

# Intusoft Newsletter

Personal Computer Circuit Design Tools

January 1991 Issue



Copyright © Intusoft, All Rights Reserved  
(213) 833-0710 Fax (213) 833-9658

## INTUSCOPE 3.1 PCU PDATE, NOW AVAILABLE

The long awaited INTUSCOPE update for the PC is now available. The new INTUSCOPE program is a view into the future. It uses a custom windowing interface that makes data analysis and program operation truly easy and intuitive. With pull down menus to access all of INTUSCOPE's functions, a simple mouse interface, and on-line help with examples, learning the new INTUSCOPE program will be fast, and using the program will be a pleasure.

For those unfamiliar with Intusoft's design tools, INTUSCOPE is an analysis program especially designed to display, analyze, and print out data from SPICE output files, including all versions of IsSPICE. The power of the original INTUSCOPE has been tremendously enhanced. Multiple resizable windows allow output data from different analyses, SPICE runs, and user generated files to

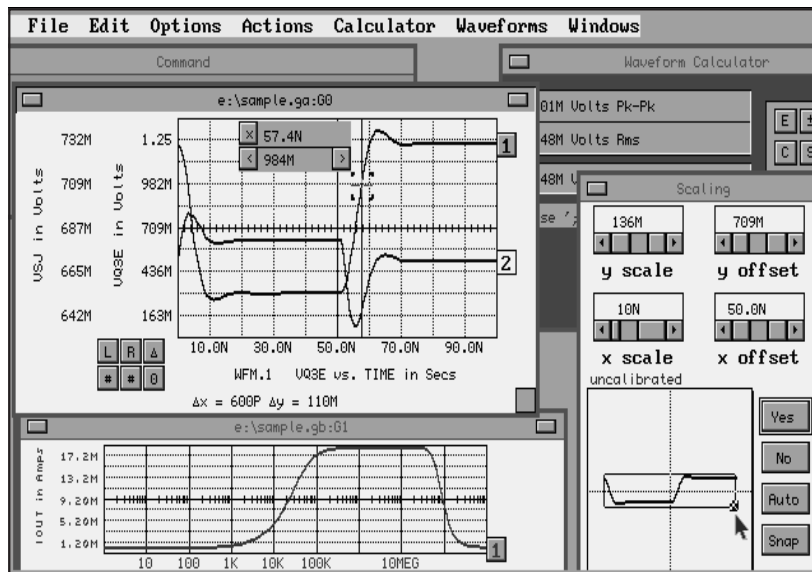
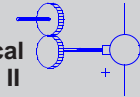
### In This Issue

1 INTUSCOPE PC Update

3 Why Windows 3.0 and SPICE Don't Mix

4 Analog Behavioral Modeling With SPICE

7 Modeling Mechanical Systems, II

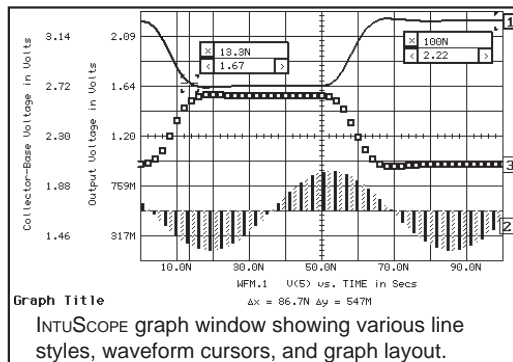
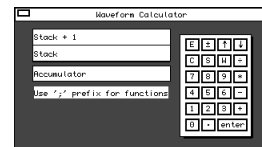
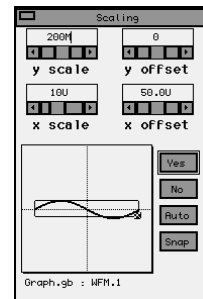
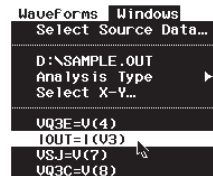


## INTUSCOPE 3.1 PC Update *(cont'd)*

be displayed and compared along side each other. All of the limitations and difficulties are gone including those associated with waveform manipulation, number of data points, function selection, and printing. A completely re-written 200+ page User's Guide is included with the update. The guide contains examples for almost every INTUSCOPE function. Also included will be a new installation and tutorial guide with several integrated examples on how to use Intusoft software and the new INTUSCOPE program.

