

Easy-Spice

Easy-Spice A/D Mixed mode Simulator For Easy-PC For Windows

Destined to be an all time winning product from Number One Systems, Easy-Spice makes its world debut.

Once again, Number One Systems delivers the kind of astounding performance normally only available in products priced in the £thousands.

Packed with the level of world-beating price/performance our customers have become accustomed to from Number One Systems, we are delighted to announce this sparkling new addition to the Easy-PC toolset.

Based on a high performance SPICE engine, the Easy-Spice simulator is capable of analogue, digital and mixed mode designs.

Easy-Spice is a substantially enhanced version of SPICE 3 and XSPICE. The underlying algorithms have been reworked to improve convergence and add new functionality. In the case of convergence, Easy-Spice simulates 98% of a suite of industry standard benchmarks compared with 60% for unmodified SPICE 3. This has been achieved with proprietary enhancements to the transient analysis algorithms and the development of automatic pseudo transient analysis.

Mixed-mode simulation with superior performance

The release of Easy-Spice marks a watershed in the history of software tools for the Electronics Engineers. Suddenly, a power-packed industry standard mixed-mode SPICE simulator is available priced within reach of almost every company and private individual.

Like all Number One Systems products, Easy-Spice has been developed to be both simple to understand and a pleasure to use. With Easy-Spice we supply a comprehensive product

manual running to almost 400 pages which includes a tutorial with a step by step guide to get you up and running within a few hours.

What makes Easy-Spice so powerful?

Easy-Spice is a substantially enhanced version of SPICE 3 and XSPICE. The underlying algorithms being reworked to improve convergence and add new functionality. This has been achieved with proprietary enhancements to the transient analysis algorithms and the development of automatic pseudo transient analysis.

Easy-Spice simulates 98% of a suite of industry standard benchmarks compared with only 60% for unmodified SPICE3.

Integrates with your Easy-PC Schematics

Easy-Spice is designed to augment and enhance your investment in Easy-PC For Windows. We have built Easy-Spice around your existing Easy-PC Schematics to present a cohesive, integrated design environment.

So, on completion of the simulation stage, if your components have PCB footprints associated, you can translate the circuit to create the PCB layout, just as you do right now. No redesign of the Schematic as with so many SPICE simulators.

Comprehensive 400 Page Users Guide

A 400 page Users Guide has been produced with full index. A complete reference for using the Schematic editor interface, simulator and model definitions.

System Requirements

- Pentium class PC
- Windows 95/98/ME/NT4.0/2000
- 32MB of RAM (Minimum)
- 55MB hard disk space for program files *
- CD-ROM drive

This specification includes full support for Windows 2000 and Windows ME.

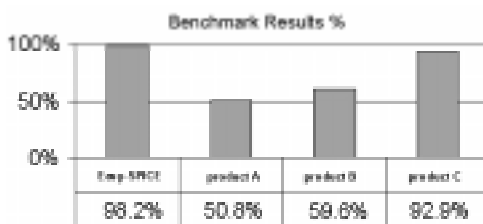
* **Easy-Spice** requires Easy-PC For Windows to also be installed. This figure is the combined requirement (NB Easy-Spice requires an additional 30Mb from Easy-PC).

Easy-Spice demonstrates superior convergence

Easy-Spice simulated 57 publicly available benchmark circuits as part of its performance testing where it presented a staggering 98% pass rate.

In comparison, we then tested 3 household name SPICE simulators, all of which are more expensive than Easy-Spice, on exactly the same set of benchmarks. The above table shows the percentage pass-rate.

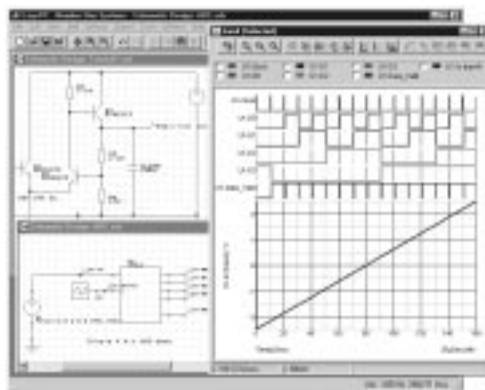
We would add, Product 'C' from a respected supplier, is regarded as the industry standard SPICE simulator. This well-known product failed 4 of the benchmarks thereby only reaching a 93% pass-rate, and is marketed at almost 10 times the price of Easy-Spice! We hope you agree, Easy-Spice really is a landmark product with the potent mix of excellent performance and realistic price.



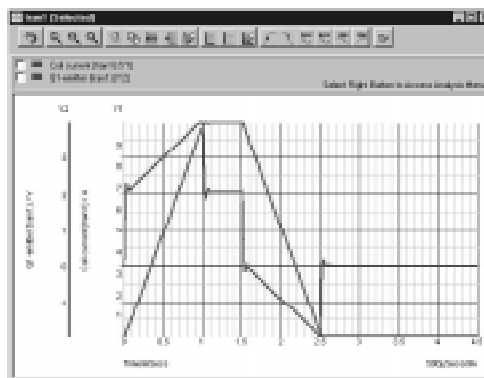
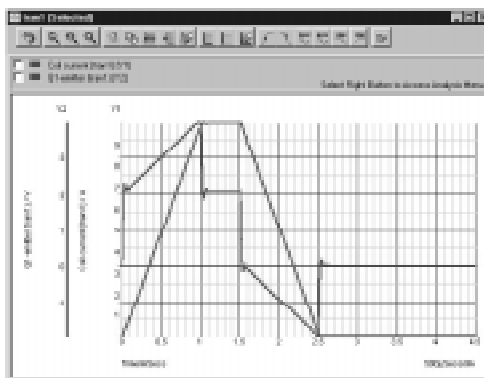
Benchmark results prove that Easy-Spice is a performant product with a very high convergence rate.

Feature Summary:

- Integrated into the Easy-PC For Windows design environment and includes Easy-PC Spice components/symbols
- Convergence performance in benchmark trials exceeds results from other vendors
- Simulates 98% of industry standard benchmarks
- True mixed-mode simulation: closely coupled direct matrix SPICE3 analogue and event driven digital simulator
- Database of over 25,000 devices, plus can import standard SPICE models from outside suppliers - many available on the Internet
- Analyses: operating point, DC sweep, transient, AC small signal, transfer function, sensitivity, pole-zero
- Comprehensive advanced waveform analysis viewer with multiple grids and axes, cursor measurements and zoom trackback



Easy-Spice is an integrated simulator environment driven by an intuitive user interface.



Easy-Spice provides a rich set of features to help you analyse your circuit. Graphs of circuit voltages, currents, and device powers are produced by attaching probes to the relevant points in the Schematic. The graph is created and incrementally updated during the simulation process. At a click of the mouse the RMS, rise and fall time, -3db point, overshoot or many other functions can be calculated and displayed alongside the graph legend.

Number One Systems
Oak Lane
Bredon
Tewkesbury
Glos
GL20 7LR

Phone: 01684 773662
Fax: 01684 773664
Email: info@numberone.com
Web: www.numberone.com