

# Intusoft Newsletter

Personal Computer Circuit & System Design Tools



Copyright © Intusoft, All Rights Reserved

Issue #56 June 1999

Tel. (310) 833-0710

Fax (310) 833-9658

## Managing Simulation

With the release of version 8.x.6, the feature set of our ICAP/4 product line was substantially increased ([www.intusoft.com/ver8x6.htm](http://www.intusoft.com/ver8x6.htm)). Virtually all of the new features are unique innovations. In fact, they are unlikely to ever be present in other SPICE programs for the following reason. Other SPICE simulators have never

**Continued on pg. 2**

### In This Issue

- 1 Why You Need ICAP/4  
**Managing Simulation Results**
- 1 Using SPICE More Effectively: **Impedance Measurements**
- 6 SPICE Model Web Links
- 8 Modeling Op-amps  
Pspice® Parts™ vs Intusoft
- 11 Modeling Motors and  
Motion Control Systems
- 15 New Magnetics  
Designer Released

## Using SPICE More Effectively

One of the more common measurements you may want to make is that of a network's impedance. With impedance equal to voltage divided by current the tendency is to drive the circuit with a voltage source. Then, when the simulation is complete, you could display the voltage and current waveforms in the IntuScope post processor and divide the waveforms to get the impedance curve. Although this is possible, it's not the most efficient method.

A more simple approach is to replace the voltage source carrying the AC stimulus (AC 1) with a current source. In the post processor, the displayed nodal waveforms are voltages. However, since the input from the current source has a magnitude of 1 (AC=1) and impedance =  $V_{out}/I_{in}$ , then the voltage waveform (magnitude) corresponds to the impedance versus frequency without any additional manipulations. The circuit in Figure 5 illustrates the

**Continued on pg. 5**

# Managing Simulation Results

Continued  
from pg.  
1

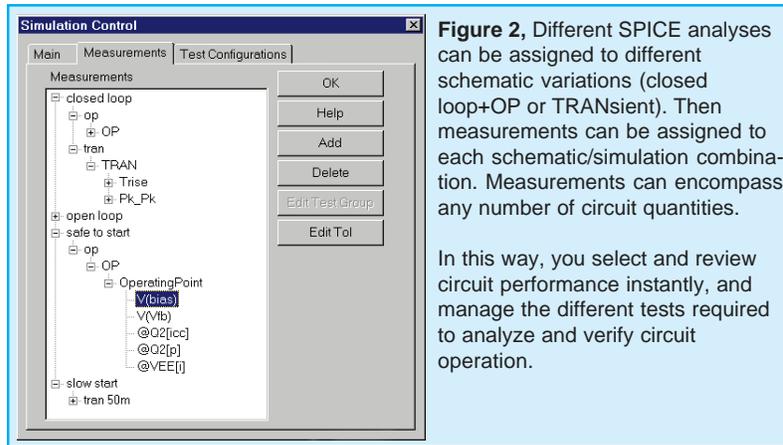
adopted a scripting methodology that can extract data from their simulator. Oddly enough, scripting is a standard Berkeley Spice 3 feature. The ability to run SPICE and process the results using a scripting language is extremely powerful and allows you to better manage the simulation results.

Other SPICE packages, such as OrCAD® PSpice®, require the user to manage simulation results apart from the software. The results must be transferred somewhere (for example, to a spreadsheet) and the user is responsible for managing the schematic/output file/waveform relationships. During the design process, hundreds of “what-if” cases are run in order to perfect circuit performance. These cases will be run using many different circuit, analysis and stimulus configurations, thereby causing an insurmountable bookkeeping problem. OrCAD PSpice has essentially abandoned its users in this regard. As the circuit topology and parameter values are tuned, the previous simulation results can't be easily updated. In fact, it is so difficult using PSpice to do this, it is unlikely to get done at all.

Intusoft is leading the way with Configurable Schematics and “Simulation Templates”, enabling technologies for true worst case analysis. Note that

**safe to start**                      **closed loop**

Figure 1, SpiceNet lets you coordinate design variations. Shown left is an amplifier with a current limited power supply in a “safe to start” configuration. At center is the same circuit with the power sources and stimuli changed. SpiceNet makes it easy to simulate these types of circuit scenarios.



**Figure 2**, Different SPICE analyses can be assigned to different schematic variations (closed loop+OP or TRANsient). Then measurements can be assigned to each schematic/simulation combination. Measurements can encompass any number of circuit quantities.

In this way, you select and review circuit performance instantly, and manage the different tests required to analyze and verify circuit operation.

although the entire process described here is script driven, the user DOES NOT have to write any scripts. All of the operations are graphically setup.