INTUSCOPE version 3.1 contains many major feature enhancements such as:

- New, easy to use, graphical interface with pull-down menus and multiple resizable graph windows. Graph windows have scaling directly on the graph.
- Ability to display and compare data from several output files and several analysis types (AC, DC, Transient, Noise) on the same graph.
- New Print Preview mode providing report quality graphics output to Laser (H-P LaserJet) and dot matrix printers, and plotters. Output file formats include TIFF, PIC, HPGL, and EPS (Postscript).
- Special Scaling Window with waveform zoom feature allowing arbitrary waveform scaling and easy configuration of waveform displays.
- Support for waveforms with large numbers of data points; only limited by available memory. Waveforms with over 10,000 points can be analyzed.
- Advanced Waveform Calculator and pull down menus providing access to all of INTUSCOPE's mathematical operations.
- An on-line help system providing help on all menu functions. Numerous examples are also included along with complete documentation.



INTUSCOPE 3.1 for the PC is totally integrated with Intusoft's other circuit design tools. The update will work with all the current versions of SPICENET, PRESPICE, and SPICEMOD. No other updates are required.

## Why Windows 3.0 And SPICE Don't Mix

Intusoft's ICAPS, SPICENET (schematic entry) and INTUSCOPE (waveform processor) programs complement the power of the IsSPICE program with an easy-to-use graphical interface. A graphical interface greatly alleviates the netlist headaches of other SPICE programs that only utilize a text interface. So its no surprise that Windows 3.0, one of the best graphic interfaces for DOS, would be an ideal place to mate with the Intusoft product line. Unfortunately, this is not the case.

Circuit simulation is an iterative process progressing from schematic entry to simulation to waveform display and then back to the schematic; as much as 10 to 20 times an hour for medium size circuits. If productivity is to be maintained, the transition between programs, as well as the speed of the simulation must be fast. As it turns out, Windows is prohibitively slow in both these areas.

Starting Windows and applications under Windows can take as long 30 seconds. This delay becomes quite bothersome unless all the circuit design modules are kept running; a process requiring megabytes of memory. And if memory is in short supply the responsiveness may become worse as Windows uses the hard disk for program swapping.

The advantages of high speed simulation and space for large circuits (greater than 200 components) offered by IsSPICE/386 are not currently possible with Windows. The only way to access large amounts of RAM is to use protected mode. Windows 3.0 only supports the 80286 (16-bit) protected mode. Real and 386 mode (Virtual 86) only allows single or multiple 640K sessions with access to slow EMS memory.

*continued on page 20-10*

## Analog Behavioral Modeling With SPICE

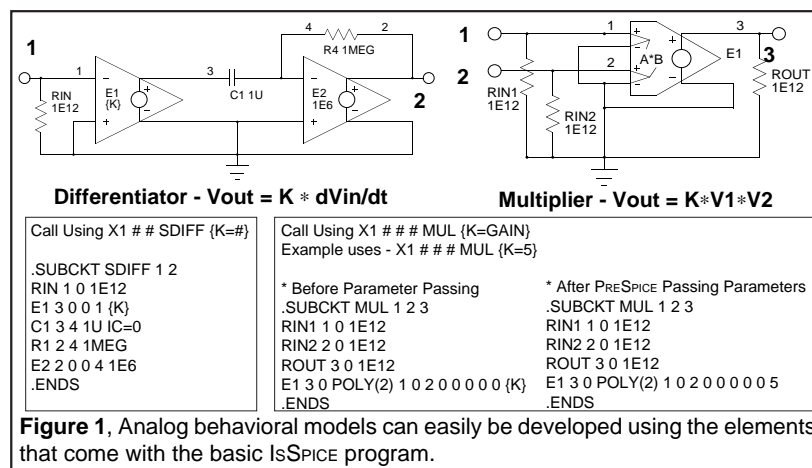
In the October '90 newsletter we initiated a study on how analog behavioral models (ABMs) can be made and used with any SPICE program. In this part, we will introduce some other basic models and show how the INTUSCOPE program's polynomial regression feature can be used in the creation of various ABMs. Then we will use the ABMs and IsSPICE to solve a set of 2nd order nonlinear differential equations describing a physical system.

