Intusoft Newsletter

Personal Computer Circuit & System Design Tools

Newsletter Goes HTML

Beginning with this issue, the internet will be the primary method of publication for this newsletter. Why are we jumping to the web? Well, using HTML hyperlinks, we can instantly link you to associated stories in our vast Newsletter Archives(http://www.intusoft.com/newsletterindex.htm) or provide you with instant access to data within our on-line documentation. You'll find that the body of the newsletter is more compact, but the content will be far richer, providing you with much more information at you fingertips. If you are currently on our mailing list, we will notify you by email when the newsletter is posted. If not, drop us a line by email at info@intusoft.com, and we'll add you to our list. Enjoy our new format!

Newsletter Subscribers: You will receive a password to the subscribers page on our web site along with a hardcopy of the newsletter. The subscriber's page will provide you with access to the supplementary models and drawings files used in preparation of this newsletter.

IntuScope5 Uses ICL

With the introduction of IsSpice4 back in November 1993, Intusoft adopted the Berkeley Script Language for controlling IsSpice. We called this language the **I**nteractive **C**ontrol **L**anguage or **ICL**. Over the years, we have enhanced the ICL; most recently for our enhanced waveform viewer, **IntuScope5**. Shown below is the Calculator menu that reflects the contents of our hierarchical "script" files.
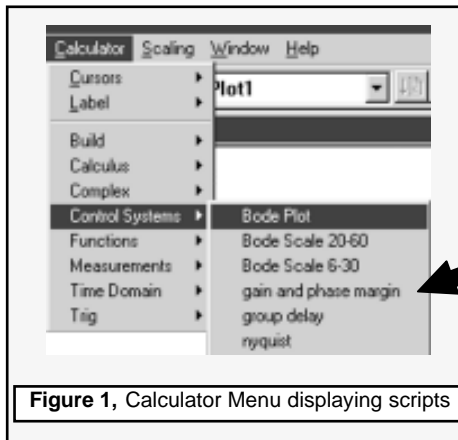

Figure 1, Calculator Menu displaying scripts

The leafs of the menu are the ".scp" files shown in Explorer view below (Figure 2).
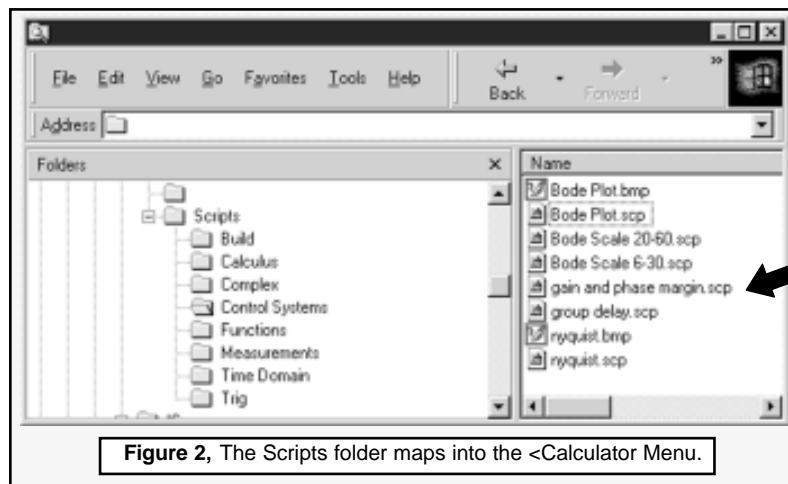

Figure 2, The Scripts folder maps into the <Calculator Menu.

We have enhanced the ICL language for use with our SpiceNet schematic in order to make measurements, retrieve the results of those measurements, and save them in the schematic's storage archive. These ICL scripts are also an integral part of our *Simulation Template technology* (http://www.intusoft.com/script/pages/templates.htm), used for extending IsSpice to perform a family of parametric simulations including worst case analysis.

IntuScope5 uses the exact same script language to manipulate waveforms. The scripts are integrated with IntuScope5 by adopting the same Plot and Vector architecture used in IsSpice. (See http://www.intusoft.com/script/pages/plotvec.htm for a description of IsSpice Plots & Vectors.)

Assuming a plot is a container for graphs that we view on the screen, each plot has one and only one default or x-axis. In IntuScope4, we allowed anything into a plot so that only the active plot has the correct x-axis. Because IntuScope5 is a bit more rigid, we can allow math between any vectors shown in a plot. When a vector is copied into a plot from another simulation, we can make a new x-axis vector that combines the 2 x-axes and interpolates all of the vectors in that plot, mapping them into the new plot space.

