

# Intusoft Newsletter

Personal Computer Circuit Design Tools

October 1990 Issue



Copyright © Intusoft, All Rights Reserved  
(310) 833-0710 Fax (310) 833-9658

## INTUSOFT PREVIEWS NEW PRODUCTS AT WESCON

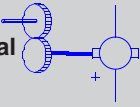
Intusoft will be previewing several new products at the Wescon/90 convention in Anaheim, CA on November 13, 14, and 15. If you are interested in learning about analog circuit simulation or just want to ask a few questions,

**Come See Us At WESCON/90  
Anaheim, CA Booth 322**

We'll be showing our complete line of circuit design tools, along with several new products including:

**SPICE Modeling Spreadsheet  
Graphical Data Post Processors  
SPICE Model Library Update  
Special RF Device Models  
Plessey Analog ASIC Toolkit**

### In This Issue

- 1 New Products From Intusoft At Wescon
- 4 **Analog Behavioral Modeling With SPICE**
- 6 **Modeling Mechanical Systems** 
- 10 IsSPICE/386 On The 486  
Intusoft On The Move

## Give SPICEMOD A Test Run

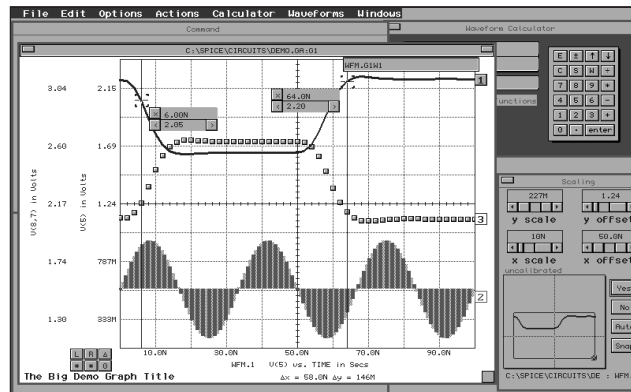
At WESCON, Intusoft will formally introduce SPICEMOD, the SPICE Modeling Spreadsheet for the PC. SPICEMOD is a software program that can be used to develop an unlimited number of SPICE models from data sheet values. If you have heard about SPICEMOD but not seen it in action, now is your chance. Come to the Intusoft booth and give SPICEMOD a try. Any model you create, can be taken home to try with your SPICE simulator. Remember to bring the manufacturer's data sheet for the components you want to model. SPICEMOD allows models to be developed for Diodes, BJTs, power transistors, Darlingtons transistors, Jfets, Mosfets, and power Mosfets.

## Macintosh Offers Powerful Simulation Options

Now Macintosh users can perform interactive data post processing and simulation with Intusoft's powerful line of analog circuit simulation products; the only simulation system that runs on any Macintosh computer at an affordable price. The newly released INTUSCOPE post processing program uses the Macintosh interface to its full extent, providing access to every feature through easy to use pull down menus. INTUSCOPE eliminates the unwieldy line printer plots and the

## Macintosh Simulation Options (cont'd)

difficulties associated with SPICE data analysis. With INTUSCOPE, no patchwork measures like cutting and pasting data or file conversion are necessary. INTUSCOPE is dedicated to SPICE data analysis and display, unlike other methods used for data display such as spreadsheets or simple graphics display programs. Old and new users can come to the Intusoft booth and see the new version in action.



INTUSCOPE version 3.0 contains unique features such as:

- New, easy to use, graphical interface with pull-down menus and multiple graph windows.
- Ability to display and compare data from several output files and several analysis types on the same graph.
- New Print Preview mode providing report quality graphics output to Laser and dot matrix printers and plotters.
- Special Scaling Window with waveform zoom feature allowing easy scaling and configuration of waveforms.
- Advanced Waveform Calculator and pull down menus providing access to all of Intuscope's mathematical operations.
- Support for large waveforms; only limited by available memory. Waveforms with over 20,000 points can be analyzed.
- An on-line help system provides help on all menu items. Numerous examples are also included along with complete documentation.

With the introduction of INTUSCOPE, IS SPICE, and PRE SPICE, Intusoft provides a powerful, integrated, low cost analog circuit simulation system for any Macintosh computer. Intusoft will be demonstrating INTUSCOPE and the entire line of simulation products at WESCON. Come and see your future simulation system at WESCON.

