

# PADS SPICEnet

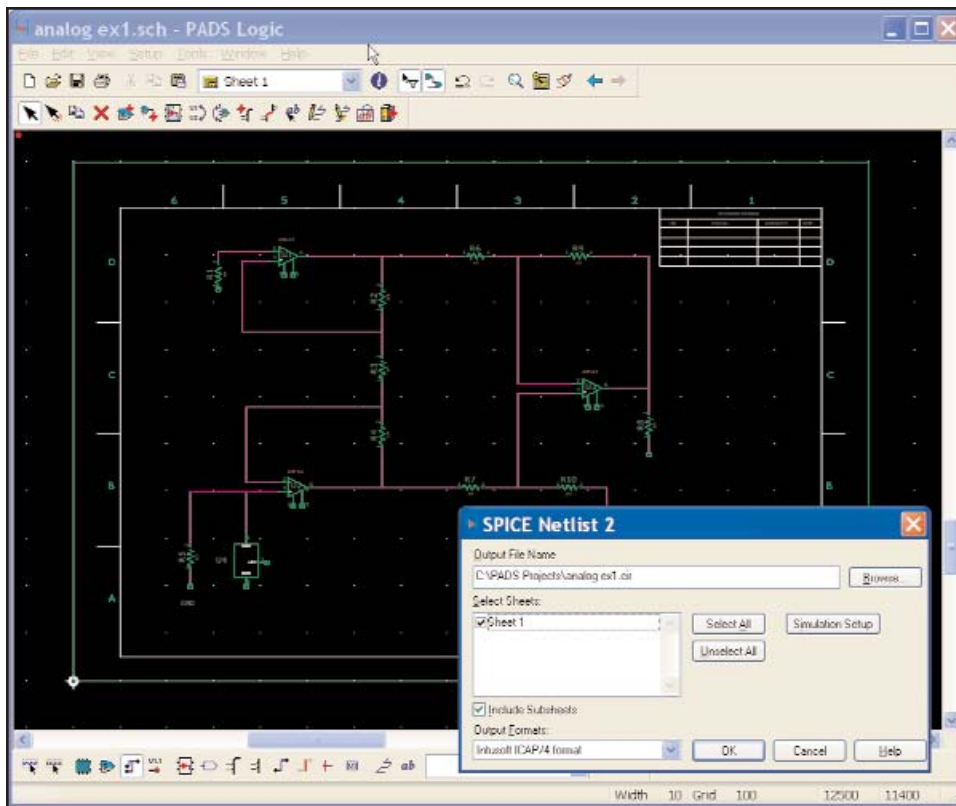


Figure 1: With PADS SPICEnet set is a snap, choose your sheets and select Simulation Setup.

## Netlist to standard SPICE programs from PADS Logic

Simulation is the key to creating quality products and PADS® SPICEnet makes that connection for you from PADS Logic. You can use SPICEnet to create netlists for many standard SPICE programs including ICAP from Intusoft, PSpice from Cadence and standard Berkeley SPICE. The netlists can be analog only or include mixed analog/digital information. You can netlist your complete design or a portion of your design and you can easily add model information to any PADS Logic component. PADS SPICEnet also allows you to place more than 80 different simulation attributes on symbols in the schematic.

### Major product features

- User-friendly simulation connection for PADS Logic
- No need to place special simulation symbols on your schematic because simulation setup can be organized with dialog boxes
- Easily add simulation properties to PADS Logic parts
- More than 80 simulation properties supported

It's easy to set up your simulation with the simulation wizard. It will guide you through the basic steps for AC, DC or Transient Analysis. You can also choose whether to output the operating point (see figure 2).

### DC Analysis

In the dialog box for DC analysis setup, simply type in the values for your source, starting and ending voltages and the voltage step you want to use. With the DC Analysis setup window you quickly set up the optimal parameters for your DC sweep to ensure efficient simulation (See figure 3).

