

# COMPARING SIMULATIONS OF THE DIGITAL VLSI-CMOS CIRCUITS USING PSPICE

Laura Kovacs

## **Abstract**

*This paper presents the comparison of some circuits VLSI-CMOS, with the help of the program of simulation PSPICE. It contains a short presentation of MOS circuits including the structure; the way of using the performances analysis and a short presentation of combinational simulated circuits. The real simulation of VLSI-CMOS circuits is presented in the simulation part. These simulations were made possible with the SPICE simulator and Xilinx - Modelsim module. The results of the simulation were obtained by varying the W breadth and the L length of the NMOS transistors channel and then of the PMOS. The last part of the paper contains the conclusions of the transit time from High to Low and Low to High of the simulated circuits.*