---

### **PRESPICE Update Contains Over 500 New Models**

A complete list of the models contained in the forthcoming PRESPICE model library update, along with various modeling application notes, will be available at the Intusoft booth. Application engineers will be on hand to discuss the models and take requests for future modeling projects. A partial list of the models in the PC and Macintosh updates includes: diodes (Schottky, PN, rectifier, switching, varactor, PIN), transistors, power transistors, darlington transistors, Jfets, Mosfets, dual-gate Mosfets, power Mosfets, op-amps (bipolar, JFET, current feedback), comparators, crystals, opto-isolators, MOV's, pulse width modulators, switched capacitor IC's, analog behavioral models, vacuum tubes, SCR's, voltage regulators, timers and more.

---

### **Models For RF Transistors Available**

Intusoft has introduced a special SPICE model library for RF devices containing models for popular transistors. The models can be used with any Berkeley SPICE compatible simulator. The library includes models for foreign (Philips) and domestic (Motorola) devices such as the BFR90, MRF901, MRF951, 2N2857, and 2N5109, to name a few.

Unlike some other RF BJT models, the Intusoft models do not use unrealistic parameters to fit device behavior to the standard Gummel-Poon model. A custom subcircuit approach is utilized that accounts for all package parasitics and matches the published s - parameter data up to 5 GHz.

---

### **Intusoft, Plessey, and Aptos Combine to Create Low Cost Analog ASIC Design ToolKit**

The Intusoft analog circuit simulation package, ICAP, has been selected by Plessey Semiconductors as the front end schematic/simulation software to support their bipolar analog semi-custom integrated circuits. The ICAP package joins the IC layout software (Cell Pro) from Aptos Systems and the design manuals, SPICE models, and breadboard parts provided by Plessey to create an "Analog Toolkit" (ATK) that designers can use to develop analog semicustom integrated circuits. The design kit supports the several analog bipolar macrochip arrays. The basic ICAP package, along with Plessey's models and kit parts, and the IC layout software (Cell Pro) is available. Detailed technical information on the "ATK" design kit will be available at the Intusoft booth at WESCON or by contacting Intusoft.

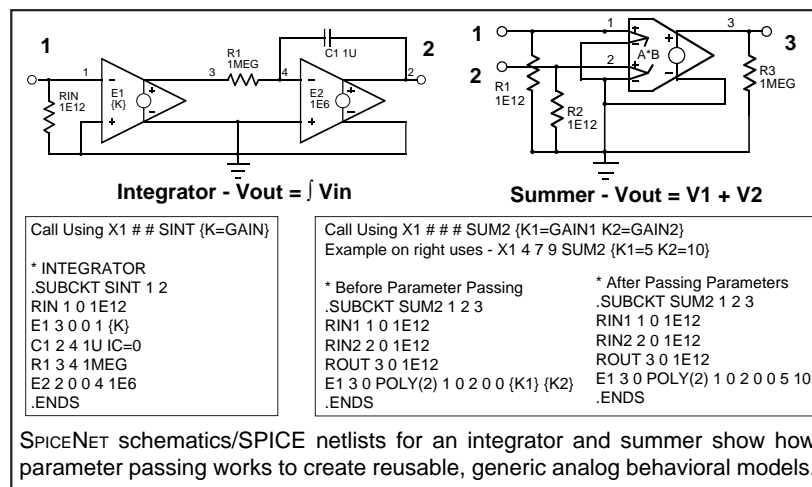
## Analog Behavioral Modeling With SPICE

The basic set of SPICE primitive elements, (resistors, diodes, dependent sources etc.) each exhibit a specific set of properties. Independently, or in combinations, these elements can be used to create a wide variety of functions or building blocks. These building blocks, more commonly known as analog behavioral models, are a class of model that emulates a specific transfer function. Although normally mathematical, the transfer functions can also represent a much wider class of relationships. In the standard SPICE program, complex equations or transfer functions cannot be directly entered. However, the large number of pre-stored functions available are capable of simulating virtually any transfer function. The analog behavioral models available from Intusoft are listed in Table 1.

### IsSPICE Has Analog Behavioral Models

Analog behavioral models have been available in SPICE since the program was first introduced. The concept and the availability are not new. All of the blocks shown in Table 1 can be built and used with any SPICE simulator, even IsSPICE 1.41. No extra expensive options are needed and you don't need to learn any additional syntax formats. In the Intusoft ICAP package, all the blocks can be accessed using simple subcircuit calls or by connecting up the desired symbols, corresponding to each block, if the schematic entry program, SPICENET is available.

