

PSpice 9.1 Studentenversion. Limits

PSpice Student Version Release 9.1 - Release Notes - February, 2000

PSpice A/D

Simulation is limited up to: **64 nodes**

10 transistors

65 digital primitive devices

10 transmission lines in total

4 pairwise coupled transmission lines.

Additional limits

The sample library includes 39 analog and 134 digital parts.

Device characterization in the PSpice Model Editor is limited to diodes.

Stimulus generation in the PSpice Stimulus Editor is limited to sine waves (analog) and clocks (digital).

Circuit optimization with the PSpice Optimizer is limited to one goal, one parameter, and one constraint.

You can not create CSDF format data files.

You can only display simulation data from simulations performed with the Student Version of the simulator.

Schematics

You can place a **maximum of 50 parts** on a schematic design.

You can only draw on size A sheets.