

Intusoft Newsletter

Personal Computer Circuit & System Design Tools



Copyright © Intusoft, All Rights Reserved

Issue #54 Nov. 1998

Tel. (310) 833-0710

Fax (310) 833-9658

INTELLIGENT WORST CASE

The new release of ICAP/4 8.x.6 includes several major feature additions. ICAP/4Windows gets new RSS, EVA (Extreme Value), Worst Case, and Sensitivity analyses. In addition the powerful Design Validation and automated measurements feature (Design Validator) has been added.

Continued on pg. 2

7.X, DOS, & 8.1.0 USERS SAVE A BUNDLE OF \$\$

A special time limited offer is now available for ICAP/4 users who have a 7.x or DOS version. From now until December 31, 1998 you can upgrade to the latest version of ICAP/4Windows, 8.x.6, for a **substantially reduced cost.**

Continued on pg. 13

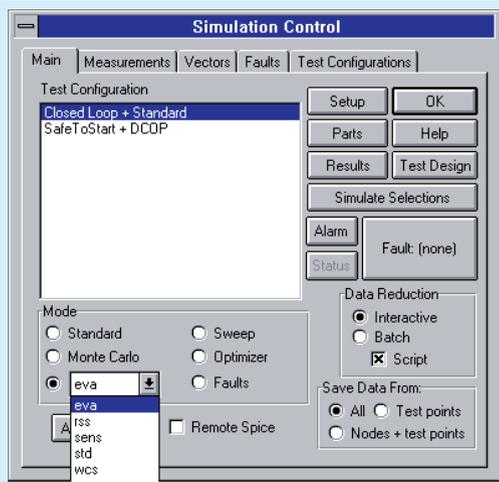
In This Issue

- 2 **New ICAP/4 8.x.6 RSS, EVA, and Worst Case Analysis**
- 4 Units Conversion
- 4 Bonus Web Stuff: IBIS Converter, Units Calculator & Models
- 6 Magnetics Design Tool, A Flyback Transformer
- 12 AC Bias Problems
- 13 **RX for Sale On the Web**
- 15 SPICE For Linux

Figure 1, ICAP/4Windows version 8.x.6 includes many new features (See Page 2).

RSS, Extreme Value, Worst Case, and Sensitivity are now available for AC, DC, Transient and operating point analysis.

Users can define their own "Simulation Template" scripts to perform complex analysis sequences like those used for RSS and EVA (See Figure 2).



NEW ICAP/4 VERSION 8.X.6

Continued
from pg. 1

ICAP/4Rx has added new mixed-signal simulation capabilities and digital models. The Design Validator product is now sold as ICAP/4 Professional and includes the automatic simulation and data presentation of all failure modes; a feature formerly limited to Test Designer. The update is summarized in Table 1.

Simulation Templates and Automated Measurements

One of the unique new features in 8.x.6 is called "Simulation Templates" (Figure 2). These templates are command scripts that direct the IsSpice4 simulator to perform complex analyses and manipulate the database of information stored in the schematic. The initial release of Simulation Templates includes scripts that perform worst case, RSS, sensitivity and EVA analysis for AC, DC and Transient analysis as well as the traditional operating point analysis.

Simulation Templates change parameters one at a time, measure and record the resulting changes in circuit operation, and

```
#tolerance
nextparam null
while param != null
  alterparam tolerance(param)/3
  #simulation
  paramvec = param
  printstatus -p paramvec
  #mprint
  nv = nextvector(null)
  while nv != null
    if length(nv) = 1 dialog
      nv = nv - ref.nv
      result.nv = result.nv + abs(3*nv)
    end
    nv = nextvector(nv)
  end
  unalterparam
  nextparam
end
echo #### RSS HI analysis Results ####
#mprint
```

Figure 2, A portion of an RSS Simulation Template script (left). The script runs transparently when the analysis is activated (Figure 1). The power of this feature is evident once you realize what is actually being done. The template enumerates all the circuit parameters with tolerances, adjusts their value, runs a simulation, and collects the data. The results are displayed in a report (below) and can be compared against simulation or user-defined limits. Automated measurements can be made for any analysis and on any circuit quantity. In the case of the RSS, EVA, Monte Carlo and Worst Case analyses, the returned values can be used to set minimum and maximum measurement limits.

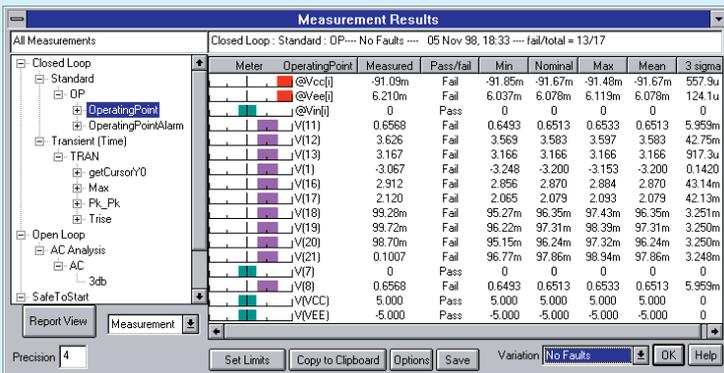


Table 1, New features for the ICAP/4 series of packages. See the Intusoft web site, www.intusoft.com, for a complete summary of features.