In ICAP/4, the user may capture as many circuit variations as desired (Figure 1). By using schematic layers, it is simple to make and store schematic variations. These variations could be completely different circuits, or a single circuit with different modifications (i.e. component values, tolerances, loads, start-up transients, stimuli, etc.). The variations are all stored in the same schematic database. You can assign measurements to each circuit configuration (Figure 2). The measurement type and circuit quantity (node voltage, current, or power dissipation) are chosen from a dialog. There are no limits to the number of nodes or types of measurements which you can assign. For example, finding the peak-peak or maximum value of every node in the circuit is a trivial matter (Figure 3).

Limits can easily be assigned to each measurement in order to form a baseline for design performance. What-if changes of any circuit variable can then be assessed against the baseline values (Figure 4). Limits can be set directly or through simulation, Monte Carlo analysis, or worst case analysis. Each circuit variation, along with its assigned measurements and limits, are stored along with the schematic and are conveniently viewable, printable and transferable to other tools (i.e. MS Excel, Word, etc.)

The example in Figure 1 shows two different circuit variations, each with different simuli. Any configuration can be selected and simulated at any time. Figure 2 shows the measurements assigned to each configuration. Performing the 50+ measurements over the 6 circuit configurations with any other SPICE program would be tedious at best. Each waveform would have to be displayed in a post processor, cursors would have to be moved, and measurements would have to be stored in a separate program. With ICAP/4, simply select the circuit configuration(s) to simulate, and the entire set of measurements are **automatically recorded**. The results, displayed against the baseline, are shown in the Results dialog, Figure 4. A meter bar quickly shows which measurements are out of tolerance. The ability to store the state of a “golden” or baseline design protects against unanticipated changes that could be detrimental.

In the next issue of the newsletter, we will explore how you can set Stress Alarms on any of the measurements. Try to perform the previous global circuit measurements with your SPICE program!

**Why is this feature important?** ICAP/4 runs an entire suite of tests automatically and creates a report that summarizes the results and pass-fail grades. This powerful capability extends SPICE well beyond its current single simulation oriented boundaries.

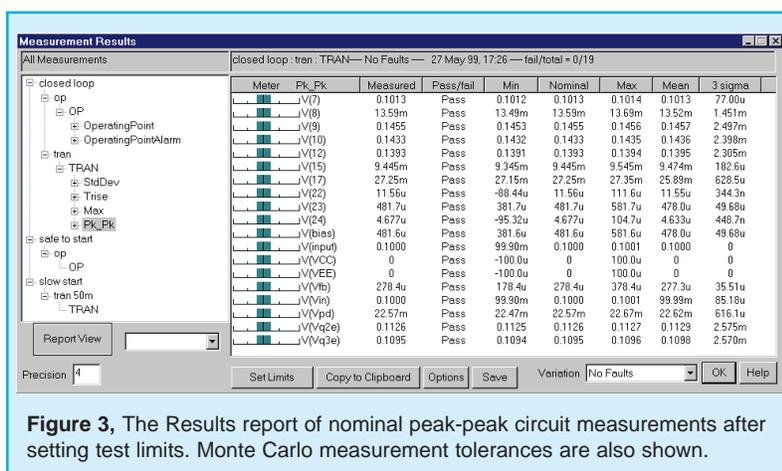


Figure 3, The Results report of nominal peak-peak circuit measurements after setting test limits. Monte Carlo measurement tolerances are also shown.

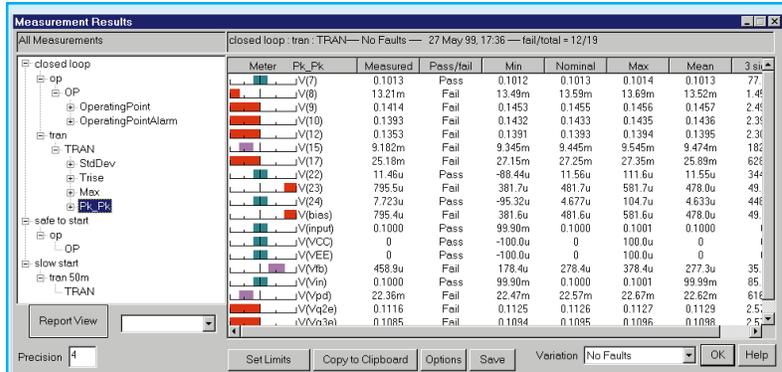


Figure 4, The Results report shows the effects (which measurements failed, and by how much) on the peak-peak values for an alternate circuit configuration using a different resistor value.

