Dr. Santanu K. Mishra
Professor
Department of Electrical Engineering
Indian Institute of Technology Kanpur
Kanpur, U.P. 208016, INDIA
Phone No: +91-0512-259-6249
Fax: +91-0512-259-0063
Email: santanum@iitk.ac.in

Enquire No.: EE/SKM/008/2018 (revised)

Kanpur
17/02/2018

Starting date: 22/02/2018
Ending Date: 22/04/2018

Sir/Madam,

Sub: OrCAD PSPICE Simulation Software License

We are inviting quotation for **Ten** licenses for OrCAD PSPICE circuit simulation software. The software will be used in a lab environment to carry out various analog/digital circuit designs and simulations.

The software should have following features:

- Able to perform complex mixed signal designs simulations, containing both analog and digital parts to support models like IGBTs, pulse width modulators, DACs and ADCs
- Contains 40,000 or more schematic symbols
- Circuit analysis features preferably include DC sweep, AC sweep, transient analysis, frequency response analysis, mixed analog and digital design analysis with accuracy,
- Circuit design features include hierarchical, flat and hierarchical design, dynamic update of hierarchical blocks, design reuse, partial design simulation, unlimited user-defined properties, centralized part information system, component property validation, intelligent PDF creation, able to import Altium schematic design
- simulation features preferably include analog behavior modelling, Auto-convergence control options, magnetic part editor and model editor for device characterization
- Results and data display features should include view of multiple plots, able to apply mathematical operations such as Fourier transform and derivatives for small signal characteristics on simulation output variables

Dr. Santanu K. Mishra
Professor
Department of Electrical Engineering
Indian Institute of Technology Kanpur
Kanpur, U.P. 208016, INDIA
Phone No: +91-0512-259-6249
Fax: +91-0512-259-0063
Email: santanum@iitk.ac.in

- Desirable advanced analysis: Monte-Carlo analysis, temperature and stress analysis, worst-case analysis, parametric sweep analysis and sensitivity analysis
- Preferable with integrated PCB designer application
- PCB designer application should include schematic capture, librarian tools, PCB editing and routing, Constraint Manager, signal integrity, auto routing and 3D visualization
- Integration with MATLAB Simulink is required for electromechanical circuit simulation
- FPGA design-in support
- Compatible and able to open simulation files of existing ORCAD PSPICE software
- Able to run on operating systems windows 7 and windows 10
- License type: Network Address type

We request to kindly send in your quotes and specs by **22/04/2018**. Please send the technical and financial specifications in sealed envelopes separately.

Thanks,

Santanu Mishra
Department of Electrical Engineering
Indian Institute of Technology Kanpur