Literatura

1 TUINENGA, P.W.: A guide to circuit simulation and analysis using PSpice. Prentice - Hall, Englewood Cliffs.

2 BANZHAF, W.: Computer - aided circuit analysis using SPICE. Prentice - Hall, Englewood Cliffs.

3 ANTOGNETTI, P. - MASSOBRIO, G.: Semiconductor device modeling with SPICE. Mc Graw Hill Book Company, New York 1988.