The only difficulty with using these elements in the past has been the constant need to re-enter and reconfigure the basic building blocks for even minor alterations. With PRESPICE and its parameter passing ability, however, these problems are resolved.



**Table 1, Analog Behavioral Models available in  
PRESPICE and the *Intusoft Newsletters***

<b>Input Stimulus Sources</b> 3 Phase, AM, Random Noise, Sawtooth, VCO, Variable Phase	Subtract - $V_{out} = V_{in1} - V_{in2}$ Sum - $V_{out} = V_{in1} + V_{in2}$ Tan - $V_{out} = \text{Tan}(V_{in})$
<b>Mathematical Functions</b> <b>Vout = f(Vin)</b> Absolute Value - $V_{out} =  V_{in} $ Attenuation/Gain - $V_{out} = K * V_{in}$ Cos - $V_{out} = \text{Cos}(V_{in})$ Cube Root - $V_{out} = V_{in}^{3/2}$ Differentiation - $V_{out} = dV_{in}/dt$ Divide - $V_{out} = V_{in1}/V_{in2}$ Exponential - $V_{out} = e^{V_{in}}$ Hyperbolic Tan - $V_{out} = \text{Htan}(V_{in})$ Integration - $V_{out} = \int V_{in}$ Multiply - $V_{out} = V_{in1} * V_{in2}$ Natural Log - $V_{out} = \text{Ln}(V_{in})$ Variable exponent - $V_{out} = V_{in1}^{V_{in2}}$ Sin - $V_{out} = \text{Sin}(V_{in})$ Square Root - $V_{out} = V_{in}^{1/2}$	<b>Logic Blocks - Digital Logic Functions</b> Inverter, And, Or, Nand, Nor, Flip Flop, Latch, Shift Register, XOR, IOR, A/D, D/A, Delay  <b>Laplace Domain - Functions of s</b> First order pole, First order pole-zero, Integra- tor - $1/s$ (w&w/o limits), Differentiator - $s$ , Second order pole, Various filter blocks  <b>Other</b> On-Off Switch, Switch with Hysteresis Variable Impedances, Potentiometer  <b>Mechanical</b> Servo Motor, Torque, Inertia, Friction Gears, Motors, Brakes, Slipage, Rate to Angle

The summer and integrator, in the figure, are two examples of behavioral models. A simple voltage summer can be created using a voltage controlled voltage source. To allow the summer to have variable scaling of its inputs the K1 and K2 scalars are passed from the subcircuit call line X into the subcircuit and evaluated by the PRESPICE Param program before running ISPICE. In the example, the scaling factors are 5 and 10. The resulting subcircuit that would be simulated is shown on the right. Parameter passing allows all of the behavioral models to be reconfigurable based on user inputs at runtime. SPICE users can perform the parameter evaluation manually.

Use of the analog behavioral models is trivial, especially if a schematic entry system is employed. The symbols representing each block can be connected together to form the desired network. Using the block diagram approach to describe a transfer function requires less debugging and is superior to the direct entry of equations. It provides visual feedback of the network, clear documentation of the design, and is less prone to typographical errors. Scaling factors, initial conditions and temperature can also be easily changed or plotted against one another or swept.

With these simple blocks ISPICE can simulate any system that can be described using a set of nonlinear differential equations, including electrical, mechanical, or physics related problems. In the next newsletter, we will look at several more analog behavioral models and solve a generalized second order system.

## Behavioral Models for Mixed Mode Rotational Systems

From disk drives to space vehicles, the ubiquitous Rotational Servo Mechanism must be designed. Integrating the design of these mechanical and electrical systems into a single simulation suite has been accomplished at Intusoft by using the features of the ICAP SPICE pre-processors. First, the schematic symbols in SPICENET have been expanded to include the mechanical parts, and secondly, their subcircuit representations have been added to the corresponding PRESPICE libraries. They are available on disk now for \$20.00, with our next PRESPICE update, or if you prefer, directly from this technical article.

## Selecting a System of Units

Coupling of electrical and mechanical systems requires the conversion of units. For our discussion, we will use the MKS system in which the following applies:

Measure	Unit	Abbreviation
Distance	Meter	m
Time	Seconds	sec
Mass	Kilogram	Kg
Force	Newton	nt
Angle	(Radian)	
Potential	Volt	volt
Current	Ampere	amp
Torque	nt - m	
Energy	nt - m	joule
Power	nt - m / sec	watt
Angular Velocity	1 / sec	
Inertia	Kg - m <sup>2</sup> / sec	

The MKS system joins the electrical and mechanical world together with no need for cumbersome unit transformations. Unfortunately, many of us have everyday experience in the English system so here are a few common benchmarks to guide your sensibilities.