### References

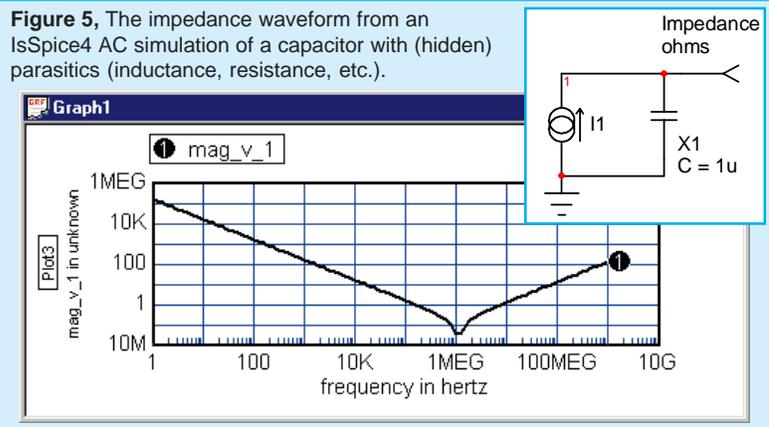
- [1] "New Techniques for Failure Analysis and Test Program Design", C.E. Hymowitz, Intusoft, <http://www.intusoft.com/tech.htm>
- [2] "New Techniques for Failure Analysis and Test Program Design", L.G. Meares, Evaluation Engineering Magazine, April/May issues 1998

# Measuring Impedance

Continued from pg.1

impedance curve of a capacitor which has parasitics. The impedance of a more complex high voltage booster amplifier is shown in Figure 6.

The .TF Transfer function analysis returns the small signal DC input and output impedance at any point in the circuit, plus the DC gain from any source to any



node. It's a convenient way of finding DC impedance and gain. The results are reported in the IsSpice4 text output file (Figure 6). To perform the analysis, you must specify an input source and an output node. The source can be an independent voltage or current source.

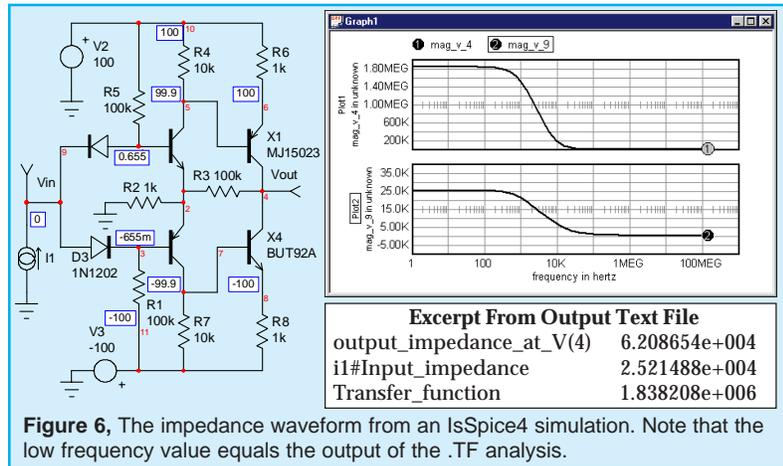


Figure 6, The impedance waveform from an IsSpice4 simulation. Note that the low frequency value equals the output of the .TF analysis.

## Download ICAP/4Rx

You can now purchase and download ICAP/4Rx from the Web. By checking into [www.intusoft.com/OrderRx.htm](http://www.intusoft.com/OrderRx.htm), you can start simulating immediately without any waiting. You'll get the latest version of ICAP/4Rx with unlimited circuit size, superior mixed-signal IsSpice4 engine, and larger SPICE model libraries. With these new features and the ease with which you can get started, it's the perfect time to step up to professional simulation power. ICAP/4Rx is both easy-to-use and powerful, and now it's available instantly.

## SPICE Model Web Links

Listed below are links to manufacturer's web sites where SPICE models can be found. If it's more convenient, you can always find the most current list at the Intusoft web site, at [www.intusoft.com/slinks.htm](http://www.intusoft.com/slinks.htm).

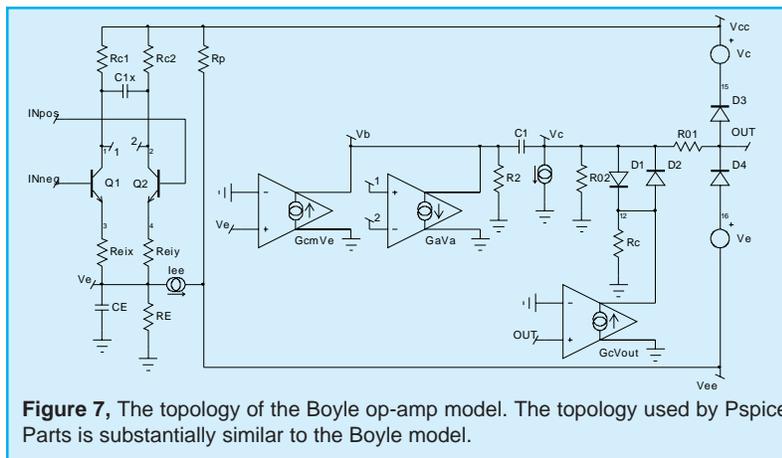
ICAP/4 Windows Deluxe, ICAP/4 Professional and Test Designer users, or those who have our vendor supplied IC/ op-amp models option should note that all of the models posted on vendor's web sites are already in your system. As new models are posted on each vendor's sites, Intusoft performs the integration work for you. Intusoft tracks the models, makes sure that they are compatible with the simulator, tests them, makes the schematic symbols, and posts the updated model library files on the Intusoft technical support web pages.