Integrator and summer models were developed in the October issue. The ABMs for two other important elements, the differentiator and multiplier, are quite similar and shown in Figure 1. Notice that the summer and multiplier use the same topology, only with different polynomial coefficients.

The IsSPICE capacitor, inductor, and dependent voltage/current source elements have the ability to use polynomial dependencies. The IsSPICE polynomial features are thoroughly described in the "SIMULATING WITH SPICE" book. The dependent source polynomial feature is especially important to our ABM discussions as it can be used to form a variety of input/output relationships.

Although best used for transfer functions that are asymptotic to zero with zero input and infinity with infinite input, many other ABMs can be described by dependent source polynomials if properly bounded. An example of such a model is the trigonometric function, Sin. INTUSCOPE's X-Y plotting and polynomial regression features are perfectly suited to ABM development.

Creation of a single input-single output transfer function starts with getting the desired input and output waveforms into the INTUSCOPE program. This can be done by using IsSPICE output or



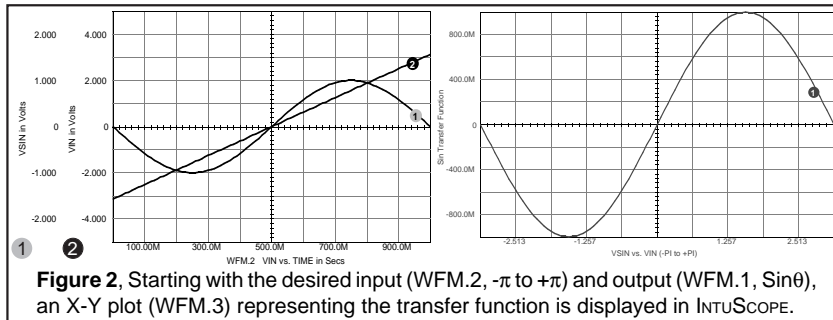


Figure 2, Starting with the desired input (WFM.2,  $-\pi$  to  $+\pi$ ) and output (WFM.1,  $\text{Sin}\theta$ ), an X-Y plot (WFM.3) representing the transfer function is displayed in INTUSCOPE.

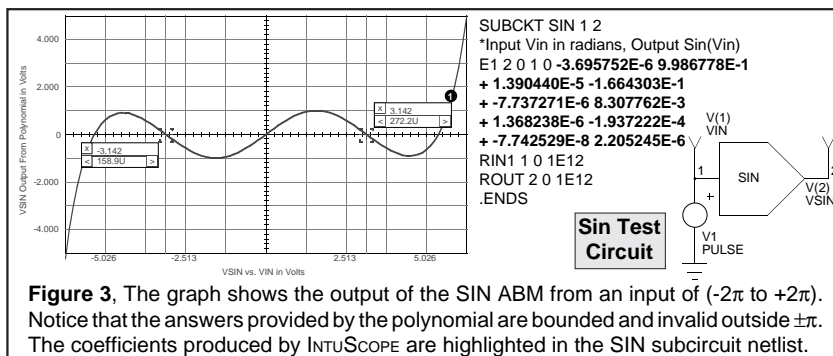


Figure 3, The graph shows the output of the SIN ABM from an input of  $(-\pi$  to  $+\pi)$ . Notice that the answers provided by the polynomial are bounded and invalid outside  $\pm\pi$ . The coefficients produced by INTUSCOPE are highlighted in the SIN subcircuit netlist.

by developing an ASCII data file; INTUSCOPE will except either. An X-Y plot of the input-output waveforms is then displayed. Next the polynomial order is selected based on the desired accuracy of the final transfer function and the polynomial regression feature of INTUSCOPE is performed. The resulting coefficients, ready to be placed in an ISPICE statement, are displayed in the Output Record window. Figures 2 and 3 illustrate the process. Use of the Sin block is not without some cautions. Noting the graph of Figure 3, one must be careful that the input to the block remain within the  $-\pi$  to  $+\pi$  bounds, otherwise, the DC or transient simulation may not converge.