When the user chooses a Calculator menu item, the file with a ".scp" extensions is read by IntuScope5 and executed. Analogous files with the same name and different extensions provide context-sensitive help and bitmaps for toolbars.

How do Scripts Work?
The first line of the script is a comment line whose text is shown in the status bar. You can then use special codes to make accelerator keys. For example, instead of writing a special code to find the average of a waveform, we execute the following script:

*&a Mean value of current waveform between Cursors 0 and 1.
loadaccumulator average(current) "Mean"

This powerful confluence of features allows you to make complex operations that can be used over and over with a single keystrokes. (Perhaps we'll get an award for fighting Carpel Tunnel Syndrome!)   For example the Bode Plot and Gain and Phase Margin scripts in the Control Systems menu can be combined allowing you to do "what – if" analyses to explore control system stability, a topic for the Power Supply Designers Notebook article included in this newsletter.

We want to hear from you, our users, about scripts that you have made or would like us to make and add to our scripts folder. We've posted a listing of all of the scripts available so you can try them out and let us know what we should ship with the final product. The list is located on our web site (http://www.intusoft.com/products/scope/scope8pic.htm#Priority List).

The major tasks we must address before finalizing IntuScope5 as a replacement for IntuScope 4 are:
- Implementation of the FFT functions
- Connection of Cross Probing
- Replacement of the IntuScope4 programmed Monte Carlo, Sweeps and optimizations with a Simulation Template
- Inclusion of statistical graph.

We've outlined our plans on the web and are interested in your feedback, so let us know what you think!

Power Supply Designer Notebook

Power supply simulations run 300 times faster using a Large Signal Average Model.

Intusoft is committed to being the best supplier of CAE tools and techniques for power electronics design. To do this, we use our tools to produce the kinds of designs that our users are making. An example of this is Magnetics Designer version 4.1.0, which provides a new sector winding capability along with improved IsSpice models. Using this new transformer design technology, we de-

signed an off-line 80-watt, multiple secondary switched-mode power supply. While creating this design, it became evident that a large signal state-average model would be of great benefit in optimizing the power supply behavior using IsSpice interactive features.

Figure 3 illustrates the ease of damping the time domain transient at turn-on by adjusting the damping resistor, R27. The average model used in this simulation ran in under one second, while a cycle-by-cycle switching solution takes over 300 seconds. Clearly, the speed of the simulation affects the quality of the design. For this example, it is unlikely that the user would have run a parametric study of the turn-on transient and would have had some difficulty constructing a laboratory test set that could repetitively switch the input power.
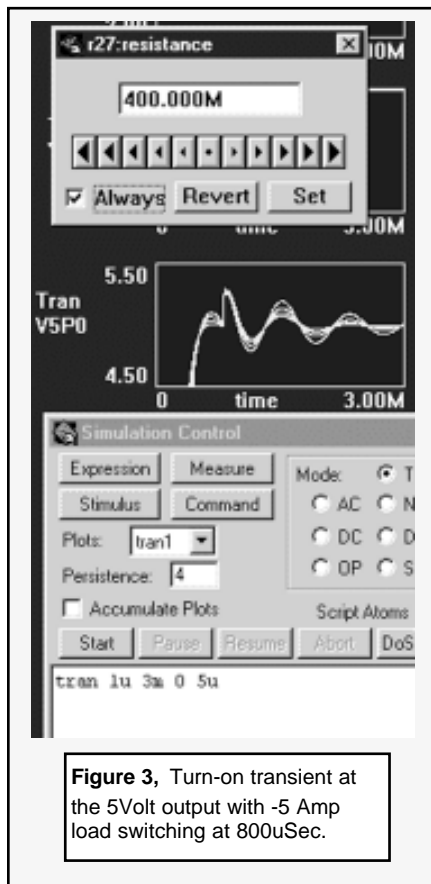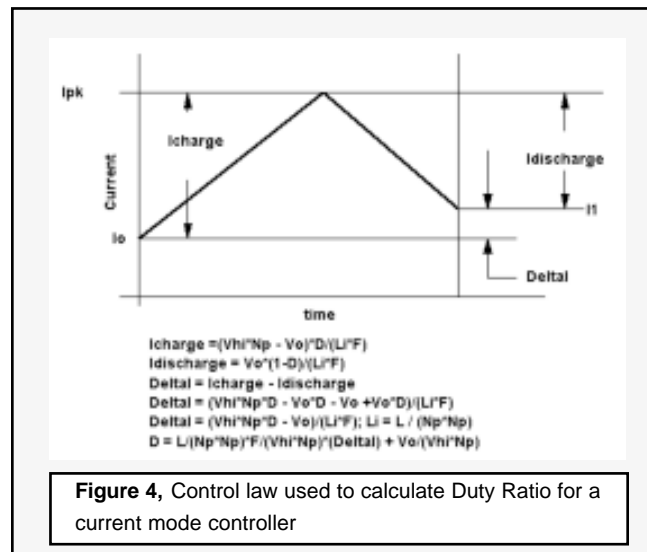
You can see in Figure 3 that the reason for performing the simulation is to study circuit behavior, vary parameters that may be impossible to do on a breadboard, and learn about destructive operating points BEFORE blowing up the breadboard. The need for inserting a damping resistor is less than obvious and the lifetime of an oscillating power supply is likely to be shorter than the time required to correct the problem.