### AC Analysis

To set up your AC analysis just choose decade, octave or linear analysis. Then fill in the starting and ending frequencies. The AC analysis in SPICE computes the AC output variables as a function of frequency. The program first computes the DC operating point of the circuit and determines linearized, small-signal models for all of the nonlinear devices in the circuit. The resultant linear circuit is then analyzed over your user-specified range of frequencies providing a transfer function (See figure 4).

### Transient Analysis

For a transient analysis you simply fill in the interval you have chosen for the time step, and the total analysis time. You can also choose to start recording data at a point later in the analysis to minimize the amount of data recorded. And with PADS SPICenet you can set the maximum time step in order to ensure useful results. Transient analysis allows you to find the response time of your circuit by computing the transient output variables as a function of time over a user-specified time interval (See figure 5).

### Summary

With PADS SPICenet you can view the netlist prior to using in a simulation. If necessary you can make edits necessary for an optimal simulation. PADS SPICenet provides a user-friendly connection between PADS Logic and industry standard SPICE and mixed analog/digital simulators to enhance your design flow.

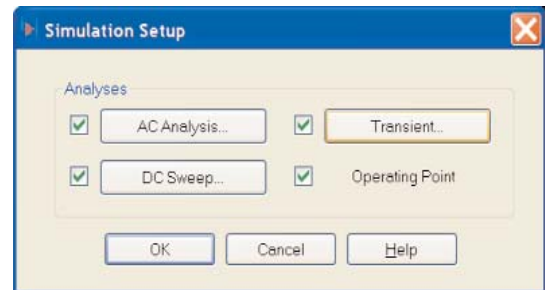
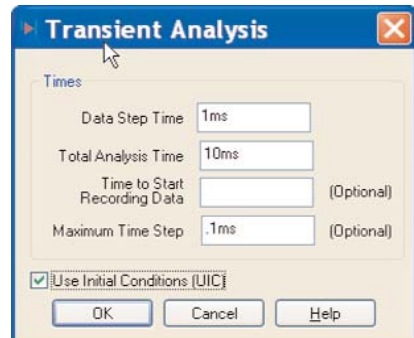
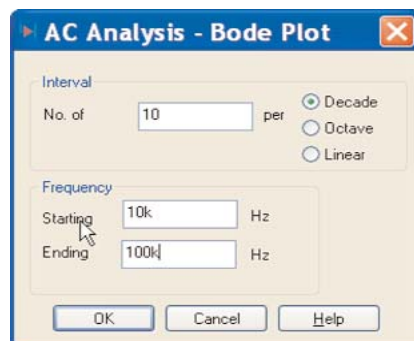
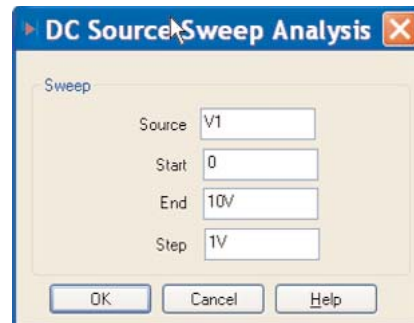


Figure 2: The simulation wizard guides you through setup for your simulation.



Figures 3, 4, and 5: Just fill in the values to organize your simulation.

Visit our website at [www.mentor.com/pads](http://www.mentor.com/pads)

Copyright © 2004 Mentor Graphics Corporation. All rights reserved. Mentor Graphics, PADS, and PowerPCB are registered trademarks and PADS Layout and PADS Logic are trademarks of Mentor Graphics Corporation. All other trademarks mentioned in this document are trademarks of their respective owners.

Corporate Headquarters  
Mentor Graphics Corporation  
8005 SW Boeckman Road  
Wilsonville, OR 97070-7777  
Phone: 503.685.7000  
Fax: 503.685.1204

Systems Design Division  
Mentor Graphics Corporation  
1811 Pike Road  
Longmont, CO 80501  
Phone: 720.494.1000  
Sales: 888.482.3322  
Email: [pads\\_info@mentor.com](mailto:pads_info@mentor.com)