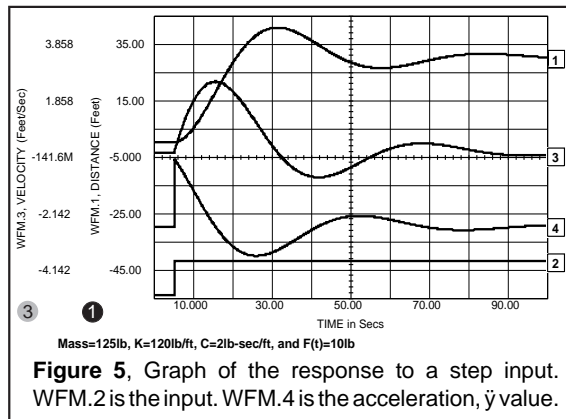
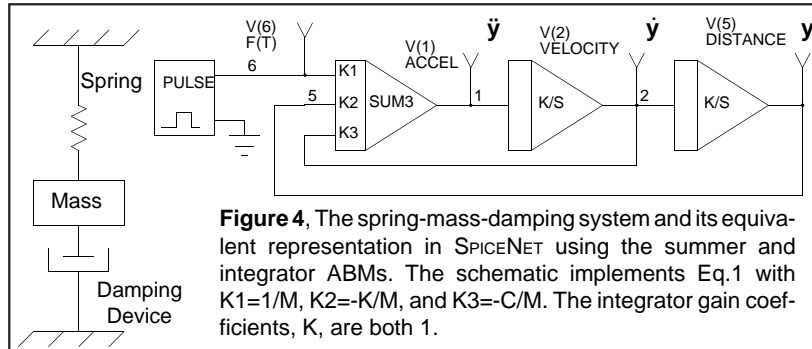
The traditional approach to solving many physical/mechanical engineering problems is to utilize electrical analogs and create a circuit representation. The problem then becomes one of solving a set of N differential equations with N unknowns; a process to which ISPICE is ideally suited. If, however, the problem can be specifically expressed as a series of differential equations, the ABMs we have described can be used to provide the solution.

A simple spring-mass-damping system is represented by the second order differential equation:

$$M\ddot{y} + C\dot{y} + Ky = F(t) \quad \text{or} \quad \ddot{y} = 1/M (F(t) - C\dot{y} - Ky) \quad \text{Eq. 1}$$

where M is the mass, C is the viscous-damping coefficient, K is the spring constant, y is the displacement, and F(t) is the applied disturbing force.

Figure 4 shows the system and the equivalent SPICENET schematic describing the above equation. SPICENET alleviates all of the netlist development headaches, allowing efforts to be centered on studying the problem rather than debugging complex SPICE syntax. All of the Intusoft ABMs come with equivalent SPICENET symbols. Figure 5 shows the response of the system to an input step function.



With ABMs, IsSPICE can simulate any system that can be described using a set of nonlinear differential equations. This includes electrical, mechanical, and physics related problems. Other applications in the mechanical field include nonlinear vibration, heat transfer, and fluid flow. For most of its life, SPICE has

been used primarily for the study of electrical problems. This will not be true in the years to come. Many other disciplines will discover the power of IsSPICE. The only obstacles are to develop the schematic symbols and their underlying electrical analogs much as was done in this application note. In the next application note we will explore how Intusoft's circuit design tools and ABMs can be extended to study mixed mechanical/electrical problems.