The average model is widely used for small signal control loop stability analysis. The frequent association with small signal analysis gives the incorrect impression that average models are poor choices for large signal behavior. For development of this model, we also wanted to

Figure 3, Turn-on transient at the 5Volt output with -5 Amp load switching at 800uSec.

include an estimate of behavior near the Nyquist frequency because the interactive adjustment of the circuit parameters can easily lead to excess control system bandwidth. We wanted to see instability caused by excess bandwidth.

We were also looking for a good correlation with the cycle-by-cycle switching simulation. The controller is a sampled data system, with sampling at the beginning of the pulse. Figure 4 develops the equation that describes the duty ratio.

Icharge =(Vhi*Np - Vo)*D/(Li*F)
Idischarge = Vo*(1-D)/(Li*F)
Deltal = Icharge - Idischarge
Deltal = (Vhi*Np*D - Vo*D - Vo +Vo*D)/(Li*F)
Deltal = (Vhi*Np*D - Vo)/(Li*F); Li = L / (Np*Np)
D = L/(Np*Np)*F/(Vhi*Np)*(Deltal) + Vo/(Vhi*Np)

Figure 4, Control law used to calculate Duty Ratio for a current mode controller

The switched mode controllers set the peak current in the system. To get good correlation, the peak current ripple needs to be added back into the system. The input is then scaled to control peak current. The overall block diagram of the average controller is shown in Figure 5.

B5 implements the control duty ratio control law while B4 provides the peak correction. Input parameters are:

Np	Secondary to Primary turns ratio
Freq	Switching Frequency
Eff	Efficiency, a fraction
L	Output filter inductor
Lmag	Input magnetizing inductance
DMAX	Duty cycle limit

Figure 5, Implementation of the PWM controller equations
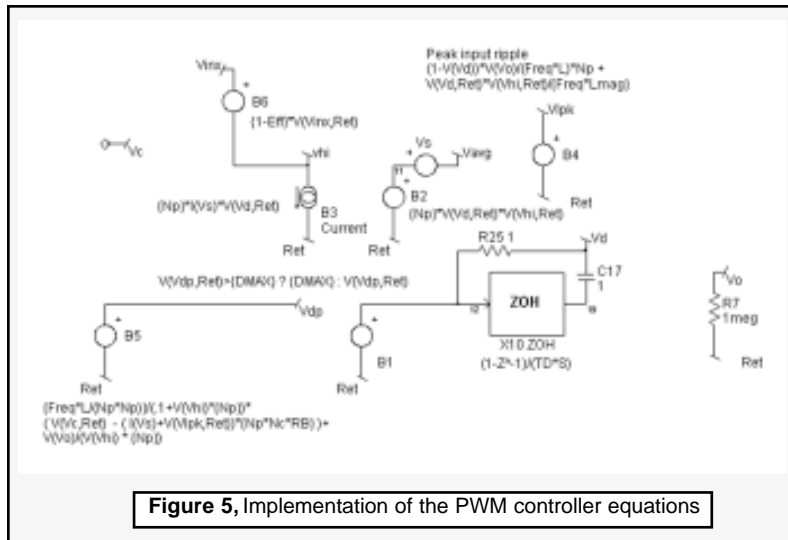
These parameters are set on the main projects Simulation Control Panel, Advanced – Parameters tab.

The ripple in the filter inductor is calculated when the driving voltage is zero:

$$Ipk = Vo * T / L$$

 where T is the off time or (1-V(Vd)/Freq. Notice that this calculation requires the addition of a terminal to get the output voltage, Vo. For multiple outputs, this would be a weighted average of the outputs and L would be a weighted average of inductors. The ripple at the input also includes the transformer magnetizing inductance. The ripple is then added to the output current in element B5 so that the control set point, V(Vc), corresponds to the peak current. The control law is then the same for the switched mode model as the average model.

