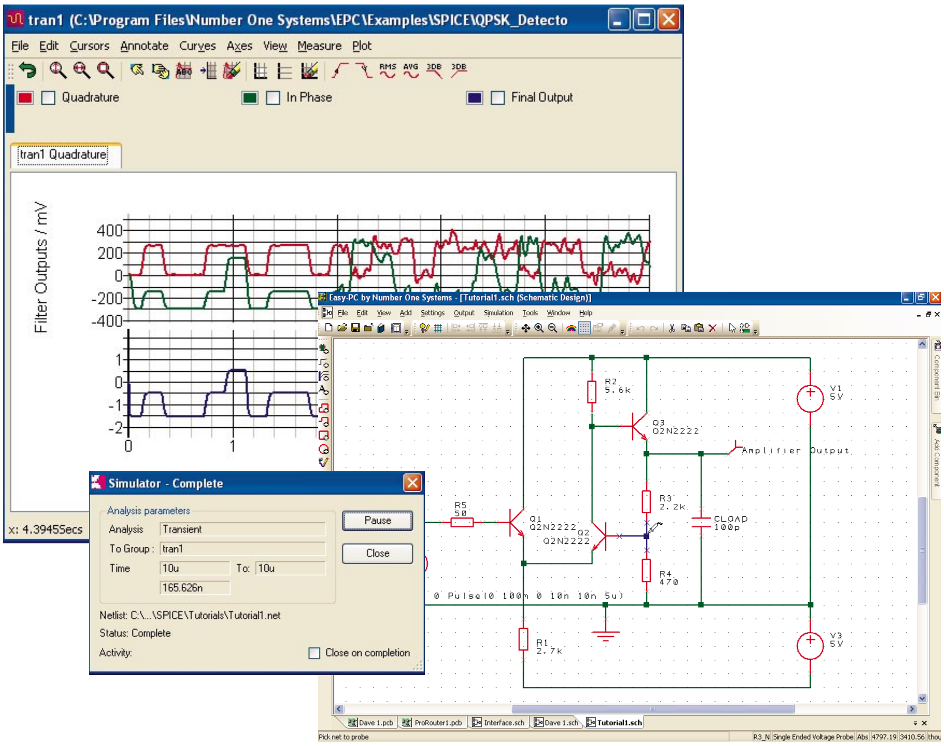
Data Sheet

Number One Systems

Easy-Spice



Mixed-mode simulation with superior performance

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

Destined to be an all time winning product from Number One Systems, the Easy-Spice A/D mixed mode simulator provides engineers with a professional level simulation unseen at products in this price range. This kind of astounding performance is 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, this addition to the Easy-PC toolset enhances an already outstanding product.

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 100% 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.

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.

Integrates with your Easy-PC Schematics

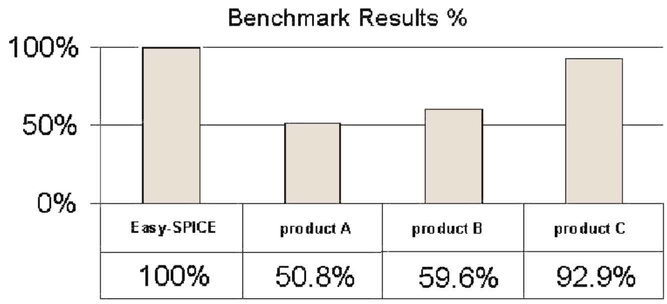
Easy-Spice is designed to augment and enhance your investment in Easy-PC For Windows. We have built Easy-Spice around your existing Easy-PC Schematics to present a cohesive, integrated design environment.

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.

Easy-Spice



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

Easy-Spice demonstrates superior convergence

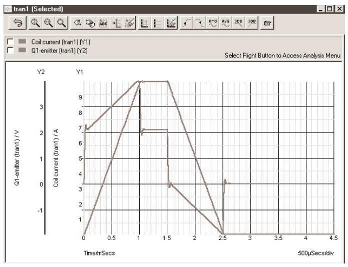
Easy-Spice simulated 57 publicly available benchmark circuits as part of its performance testing where it presented a staggering 100% pass rate*.

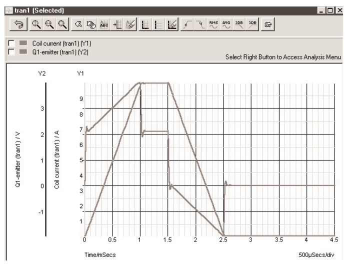
In comparison, we then tested 3 household name SPICE simulators, all of which are more expensive than Easy-Spice, on exactly the same set of benchmarks. The above table shows the percentage pass-rate.

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.

Feature Summary:

- Integrated into the Easy-PC For Windows design environment
- True mixed-mode simulation: closely coupled direct matrix SPICE3 analogue and event driven digital simulator
- Includes Easy-PC Spice components/symbols
- Simulates 100% of industry standard benchmarks*
- Convergence performance in benchmark trials exceeds results from other vendors
- Database of over 25,000 device names and 15,000 models
- Imports 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
- Includes Randon probing and Bias annotation markers
- Comprehensive 400+ page Users Guide supplied
- Requires any Easy-PC For Windows variant to run





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, over-shoot or many other functions can be calculated and displayed alongside the graph legend.

* *Easy-Spice version 2.0 used for product comparisons.*

For information on pricing contact:
Number One Systems. Oak Lane, Bredon,
Tewkesbury, Glos. GL20 7LR. UK

Tel: +44 (0)1684 773662
Fax: +44 (0)1684 773664
e-mail: sales@numberone.com

Further information on all products and prices available at:
www.numberone.com