Vendor	Web Address	Model Types
<b>Linear Technology</b>	<a href="http://www.linear.com/apps/download.html">http://www.linear.com/apps/download.html</a>	Op-amps, Instrumentation Amps
<b>Maxim</b>	<a href="http://www.maxim-ic.com/">http://www.maxim-ic.com/</a>	Op-amps
<b>Motorola</b>	<a href="http://sps.motorola.com/home2/models/">http://sps.motorola.com/home2/models/</a>	Analog ICs, Diode, BJTs, Logic ICs, Op-amps, Opto-electronics, Power Mosfets, Pressure Sensors, Rectifiers, RF BJTs, RF LDMOS, IGBTs
<b>Newport Components</b>	<a href="http://www.newport-comps.com/frames/applications-info.html">http://www.newport-comps.com/frames/applications-info.html</a>	DC-DC Converter and Inductors
<b>National Semiconductor</b>	<a href="http://www.national.com/models/index.html">http://www.national.com/models/index.html</a>	Op-amps (CMOS, BJTs, FET), Buffers
<b>Philips</b>	<a href="http://www-us2.semiconductors.philips.com/discretes/">http://www-us2.semiconductors.philips.com/discretes/</a>	Discretes: RF, Power
<b>Philips</b>	Philips - <a href="http://www-us.semiconductors.philips.com/logic/">http://www-us.semiconductors.philips.com/logic/</a>	Logic Family Models
<b>Polyfet</b>	<a href="http://www.polyfet.com/files.htm">http://www.polyfet.com/files.htm</a>	RF Power Mosfets
<b>SGS-Thomson</b>	<a href="http://www.st.com/stonline/products/support/index.htm">http://www.st.com/stonline/products/support/index.htm</a>	Op-amps, timers
<b>Siemens</b>	<a href="http://www.siemens.de/semiconductor/products/36/368.htm">http://www.siemens.de/semiconductor/products/36/368.htm</a>	Mosfets, Diodes, IGBTs, MOVs, Inductors
<b>Teccor</b>	<a href="http://www.teccor.com/spice.htm">http://www.teccor.com/spice.htm</a>	SCRs, Triacs, Sidactors
<b>TEMIC (Siliconix)</b>	<a href="http://www.temic.com/www/product/spice.htm">http://www.temic.com/www/product/spice.htm</a>	Power Mosfets
<b>TI</b>	<a href="http://www.ti.com/sc/docs/msp/download.htm">http://www.ti.com/sc/docs/msp/download.htm</a>	Op-amps, Comparators, Voltage References
<b>Zetex</b>	<a href="http://www.zetex.com/spice1.htm">http://www.zetex.com/spice1.htm</a>	BJTs, Power BJTs, Darlington BJTs, Diodes, Power Mosfets

# Comparing Op-amp Models

SPICE does not have a generalized built-in op-amp model. Given the variations of the internal topologies of op-amp circuits, it's no wonder. Op-amps are modeled using subcircuits (collections of elements) which can better account for the design variations. Op-amp topologies typically don't emulate all characteristics; only a subset is supported. Op-amp vendors use different internal circuit topologies and usually don't provide much documentation. This makes it difficult to evaluate model performance.

One of the more common topologies used by many op-amp vendors is the Boyle model. The first op-amp subcircuit is credited to G. Boyle [1] (Figure7). Boyle eliminated all but two transistors that formed the input differential stage. The rest of the op-amp is modeled using behavioral and passive elements. Though the Boyle topology achieves significant gains in simulation performance over transistor level topologies, it has many deficiencies. As shown in table 1, it has only 2 poles and its internal node voltages are referenced to ground. Therefore, simulation of many power supply topologies is impossible. The output current flows from a controlled source connected to ground rather than the power supplies. It does not model noise or temperature effects accurately, and its output impedance is not



**Table 1, Op-amp Characteristic Comparison**

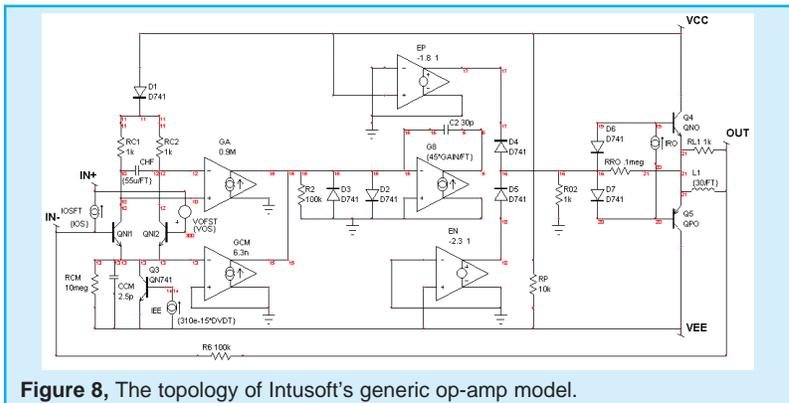
Feature	Boyle	Pspice	Intusoft
Vos, Ib, Ios	X	X	X
DC Gain	X	X	X
GBW	X	X	X
Slew Rate	X	X	X
Phase Margin	X		X
Iout Short circuit	X		X
Vout range	X	X	X
Input Impedance	X	X	X
Output Impedance			X
Vos, Ib, Ios Temperature Dependency			X
Noise Performance			X
CMRR vs. Freq.			X
Isupply Dependent on Iout			X
Internal Ground	X	X	
More than 2 poles & zeros			X
I <sub>sat</sub> Independent of I <sub>+/-</sub>			X
DC Quiescent Power Drain	X	X	X
V/I Limiting	X	X	X