A zero order hold, X10, models the transfer function near the Nyquist frequency. The theoretical gain of X10 is indeterminate at DC because the numerator term goes to zero as the denominator term goes to infinity. We use an integrator that has finite gain so that the DC gain of X10 becomes zero. R25 and C17 solve the problem by providing a low frequency crossover network. The exact values

for R25 and C17 aren't critical as long as the time constant is much larger than the switching period. B2 and B3 are the energy conserving equations for the pulse width modulator and B6 accounts for the efficiency of the block.

The PWM model is in the PWM configuration of *ForwardCCM.dwg*. It was saved as a sub-drawing that is used in *80ForwardCCM.dwg*. The AVG configuration in *ForwardCCM.dwg* is used to test the average model and the TRAN configuration checks out the switched mode model. These test cases use an idealized transformer in order to speed up the transient simulation. The results, shown in Figure 6, illustrate an excellent agreement between the switched and average models.

**Figure 6,** Switched and Average model agreement
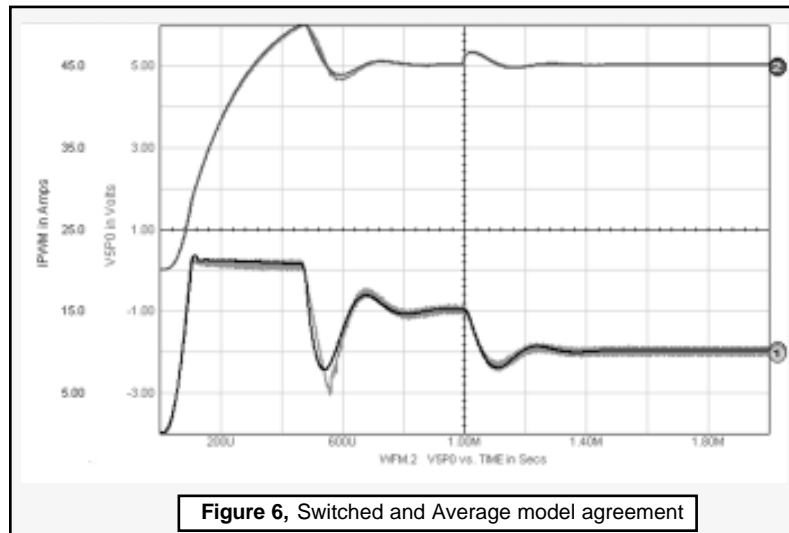
The results are quite impressive. The simplified transient simulation ran in 55 seconds while the Average model ran in 1.5 seconds. The resulting accuracy is more than sufficient to zero-in on the best design parameter.

The switching model still doesn't have real transistors or transformers. Next the models are applied to an 80-watt converter using actual transformer and switching transistors. The fidelity of the transformer model forces the addition of snubbers to reduce EMI and switching load line stress.

In this more complex circuit, the switched mode simulation took 269 seconds while the average model took only 0.87 seconds, which is **over 300 times faster**. The reduced rate of inductor, charging around 100 uSec, is caused by high turn-on stress in the switching semiconductors that reduces conversion efficiency. Major waveform features are still impressively accurate. The average model used here assumes constant conversion efficiency and continuous conduction mode, CCM, in the filter inductor. Generally speaking, these assumptions are valid. Current flow in the filter inductor becomes discontinuous when the commutating diode cannot support negative current required at small loads. This effect, at steady state, causes the regulator to be incapable of control. The controller is a peak rectifier at no load. Something must be done to keep the output voltage in tolerance. The solution generally brings the mode to CCM. One of the more interesting solutions makes the inductor highly nonlinear so that its inductance is up to 10 times higher at low current. The nonlinear characteristics would have to be included in the averaged model.