[1] Thaler, George J. and Brown Robert G., "Analysis and Design of Feedback Control Systems", McGraw-Hill, 1960

[2] Jackson, Albert S., "Analog Computation", McGraw-Hill, 1960

[3,4] Russo, Edwin P. and Trahan, Russell E., "Using Modern Electric Circuit Analysis Computer Programs To Solve Nonlinear Vibration Problems", and "Using Modern Electric Circuit Analysis Computer Programs To Solve Mechanical Engineering Problems", University of New Orleans

## Behavioral Models for Mixed Mode Rotational Systems

(continued from Oct. 1990)

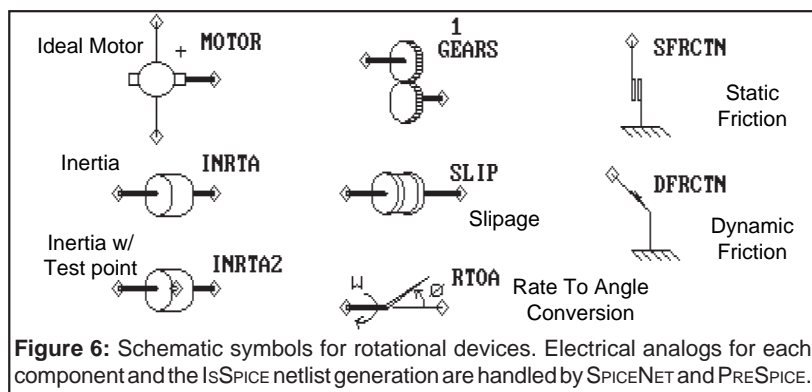
In our last newsletter, we discussed the concept of behavioral models and their implementation using `IsSPICE`. Unlike other simulators that use a programming language or mathematical expressions to develop these models; `IsSPICE` uses subcircuit representations. This approach works nicely for electrical engineers since they are trained in this "language". `IsSPICE`, however, solves a matrix of nonlinear equations that can describe many other physical and biological systems.

By using our `SPICENET` schematic package, we can alter the electrical representations, making the schematic objects represent components such as gear trains, inertia, and friction. This process was illustrated in the previous application note on analog behavioral models.

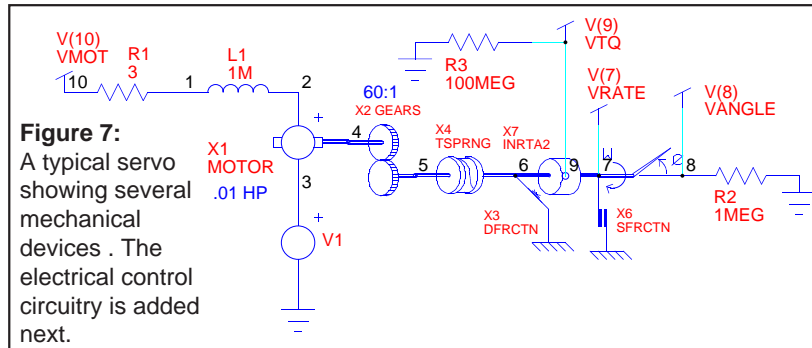
The equations that underlie these objects are described by their block diagram representation in other subcircuit schematics; connected to the main schematic through the `IsSPICE` subcircuit netlist hierarchy. From the engineer's point of view, familiar mechanical components are manipulated while their underlying equations are composed and calculated out of sight by `SPICENET`, `PRESPICE`, and `IsSPICE`.