accurately represented. This is essentially the model topology which is used by the OrCAD® PSpice® Parts modeling tool. Several op-amp vendors also use this topology or a somewhat improved version.

The generic Intusoft op-amp model (Figure 8) is much more sophisticated and represents an excellent compromise between simulation performance and the op-amp characteristics which it emulates.

**References**

G Boyle, B. Cohn, D. Pederson, Macromodeling of integrated circuits operational amplifiers, IEEE Journal of solid State Circuits, Vol. SC-9, No. 6 pp. 343-363, 12/74



**Figure 8, The topology of Intusoft's generic op-amp model.**

## Side Bar: Obtaining open loop gain and phase characteristics using a closed loop test.

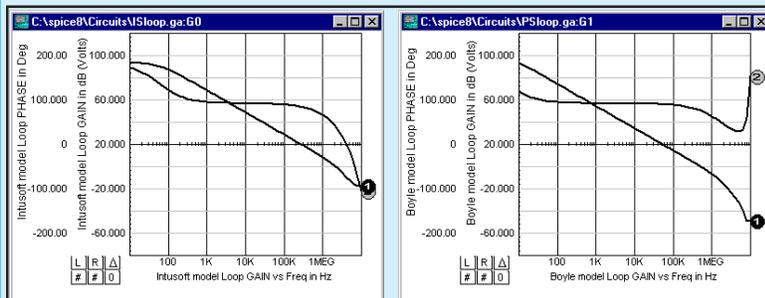
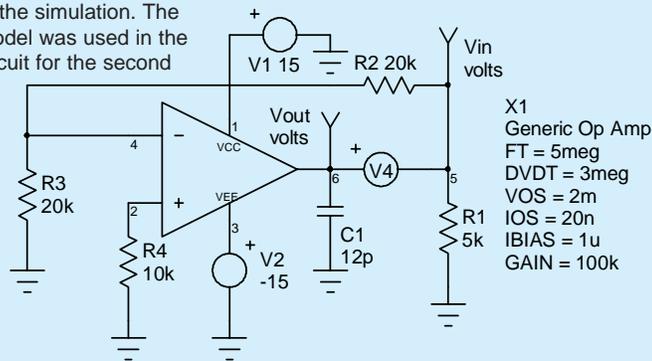
Making a stable bias for a circuit under test without designing a special bias network is a recurring problem. This problem was solved a number of years ago for power supplies, by a number of researchers including Dr. Middlebrook. A signal was inserted via a current probe, and vector {real, imaginary} measurements were made on each side of the probe. The loop gain and phase was then computed as follows:

$$V_{out}(s)/V_{in}(s) = (ReV_{out} + jImV_{out}) / (ReV_{in} + jImV_{in})$$

By converting to gain in dB and phase in Degrees, the respective quantities can simply be subtracted in order to obtain the desired result.

This technique is naturally suited to Spice simulations via the use of a voltage source in place of the current probe. The results of this process are shown below (Figure 10). The phase goes through 360 degrees from DC to infinity, exposing the low frequency, primary stabilization pole and the less desirable high frequency effects.

**Figure 9**, Circuit which was used for the simulation. The Boyle model was used in the same circuit for the second case.



**Figure 10**, A comparison of the results using the two different op-amp models.

# Motion Control Systems

By  
Johan  
Scolliers,  
VTT  
Automation

Motion control systems consist of subsystems from different disciplines such as electrical, hydraulic, and mechanical. The extensive behavioral modeling capabilities, along with the ability to model each part at different levels of complexity, make IsSpice4 perfect for simulation of such Mechatronic systems.