Angular velocity	100 (rad) / sec	954 rpm
Force	1 nt	.224 lbs (102 grams)
Torque	100 ounce-inch	.708 nt - m
Power	1 Horsepower	746 watts

## Modeling an Electric Motor

The first model we will develop is for an electric motor. An ideal motor will bidirectionally convert electrical power and mechanical power. For our electrical analog, we will input voltage and current and the output speed and torque will be scaled as:

1 volt =  $KS \text{ (radian) / sec}$   
 and 1 amp =  $KT \text{ nt - m}$ ;

where  $KS$  and  $KT$  are speed and torque constants. Since mechanical power must equal electrical power, only one of the constants needs to be specified; we have chosen  $KS$  since the speed voltage relationship is usually known. The motor horsepower is limited by the series resistance and drive circuitry which are external to the model. The simplicity of the model scaling comes directly from choosing a system of units in which both electrical and mechanical power are measured in watts.

The SPICE<sub>NET</sub> symbol and its model are shown in Figure 1. This model is strikingly similar to the ideal transformer and pulse width modulator models. All three models conserve energy; that is the power at the input terminals is equal to the power at the output terminals.

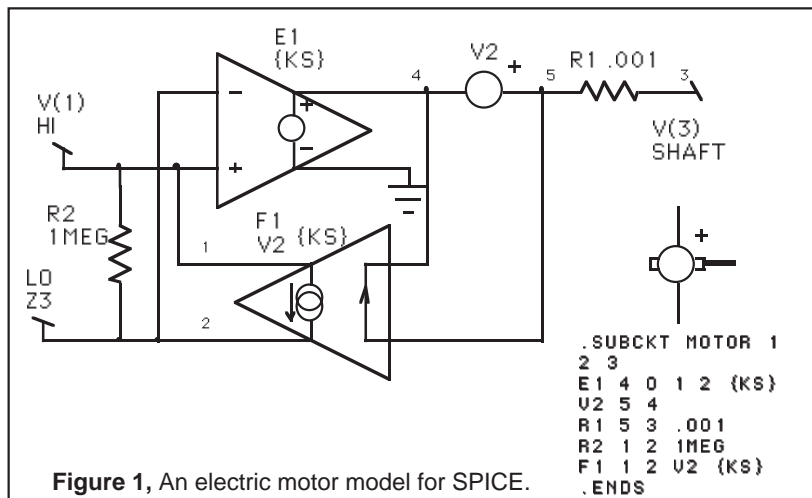


Figure 1, An electric motor model for SPICE.

## Converting From Speed to Angle and Back

Many properties of servos, including their output, require the viewing of angular data rather than speed. A simple integration will provide an angular output test point, but suppose a deadband is to be modeled? Making the model bidirectional would solve this problem. With the voltage/current vector representing speed and torque; then angle and torque could be represented at the output by including a torque to torque feedback. This feedback can be seen to place the integration in a feedback loop resulting in the differentiation of angular inputs in the "reverse" direction. The resultant model and its subcircuit listing are shown in Figure 2.

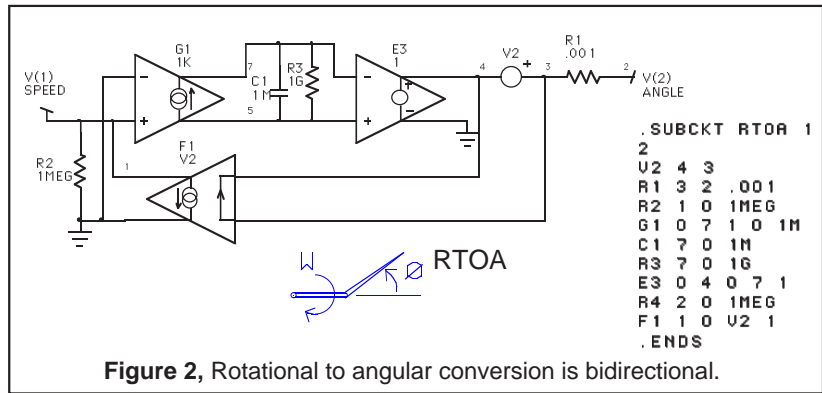


Figure 2, Rotational to angular conversion is bidirectional.