Figure 7, Turn-on and step load change at 800usec for an 80-watt forward converter.
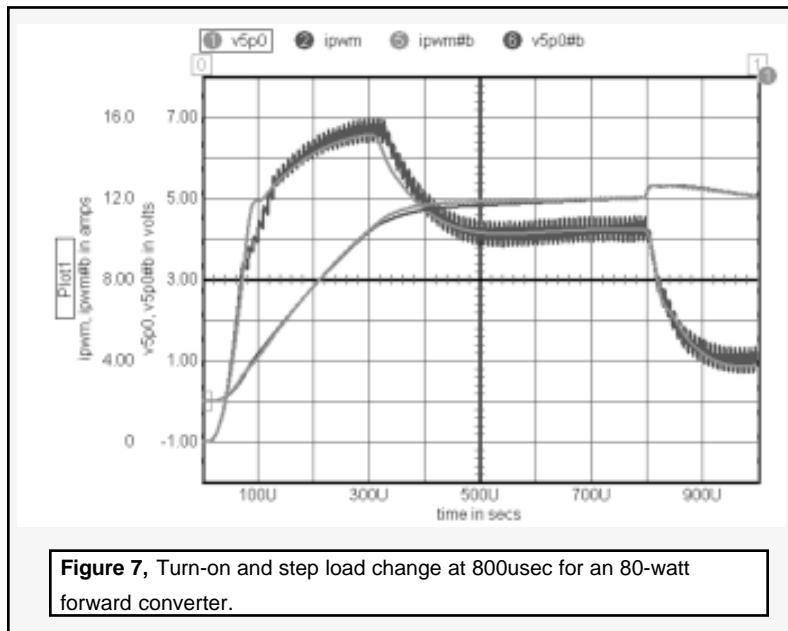
Notice that the compensation circuitry was rearranged for the final circuit (Figure 8). Using lag-lead compensation in the forward loop causes overshoot at turn-on. The op-amp comes out of saturation when the voltage gets to the set point. The control voltage at the op-amp output is too large and stays that way until the integrating capacitor can discharge. An alternate method of compensation is to use lead-lag compensation in the feedback loop and pure integration in the forward loop. The lead-lag compensation causes the integrator to come out of saturation before reaching the output voltage set point. Using lead-lag feedback compensation eliminates the need for additional circuitry, which slows down the start-up in order to eliminate output voltage overshoot.

Figure 8, Compensation circuitry comparison
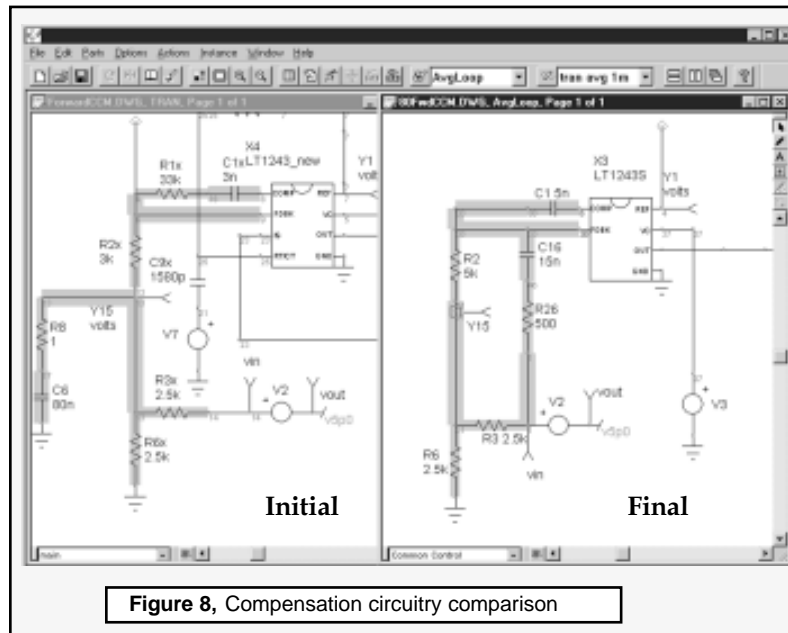
Scaling the output filters (C17, R27) and damping the input filter (R5) relied heavily on AC analysis using the average model. IntuScope5 provided insight and rapid parametric evaluation using its new bode plot feature. The IntuScope5 generated bode plot is shown in Figure 9.
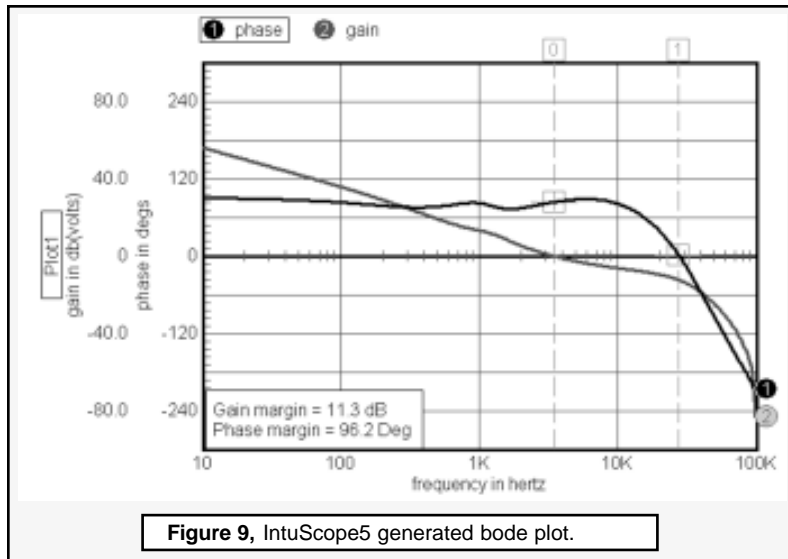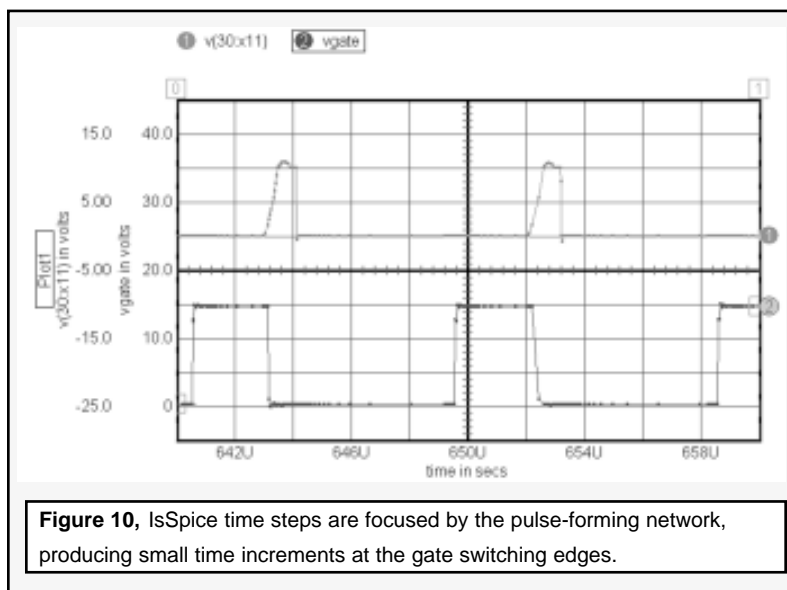
Figure 9, IntuScope5 generated bode plot.