The following example describes a cascade current-velocity-position control system (Figure 11). The desired position,  $\phi$ , is compared with the (sampled) feedback signal  $\phi_S$  from an encoder, and sent through a limiting PI-controller (LPIC). The output is the desired voltage, which is compared with the return signal  $\omega_S$  from a tachometer, and sent to a second LPIC, which calculates the desired motor current  $i_D$ . The current signal is sent through a limiter (LIM), compared with the signal  $i_S$  of a current sensor (CSEN), and sent through a third LPIC. The output,  $i_A$ , of this LPIC is the control signal for the current controlled power amplifier. The power amplifier consists of a 1-phase full bridge diode

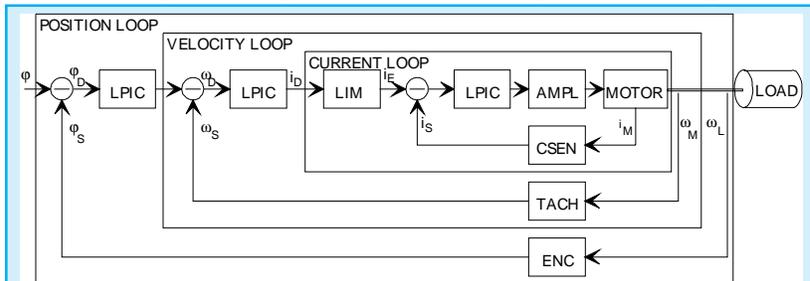
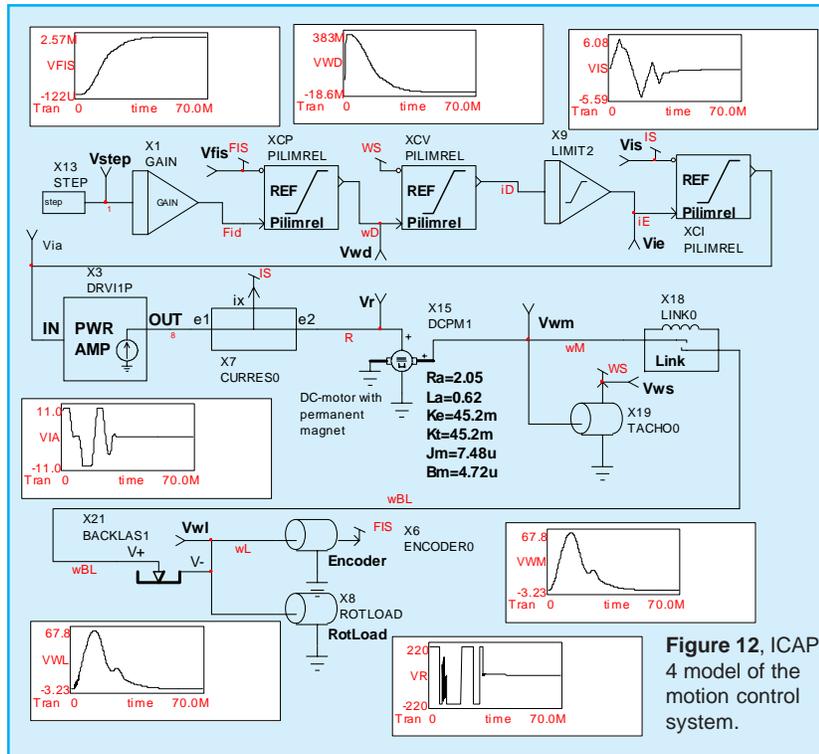


Figure 11, Cascade current-velocity-position controlled drive system.

rectifier which converts the AC net voltage to a DC voltage, and of a full bridge switch-mode converter, built from 4 MOSFETs. The algorithm which is used to control the MOSFET gates is hysteretic current control. The motor is a permanent magnet DC-motor. A flexible coupling with backlash connects the motor to the rotational load, which consists of inertia, damping and friction. Figure 12 shows the ICAP/4 schematic. Figure 13 shows parameter values used for key variables, along with the netlist for the DC motor model. The

(current controlled) power amplifier is modeled on a behavioral level, i.e. as a first order system, with a limiter on the output voltage. For the idealized tachometer and encoder models, no mechanical parts have been used. The parameters of the PI-controllers have been



optimized using classical control techniques. To determine the system control parameters, the system was linearized by making the friction in the motor zero, and by removing the backlash element.

Figure 14 shows the IntuScope simulation results for the load position and the load and motor velocity, for a step change input of the desired position. The oscillations in the load velocity are due to the flexibility between the motor and the load. Figure 16 shows the motor current and the motor voltage.

At the start of the simulation, the desired output motor current is very high. Due to the inductance of the motor, the current cannot immediately reach the desired value,

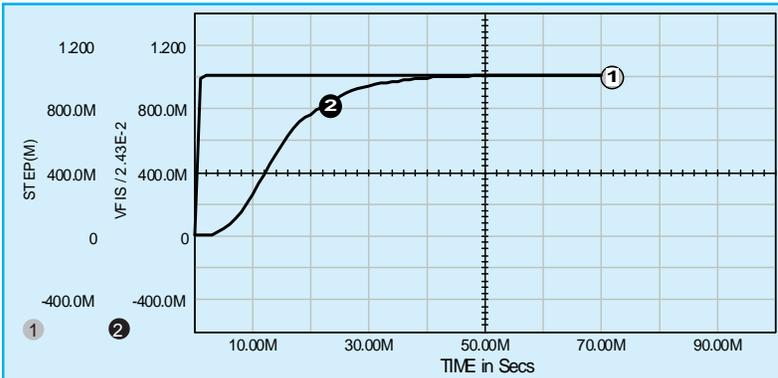
and the output of the voltage is as high as allowed. As the motor current  $i_s$  increases rapidly, the input current  $i_a$  for the amplifier decreases rapidly. After about 6 ms the (scaled) current  $i_a$  becomes smaller than the actual motor current  $i_s$ . Since  $i_a$  increases rather rapidly, the motor current cannot follow the desired current, and the motor voltage is as negative as possible. After 7 ms both currents are again approximately equal to each other,

