HSPICE P-2019

Synopsys HSPICE® circuit simulator is the industry's "gold standard" for accurate circuit simulation and offers foundry-certified MOS device models with state-of-the-art simulation and analysis algorithms. With over 25 years of successful design tape outs, HSPICE is the industry's most trusted and comprehensive circuit simulator.

- For on-chip simulation: Analog designs, RF design, custom digital design, standard cell design and characterization, memory design and characterization, and device model development.

- For off-chip signal integrity simulation: Silicon-to-package-to-board-to-backplane analysis and simulation

Key Features:
- Used by engineers, professors and students of Electronics and Electrical
- It is possible to simulate the operation of circuit devices
- Has a simple user interface
- Output template support for generic MOSFET wrapper
- Draw a variety of charts to show the output of the parts
- Supports AC Noise output
- Includes complete library of electronic components
- Checking how parts work in alternating or direct current circuits
- And more