Improving Convergence and Speed of PWM Switching Models.

Speeding up the switched mode models helps get through some of the more complex analyses in a reasonable time. One of the main stumbling blocks is the need to set the maximum step in the simulator so that important features don't get passed over. The variable step algorithm used by Spice can expand the step time beyond the Nyquist limit. Sometimes a stable sub-harmonic solution is reached. This occurs in the LT1243 oscillator model. Since the oscillator waveform resets in several hundred nanoseconds, it becomes necessary to set the maximum step to 25-50nsec to get an accurate reset pulse. This causes the simulator to take more iterations than are really needed. A straightforward solution is to use a pulse source instead of simulating the oscillator. To get the correct waveforms at the IC pins, the capacitor discharge is simulated. Spice will then set breakpoints at the clock edges. Inputting the RT and CT values as parameters allows the PWM model to calculate its frequency.

Gate turn-off switching occurs at the intersection of 2 analog signals, where the time step could be at the maximum if no other activity is taking place. The B element comparator doesn't create break points by itself.

Page 11

The time step can be set to the gate fall time or the circuit can be configured to create breakpoints near the gate switching points. An auxiliary pulse-forming network was introduced in the PWM model to reduce the time step near the gate switching time. The result is shown in Figure 10 with the 1 MegHz secondary regulator turned off. Since the work was done within the model, the user gets the benefit without hassling with the simulator.

Figure 10, IsSpice time steps are focused by the pulse-forming network, producing small time increments at the gate switching edges.

Summary

The complete model for Transient and Average operation is in *1243is.dwg* in the LT1243new configuration. This was saved in a sub-drawing that is used in *Forwardccm.dwg* and *80FwdCCM.dwg*. Finally, the resulting sub-circuits have been added to the Power Supply Designers libraries for all of the LT12xx family parts and are identified as Improved Performance. You can access the library updates and ".dwg" files on our web site in the paid maintenance support page for ICAP/4 Power, ICAP/4 Professional, Test Designer and all Power Deluxe packages.

Newsletter Subscribers: You can also get the ".dwg" files using your private password.

Mechatronics
An Improved Hydraulic Library

You've been asking for tools to help with the incorporation of mechanical devices into electronic systems. In response to your requests, Intusoft is now releasing the first in a series of libraries in the area of Mechatronics. This new extension of the ICAP/4 libraries, which includes Power Designer, RF Device, and Vendor-Supplied IC Library, will be an invaluable asset for anyone using both electrical and mechanical components.

