V(3)

CMOS LOGIC INVERTER

\*Enter the voltage sources into SPICE. The value of VIN can be arbitrary

\*because it will be incremented over a range of values by the .DC command

VDD 1 0 5V

VIN 2 0 0V

\*Enter the MOSFETs and set the geometrical factors W and L:

\* D G S B

M1 3 2 1 1 PCHAN W=0.75U L=0.5U

M2 3 2 0 0 NCHAN W=0.75U L=0.5U

\*Specify the parameters of the two MOSFETs:

.MODEL PCHAN PMOS (VTO=-0.8 KP=21e-6 LAMBDA=0.03)

.MODEL NCHAN NMOS (VTO=0.7 KP=73e-6 LAMBDA=0.01)

\*Instruct SPICE to plot the voltage transfer characteristic for 0 < VIN < 5V

.DC VIN 0 5V 0.001V

.PRINT DC V(3)

\*Instruct the .PROBE utility to save VOUT during the .DC sweep

.PROBE V(3)

.END