### Gears, Another Energy Conserving Model

Like a lossless electrical transformer, the gear model conserves power by transforming speed in one direction and torque in the reverse direction, satisfying the constraint that the torque speed product at each side of the gear train is the same. The model and its subcircuit listing are shown in Figure 3.

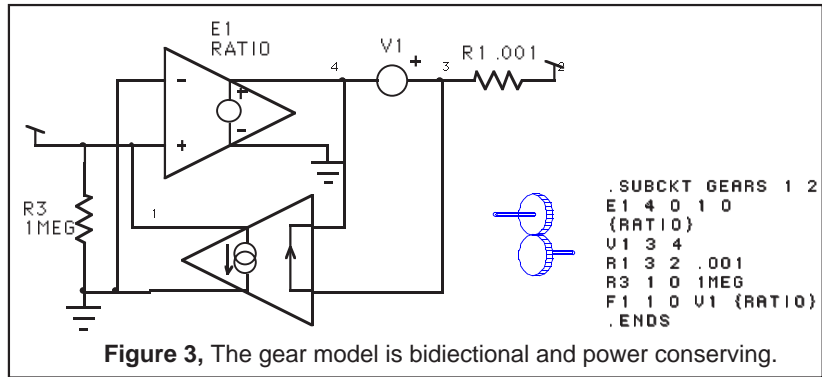


Figure 3, The gear model is bidirectional and power conserving.

### Friction Comes in 2 Flavors

As we have all learned, resistance is the electrical analog of friction; however, starting friction, or stiction, is the really interesting effect. All mechanical systems seem to be corrupted with this non-linearity causing all sorts of problems in control systems design. This is handled in our model by requiring a threshold torque, TRQ, to be exceeded before motion is allowed. The back to back diodes are used to implement this effect. The diode leakage current sets a minimum value of  $10^{-4}$  nt - m or .00019 ounce-inches. The diode emission coefficient was reduced to .07 so that conduction will occur at about 1 deg /sec. Remember, sharp non-linearities can result in convergence problems, so we added a little dynamic damping;  $10^{-6}$  nt - m - sec / (rad). SPICE models for both kinds of friction are shown in Figure 4.



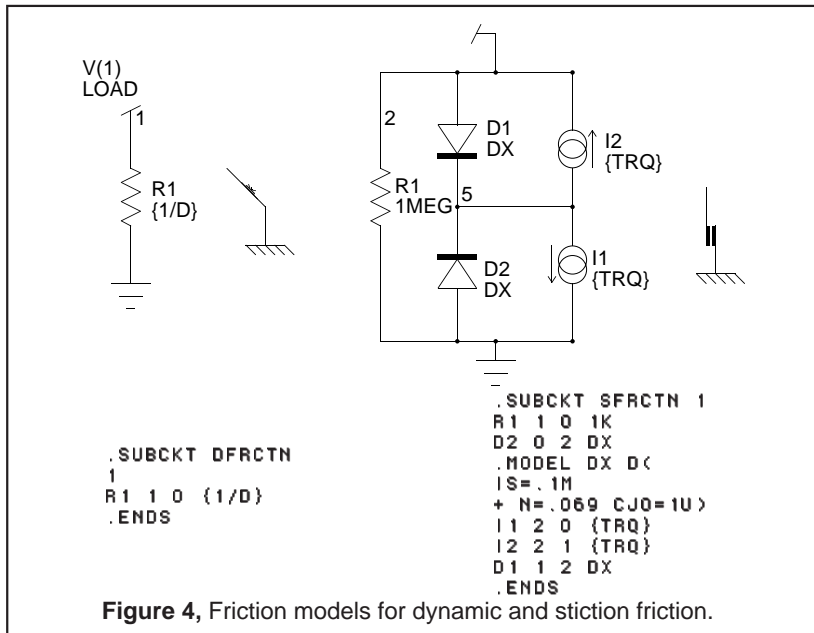


Figure 4, Friction models for dynamic and stiction friction.

### Inertia and Torsion Spring Round Out the Model

The final 2 elements are the mechanical energy storage elements; the capacitor models inertia and the inductor is used for torsion. Figure 5 shows the SPICE models. Notice that the inertia model contains a voltage source with no value. This was done to allow it to have 2 connection points on the schematic. It will also allow you to attach a torque test point. We did not bring the test point through this subcircuit symbol to avoid confusion. If a subcircuit test point is generated, it will have to be connected!