ICAP/4Rx

- Mixed-Signal Simulation - ICAP/4Rx now includes a 12 state logic simulator allowing it to simulate full mixed analog-digital circuits
- More Digital Models - Version 8.x.6 adds over 500 digital parts including over 50 new primitive digital elements like flip-flops and a state machine
- More Analog SPICE Models - Over 1,000 more models are included, bringing the total to over 5,000 parts

ICAP/4Windows

- Simulation Templates - Powerful analyses driven by ICL scripts
 - RSS, EVA, and Worst Case Analyses
- Sensitivity Analysis for Transient, AC, and DC Analyses
- Design Validation and Automated Measurements
- Automatic Stress Alarms
- Scripted Monte Carlo - A new easy way to run a statistical yield analysis
- More SPICE Models - Total now over 14,000 (with Deluxe Option)

ICAP/4 Professional (previously called Design Validator)

- Automated Fault Analysis - This feature lets you simulate all failure modes in your design and record any set of measurements. All Automatically. This is a huge time savings over manual fault analysis.
- All New ICAP/4Windows features
- RF and Power Deluxe Options Plus SALT

provide a summarized report consisting of user-defined measurements (Figure 2). The measurement scripts are setup using an easy-to-use Wizard approach. Extensive waveform measurements across all simulation domains are provided. Programming and script writing is not necessary. Once the measurements are setup, they happen automatically when a simulation is finished.

Expanding these traditional SPICE analyses using simulation templates requires single value measurements to be presented for perturbation (sensitivity) analysis. The capability offered by automatic measurement scripts is needed to make these easily applicable to analyses that produce vector data, such as a transient analysis. For example, we can speak of the sensitivity of the output's rise time with respect to the change of a parameter value. The sensitivity of the entire output waveform would be possible to compute, but we couldn't mathematically identify the best output waveform - while we could identify things like the smallest rise time or the greatest standard deviation. These scalar results are needed to make decisions about their relation to parameter values and to use that decision to find parameter values that correspond with a goal; for example, to make a best or worst case result.

Continued on pg. 14

UNITS CONVERSION

SPICE models for nonelectrical devices often use analogs of voltage and current to signify other quantities. To SPICE it's all differential equations, but from the user's viewpoint, different models may communicate in different sets of units. Bidirectional conversion conserves power/energy across its boundary. Electrical engineers encounter this most frequently in transformers and power converters. When we extend IsSpice4 to mechanical systems, the bidirectional process is also seen in motors and gears. Using bidirectional conversion to convert from one system of units to another, for example, converting rad/sec to RPM has the advantage of being able to initialize a system in a more familiar regime.

The subcircuit in Figure 3 solves the unit conversion problem by using a transformer and a smooth transitioning source to "slowly" establish the initial condition. The cosine shaped transition (see Intusoft Newsletter #53) limits the acceleration and jerk (first and second derivatives).

```
.Subckt rad2deg 1 2 ; Bidirectional conversion Radians, Degrees
* Nodes : 1 : input node (radians)  2 : output node (degrees)
x1 11 0 2 0 xfmr {ratio=57.29577951} ; gain = 180/pi
r1 2 0 1meg
rin 1 11 1u
.ends
.subckt kwh2tons 1 2 ; Bidirectional conversion kw-hr, tons tnt
* Nodes : 1 : input node  2 : output node
x1 11 0 2 0 xfmr {ratio=845.505455u}
rin 1 11 1u
rout2 2 0 1meg
.ends
```

Figure 3. Two examples of bidirectional unit converters. The ratio value controls the parameter relationship.

BONUS STUFF ON THE WEB

Intusoft offers free software utilities and useful downloads on its web site (www.intusoft.com). A new Units Converter/Calculator and an IBIS-to-SPICE converter are now posted; other free items (SPICE evaluation versions, models, technical articles, wallpaper and application notes) are also on the site.

The Units Calculator is a novel program that converts between various types of quantities, i.e. distance (inch, foot yard, mile, meters), energy (erg, joule, ft-pound), force (dyne, newton, pound), and pressure (millibar, bar, pascal, mm Hg). It also evaluates sequences of mathematical expressions. It is programmable - users may add their own unit conversions and constants. The calculator supports trigonometric, transcendental, and many common math operations.