**Figure 13**, The subcircuit for the DC model used in Figure 13.

```
.Subckt DCPM1 1 2 3 4 {ra=1f la=1f ke=1 kt=1 Jm=1f Bm=1f Wm=0 w0=0}
* input node : 1 2 Electrical Circuit
* output node : 3 4 End Shaft Velocity, Reference
xwind 1 107 winding {r=ra l=la} ; winding model
Bemk 107 105 V={ke} * v(3,4) ; back-emf
vsense 105 2 dc=0 ; current in electric circuit
Bgtrq 4 3 I={kt} * i(vsense) ; induced torque
xrot 3 4 load {m=Jm B=Bm w=Wm v0=w0}
.Subckt WINDING 101 102 {R=1 L=1m}
* WINDING : model of a motor winding
* parameters : R : resistance
* L : inductance
R1 101 1 {R}
L1 1 102 {L}
.ends
.Subckt load 1 2 {m=1 b=1n w=0 v0=0 g=0}
* LOAD : Load (i.e. mass with connection to ground)
* input node : 101 x-component of velocity/force
* output node : 102 (reference, e.g. ground)
* parameters :
* m : load inertia,
* b : load drag,
* w : load friction,
* v0 : initial mass velocity,
* g : gravity
xm 1 massg {m=m v0=v0 g=g}
xd 1 2 damper {b=b}
xw 1 2 frict1 {w=w}
.ends
```

**DCPM1: DC-Motor with Permanent Magnet**  
**Inductive component armature circuit, La=.62**  
**Resistive component armature circuit, Ra=2.05**  
**Electrical EMF constant ,Ke=Kt=45.2m**  
**Rotor Inertia, Jm=7.48u**  
**Rotor Drag, Bm=4.72u**

**Other Selected Circuit Parameters (MKS Units)**  
**LIPC:** PILI1 Kp=proportional gain=150 Ki=integral gain=0,  
PILI2 Kp=50 Ki=30, PILI3 Kp=2.4 Ki=2400  
**Power Amp:** DRV11P Ka=current gain=2.4 Tau=time constant=.2u  
Vs=output voltage=200  
**Linkage:** K=spring stiffness=0.885 b=viscous damping of link=79u  
**Tachometer:** K=gain ( V / (radians/s) )=4.17m, J=inertia of the tachometer-rotor=1n b=damping=1n  
**Backlash:** XBL=total width of the backlash=3.34m K=contact stiffness=211K b=damping=1.25u



**Figure 14**, Step response of the motion control system showing the desired (Vstep) and actual (VFIS) position of the load.

and a (temporary) equilibrium condition is achieved. In order to analyze the motor control more exactly, the behavioral model of the power amplifier can be replaced by an electric model of the amplifier.

[1] Johan Scholliers, Timo Yli-Pietila, The Modeling of Motion Control Systems With An Analog Circuit Simulator, 16th Intl. PCIM, 1995

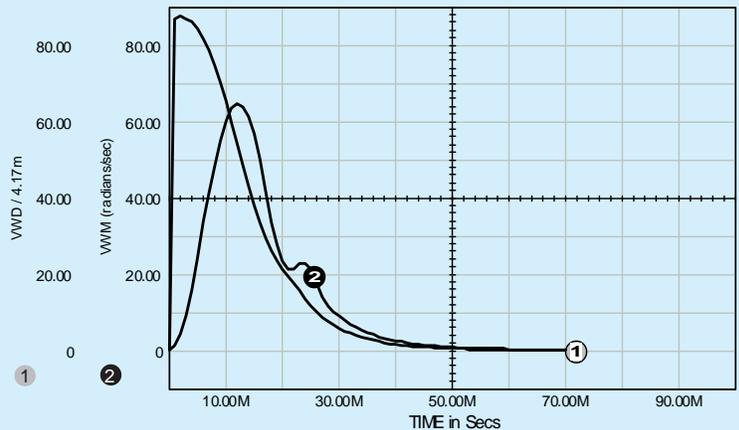


Figure 15, Desired VWD ( $\omega_d$ ) and actual VWM ( $\omega_m$ ) motor velocity.