### Rotational Device Models

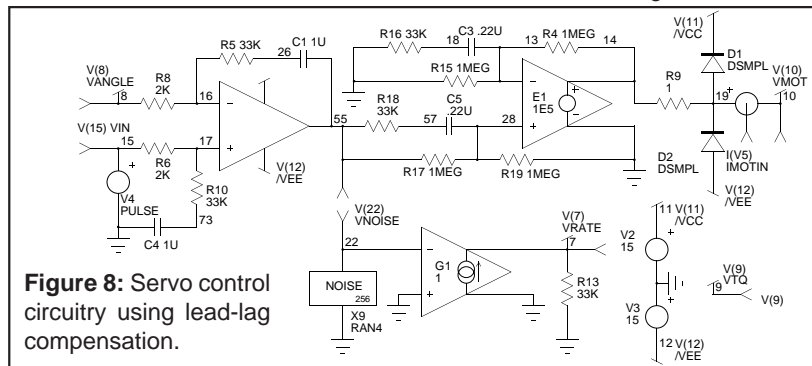
In October, we had completed the model descriptions for the rotational components and had connected them to form a simple positional feedback servo. The library of mechanical components is shown in Figure 6, and the servo schematic is repeated in Figure 7. In this issue we will add rate feedback and some op-amp control components to illustrate the mixed mode simulation (electrical and mechanical in this case).



## Simulating a Typical Servo



Control circuitry needs to be added to improve the response characteristics as shown in the October newsletter. The schematic for the servo control circuit is shown in Figure 8.



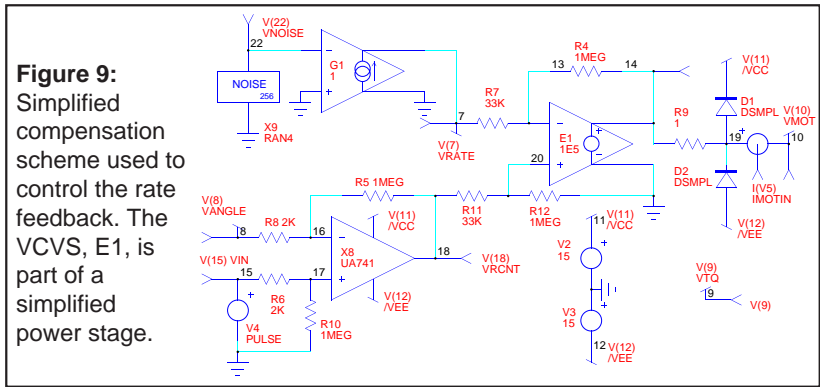
Without rate feedback, the control transfer function takes the form

$$\left(A + \frac{B}{s}\right) \times \frac{(s + W1)}{(s + W2)}$$

where, W1 and W2 are lead and lag frequencies used to maximize bandwidth. The integrator (Figure 8) is needed to reduce the static error. With rate feedback, it is possible to achieve better performance with a much simpler compensation scheme as shown in Figure 9.

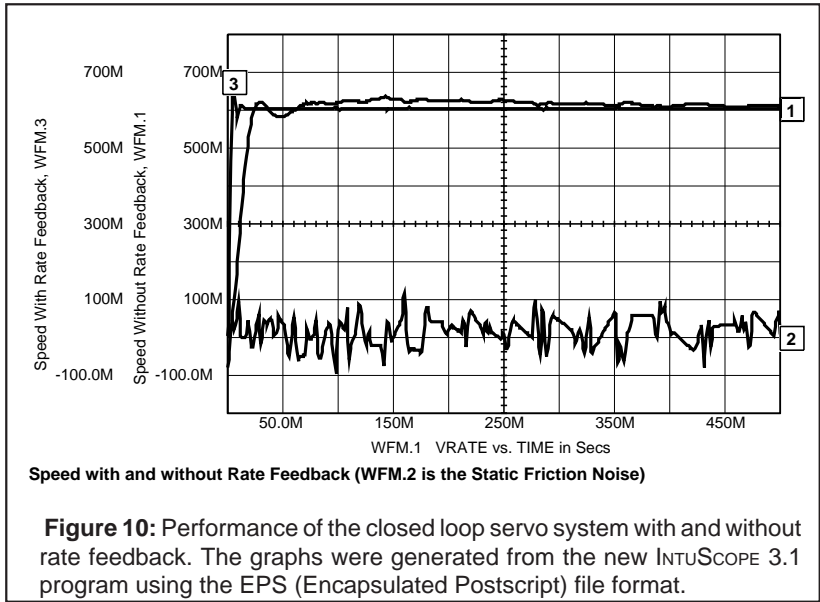
Noise was introduced in the static friction component in order to bring some real world flavor to the simulation. The random noise was converted to a current (10mA RMS) and injected into the rate feedback node, V(7), producing an RMS torque of 2.68E-3 oz.-in. Figure 10 shows the results for the two feedback schemes. This is probably good enough for constant speed; however, the friction is actually dependent on position. A further refinement of the model would involve a more detailed study of noise; perhaps using Chaos theory to make a recursive noise source.