The first chapter of this exciting new library offers a range of models for the development of hydraulic systems. Servicing the fields of agriculture, forestry, mining, machine tools, and mobile equipment, this library is the perfect addition for the ICAP/4 Windows user (Figure 11). Models for piston accumulators, hydraulic cylinders, valves, and losses will all be included in this first release.
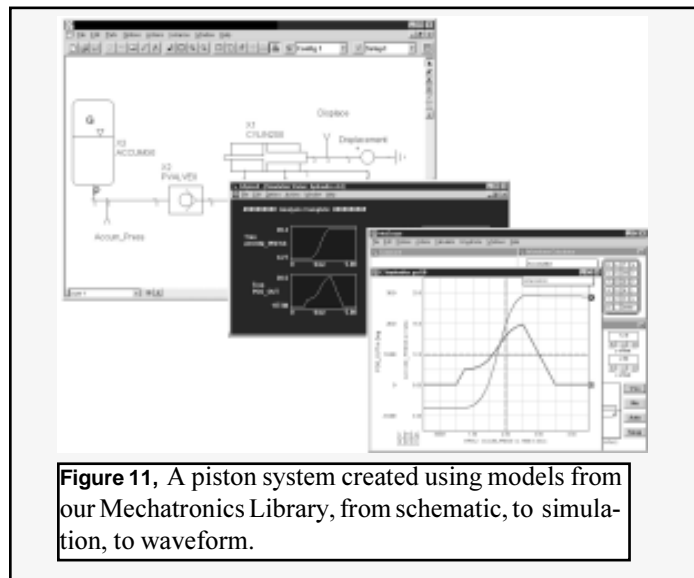

Figure 11, A piston system created using models from our Mechatronics Library, from schematic, to simulation, to waveform.

Spring-loaded and gas-loaded models are offered for accumulators and hydraulic cylinders. Separate models are also offered for the analysis of compressible and incompressible fluid systems. Dual-chambered cylinders as well as standard, single-chambered cylinders will be available, offering a wide degree of freedom and variability. Major and minor head losses in pipe flow are easily simulated using a variety of different models. Pressure valves also provide added versatility for ICAP/4 users.

And, as with the rest of the Intusoft models, each of the new mechatronics models can be custom fitted for use with your hardware. Most standard parameters, such as size, shape, fluid, and material properties, can be modified using the ICAP/4 Windows interface. The ability to edit Spice code is not necessary to customize the library to fit your needs. However, if advanced changes or customization is needed, this feature is still available to you.

Even if you are not experienced with hydraulics, you can simulate systems with ease. The on-line help included with this library was written with the electrical designer in mind. Tables including standard values of common materials are provided so you don't have to search for parameters for preliminary designs and calculations. Also, the general information guide gets you up-to-speed on all of the key differences used in mechanical simulation.

The beta version of this library is now available in the paid support section of our web site for all ICAP/4 Windows, Professional, Power and Test Designer owners. And, if you download the beta version and provide us with feedback, you will receive the final library for the cost of one year's maintenance.

SpiceMod for Windows

You asked for it, we did it! Coming soon, SpiceMod will switch from DOS to Windows. A beta version of SpiceMod will be available on the paid support web pages for those who currently own the DOS version.

We expect to add tabs for many of our generic models, including Soft Recovery Power Diodes, Tantalum and Ceramic capacitors. Let us know what you need and we'll put it on our list.

Purchasing and Technical Notes

System Packages: All ICAP packages are complete systems including SpiceNet, Model Libraries, IsSpice4, and IntuScopeand run on **Windows95, Windows98, or WindowsNT**. ICAP/4 Windows versions 8.x.x, Power Supply Designer, and Test Designer also include vender-supplied IC libraries.

Deluxe option includes the following items: SPICEMOD, RF Device Models OR Power Supply Designer's Library, the **"SPICE APPLICATIONS HANDBOOK"** and **"A SPICE Cookbook."** The Deluxe option can be added to any ICAP package except ICAP/4 Students.

CMSDK: The CMSDK is a toolkit for developing AHDL (C code) models. It requires Windows NT or 95 and Microsoft Visual C++ 4.x-5.x. The CMSDK Bundle includes ICAP/4Windows.