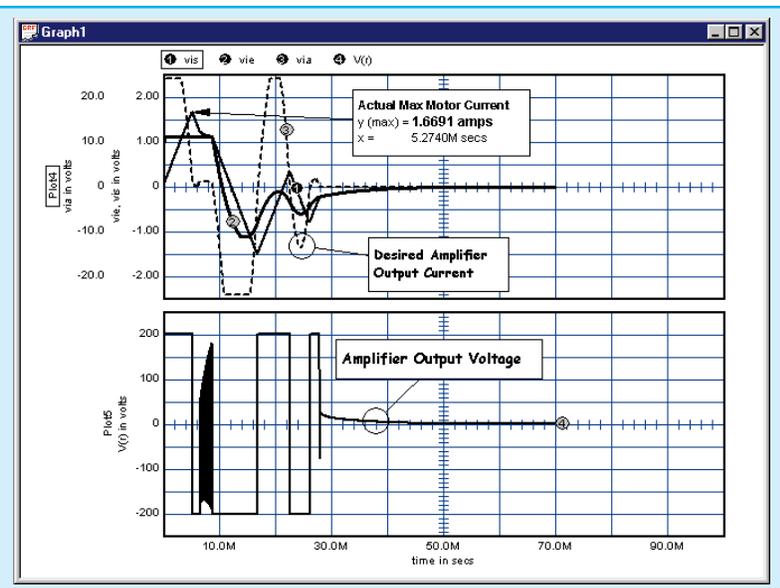


Figure 16, Desired ( $i_d$ ) and actual ( $i_s$ ) motor current and desired output current of the amplifier ( $i_a$ ); Output voltage of the amplifier.

## New Magnetics Design Program

A new version of the Magnetics Designer transformer and inductor design tool will be released in mid-summer (July-August). It contains many powerful feature additions (<http://www.intusoft.com/Mag.htm>).

However, if you're waiting for the release to buy, we suggest that you don't. The price is going up. If you purchase now, you can get the new version 4.0 as soon as it's released, and you'll get all of the other enhancements for the next year, as part of your maintenance. If you wait, you'll pay more for both the product and for maintenance. A beta version of Magnetics Designer 4.0 has been released and is NOW available for downloading for registered users with current maintenance (<http://www.intusoft.com/support.htm>).

PC PRODUCTS	PRICE	DELIVERY			
ICAP/4Rx 1*	\$995	immediate	FilterMaster Active	\$790	immediate
ICAP/4Windows* 1	\$2,984	immediate	FilterMaster Passive	\$790	immediate
ICAP/4Windows* w/Deluxe Option	\$3,657	immediate	FilterMaster Prof.	\$2,225	immediate
ICAP/4 Professional*	\$5,813	immediate	<b>Other PRODUCTS</b>	<b>PRICE</b>	<b>DELIVERY</b>
Test Designer*	\$13,800	immediate	Magnetics Designer* v3.11	\$1,725	immediate
Power Supply Designer*	\$5,385	immediate	SpiceMod* 2.4 (PC only)	\$345	immediate
ICAP for Protel	\$2,995	immediate	Power Designer's Library* 1.1	\$454	immediate
ICAP for OrCAD	\$2,995	immediate	RF Device Models* 3.0	\$454	immediate
ICAP/4Students	\$75	immediate	Vendor supplied IC models*	\$114	immediate
RF Deluxe Option 2*	\$673	immediate	A Spice Cookbook 2,3	\$50.00	immediate
Power Deluxe Option 2*	\$673	immediate	Spice Applications Book, 2nd 2,3	\$49.95	immediate
SALT*	\$569	immediate	SMPS Simulation With SPICE3	\$55.00	immediate
CMSDK* (AHDL Model Kit)	\$4,025	immediate	SPICE Training Class	\$1,200	Inquire

Prices listed are U.S. domestic and are subject to change without notice.

\* One year maintenance is included in these prices.

**Note 1:** ICAP/4Windows and ICAP/4Rx v8.x.6 run on Windows 95/98 and NT only. All ICAP packages are complete systems including SpiceNet, Model Libraries, IsSpice4, and IntuScope.

**Note 2:** The Deluxe option includes: SpiceMod, RF Device Models OR Power Supply Designer's Library, all vendor supplied IC libraries, the "Spice Applications Handbook" and "A Spice Cookbook". The Deluxe option can be added to any ICAP package except the Student Version.

**Note 3:** The "Spice Applications Handbook" contains the first 34 *Intusoft Newsletters* (from 6/86-1/94). Models, schematics, and netlists are included on floppy for the Applications Handbook/A Spice Cookbook.

**Note 4:** The CMSDK is a tool for developing AHDL (C code) models. Requires NT/9x and Microsoft VC++ 5.x.

**Payment:** Check/money order drawn on a U.S. Bank in U.S. Dollars, MasterCard, Visa, American Express, or C.O.D. (U.S.A. only). Programs include a 30 day money back guarantee. Contact factory for shipping charges. Pspice is a registered trademark and Parts is a trademark of OrCAD. All company/product names are trademarks/registered trademarks of their respective owners. Prices subject to change without notice.