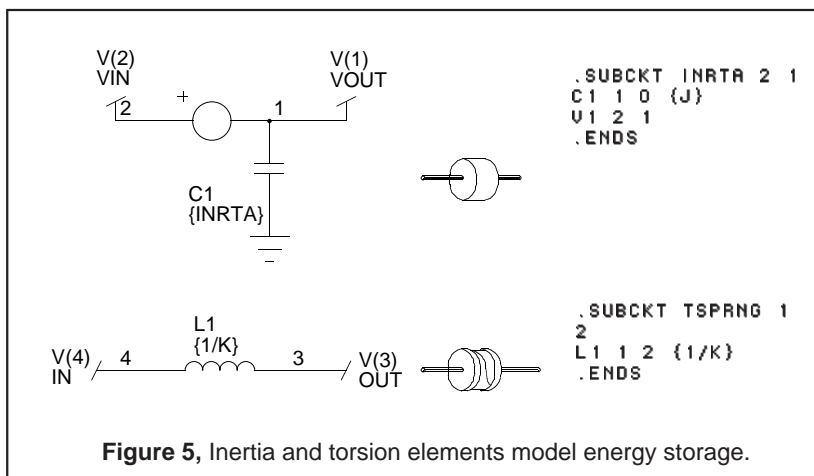


Figure 5, Inertia and torsion elements model energy storage.

## Simulating a Typical Servo

Connecting the new elements together to make a position controlled servo requires adding electrical devices for motor inductance and resistance along with some control circuitry. The schematic for the servo, with some simple position feedback is shown below. In our next newsletter we will show what happens when rate feedback is added.

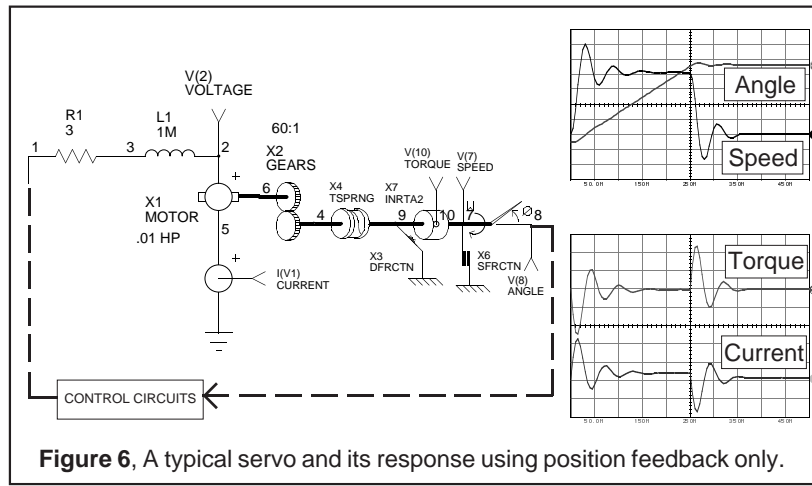


Figure 6, A typical servo and its response using position feedback only.

## IsSPICE/386 On 486 Computers

With prices of 80486 computers coming into a reasonable realm, there have been a number of questions about using IsSPICE/386 on 486 CPU's and whether an IsSPICE/486 is in the future. IsSPICE/386 is a full 32-bit implementation of SPICE and as such uses the 32-bit nature of the 386/486 chips to their fullest extent. IsSPICE/386 also includes special optimizations for the 387, making it even more suited to the 486's on-chip coprocessor. IsSPICE/386 is by far the fastest SPICE program available on either 386 or 486 based machines. It is unknown at this time whether any special 486 extensions would produce tangible speed benefits. In any case, upgrades would be available. SPICE benchmarks using circuits ranging from 3 components to 3000 are available from Intusoft.

## Intusoft On The Move

In addition to moving to larger facilities in beautiful San Pedro, CA this month, Intusoft has shipped its 4000th copy of IsSPICE, (does not include upgrades, updates, or student versions). We at Intusoft would like to thank all of our customers for their support. We have many new products planned for the future and our whole hearted effort to support our current products will continue unabated. If you've chosen IsSPICE for your simulation tool, thank you. If not, we hope our newsletter will assist you in your design work and provide you with a small sample of our commitment to circuit simulation.