Rate feedback is clearly superior because the differentiation required for the lead-lag compensation increases noise. On the other hand, it is expensive to add a tachometer so we must make a decision based on cost-performance requirements. The electrical performance part of the decision can be illustrated using the IsSPICE simulator. Studies of the open and closed loop response can be performed to evaluate the system's stability under various conditions. Cost and market related questions must then be considered to make the best decision.

The models for the rotational components will be released with our next PRESPICE update. If you would like to obtain these models now, you can order the models, circuits, and schematics from this issue and the October issue on floppy disk by contacting Intusoft about a newsletter subscription.



---

## WINDOWS 3.0 AND SPICE

continued  
from page  
20-3

Programs running in 286 protected mode only use 16 bit instructions and memory addressing resulting in slower than DOS performance (like OS/2). Windows intensive graphics overhead makes operation in a window even slower.

For years, PC users have been trying to improve SPICE's performance and IsSPICE/386 accomplishes this; truly using the full resources of the 386 machine (and all that expensive hardware

### IsSPICE Run Times

DOS IsSPICE	31.53
Windows (Real) IsSPICE	31.42
Windows (Std) IsSPICE	31.80
Windows (386) IsSPICE	49.27
DOS IsSPICE/286	48.94
DOS IsSPICE/386	13.04

Simulation times run under DOS and Windows reveal the power of the 386 protected mode (PM). Times for a Windows 286 PM SPICE version would be similar to IsSPICE/286, a factor of 3 slower than IsSPICE/386.

IsSPICE=version 1.41. (Real)=real mode, (Std)=standard 286 protected mode, 386=Virtual 86 mode. Times are in seconds. Circuit; Sample.CIR (Bipolar amplifier, 15 elements-AC, DC, Transient, and Noise analyses). IsSPICE/286 and IsSPICE/386 could not be run under Windows because of DOS extender incompatibilities. Computer: Compaq 386/20 w/387.

**Note:** complete benchmarks testing several versions of SPICE on various PC/Macintosh platforms are available. Contact Intusoft.

you paid for). IsSPICE/386 is a 32-bit application running in 386 protected mode under DOS. Most other PC SPICE programs only use 286 protected mode. At this time IsSPICE 1.41 runs in all Windows modes. In the future it may also be possible to run IsSPICE/386 in all modes, albeit in the DOS prompt window shell.

In all, use of Microsoft Windows, will greatly increase your simulation time. But more importantly, Intusoft realizes that a good user interface is one of the keys to high productivity and getting people to actually use IsSPICE's power. That's why we created our own interface. It has many of the features of the Macintosh and Windows. Pull down menus, movable windows, and on-line help; but it still allows us to take advantage of DOS extenders and IsSPICE/386. When Microsoft and Windows provide a true 32-bit environment, Intusoft will be there. Until, then, we are striving to give you the best of both worlds now. In the next newsletter, we will provide some hints to help IsSPICE users coexist with Windows.

---

## Eastern Block Countries Open Up

Intusoft has been given the go ahead by the U.S. Dept. of Commerce to advertise and sell its complete line of analog circuit simulation products throughout the eastern block countries including the USSR and its satellite nations. All interested parties wanting information should inquire directly. Intusoft looks forward to supporting and servicing this increasingly important market. If you know of any engineers who would be interested in receiving the *Intusoft Newsletter*, please contact us by fax with the parties name and addresses.