Intusoft Technical Publications: "SMPS Simulation With SPICE 3" is a SPICE reference book for Power Supply/Circuit designers. It includes information on modeling, simulating, and designing switched mode power supplies. The **"SPICE APPLICATIONS HANDBOOK"** contains past Intusoft newsletters (from 6/86 to 1/94, 34 in all). The newsletters include SPICE modeling information, simulation tips and techniques, and much more. **"A SPICE Cookbook"** contains over 100 practical circuit examples encompassing a wide array of topics (RF, Power, Filters, Digital, Microwave). Each example includes a technical explanation, SPICE simulation tips, netlist, schematic, and output graphs. The SPICE models, schematics, and netlists from the above reference books are included on floppy disk along with the book.

Intusoft's World-Wide Support Staff

Listed below are dealers where Intusoft products, updates, information, and support may be obtained.

Dahan Tech Inc. - Sang Y. Cho
South Korea
Tele: 82-2-515-2845/FAX: 82-2-515-2844
email - sycho@dahan.co.kr
Web: http://www.dahan.co.kr

Technology Sources Ltd. - Dr. Graham Plows
Algeria, Denmark, France, Italy, Norway, Sweden, UK
Tele: 44-01223-516469/FAX: 44-01223-729-916
email - info@softsim.com
Web: http://www.softsim.com

DS-Design Systems OY-Hannu Tikkanen
Finland
Tele: 358 14-652588/FAX: 358 14-610725
email - sales@designsystems.fi
Web: http://www.designsystems.fi

Thomatronik GmbH - Herbert M. Müeller
Germany, Austria, Croatia, Slovenia, Chech Rep.
Tele:49 8031-2175-0/FAX: 49 8031-2175-30
email - info@thomatronik.de
Web: http://www.thomatronik.de

Cho Chieh Enterprise Ltd. - Tennyson Lin
Taiwan, Hong Kong, China
Tele: 886-2-2289-2605
FAX: 886-2-2289-0151
email - chochieh@ms2.hinet.net

Cadforce Systems, Inc - Reg Harrison
Montreal Canada
Toll Free (800) 249-7332/FAX (514) 426-0266
Ottawa Canada
Toll Free (800) 469-3086/FAX (613) 228-1021
Toronto Canada
Toll Free (800) 444-8794/FAX (905) 281-0466
e-mail: reg@cadforce.ca or Jean@cadforce.ca
Vancouver Canada - Monique Masserey
Toll Free (888) 464-8552/FAX (604)464-8556
e-mail: monique@cadforce.ca

PCB Graphtech Pte Ltd - Francis Wong
Singapore, Malaysia
Tele: 65-2611-933/FAX: 65-2611-938
email - frawong@pcb-graphtech.com.sg

Elbo - Bogdan Olejarczyk
Poland
Tele/FAX: (48-22) 838-23-91
email - elbo_@astercity.net

TECH 5 - Geert Mosterdijk
Netherlands, Belgium, Luxenburg
Tele: 31-184-6155-51/FAX: 31-184-6154-51
email - info@tech5.nl
Web: http://www.tech5.nl

Trident Technologies - Praveen Kapoor
India
Tele: 91-11-578-7055/FAX: 91-11-576-7055
email - ttpl@del2.vsnl.net.in

ChipCAD - Tibor Berky
Hungary
Tele: 36-1 399-4290/FAX: 36-1 399-4299
email - tholman@chipcad.hu
Web: http://www.chipcad.hu

Computer Controls AG - Peter Wannenmacher
Switzerland
Tele:41-1-308-66-66/FAX:41-1-308-66-55
email - peterw@ccontrols.ch
Web: http://www.ccontrols.ch

ME Technologies - Peter Stein
Australia, New Zealand
Tele: 61-2-65-50-2254/FAX: 61-2-65-50-2341
email - info@me.au.com
Web: http://www.me-au.com

IVIS Co., Ltd. - Hiro Nagano
Japan
Tele: 81-45-332-5381/FAX: 81-45-332-5391
email - sales@i-vis.co.jp

EDA Software - John Meltezos
Greece
Tele/FAX: 301-884-1016
email - sales@edasoft.gr
Web: http://www.edasoft.gr

INTSYS Europe SA - Claude Masseboeuf
France, Tunisia, Algeria, Morocco
Tele:(33)01-60-81-00-69/FAX:(33)01 60 81 00 70
email - claudem@intsys-europe.fr
Web: http://www.intsys-europe.fr

*DFM - Tuvia Liran
Isreal
Tele: 972-4-9533059/FAX: 972-4-9533057
email - dfm@netvision.net.il
Web: http://dfm4vlsi.com

*New information   Intusoft welcomes our new representative in Isreal.