FREE IBIS to SPICE Converter

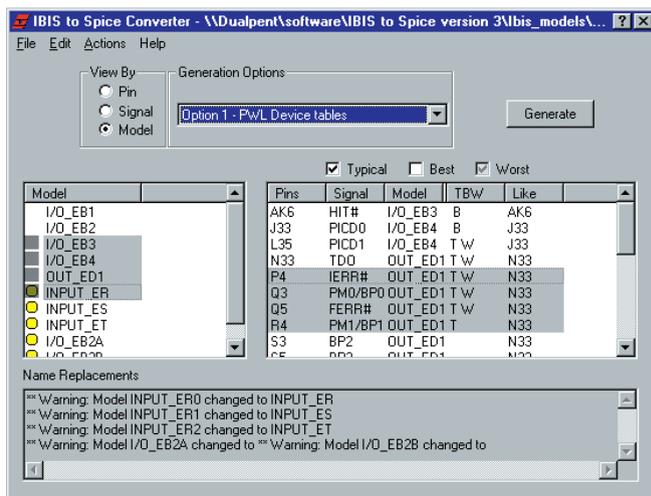
The IBIS-to-SPICE converter translates IBIS (I/O Buffer Information Specification) data sheet information into SPICE models and supports IBIS versions 1.1, 2.1, and 3.0. The converter can be used with ANY vendor's SPICE program including IsSpice4, Pspice® and Hspice®.

The converter parses the IBIS data sheet and extracts the appropriate data tables, pin parasitics, package parasitics, and ramp time information and merges the data into a model template file, yielding a SPICE subcircuit. The default model template that is provided is for Intusoft's IsSpice4 syntax; templates can easily be constructed for other simulators in order to accommodate differences in behavioral element syntax. Instructions are included in the on-line help file. The converter in Figure 4 generates models for the multiple pins and signals that may be included in a single IBIS data sheet, and can also extract typical, best, or worse case models.

The "Model Template" file describes the structure of the SPICE subcircuit that the converter will produce. The format is flexible, allowing you to produce syntax for virtually any simulator and for any subcircuit topology.

IBIS is a standard for electronic behavioral specification of integrated circuit input/output analog characteristics. It includes V/I data so as not to disclose any circuit/process information. It can be thought of as a behavioral modeling specification suitable for transmission line simulation of digital systems, and is applicable to most digital components. IBIS

Figure 4, The free IBIS-to-SPICE converter posted on the Intusoft web site generates SPICE subcircuit models directly from IBIS data sheets.



Continued on pg. 13

DESIGNING A FLYBACK TRANSFORMER

Magnetics Designer is a versatile tool that designs all types of layer wound transformers and inductors. It is unique in the marketplace. The frequency range of the magnetics that can be designed extends from DC to over 5MHz. Unlike Finite Element Analysis (FEA) programs, Magnetics Designer is a synthesis tool as well as an analysis tool. With FEA based tools you must enter far more information about the design than you usually have, especially if you are in the initial design stages. In addition, the FEA analysis times can be quite lengthy, possibly hours. This negates the interactive nature of the design process. Magnetics Designer, on the other hand, operates virtually instantly allowing far more optimizations and “what-if” scenarios to be explored.

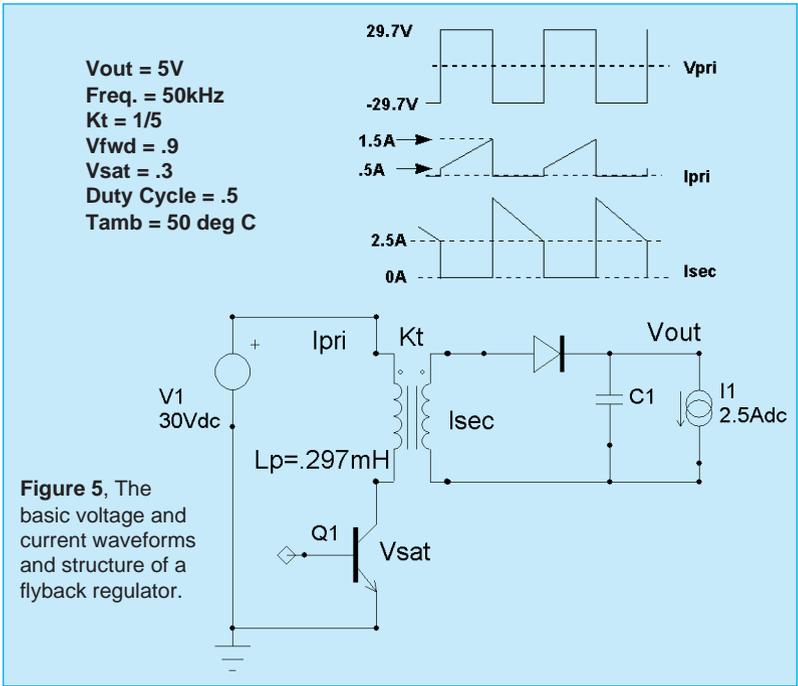
With Magnetics Designer all you need to design a transformer is the operating frequency, temperature rise, core type, and the desired voltage and current for each winding. Magnetics Designer does the rest through its sophisticated synthesis algorithms. The result is a fully designed and characterized magnetic part which meets all of your specifications.

Here we will take a look at how Magnetics Designer can be used to design a flyback transformer. Although it's called a transformer, the principal magnetic component in a flyback regulator is actually an inductor. It is best described as an inductor because energy is intentionally stored in the core or core gap. Therefore, we will use the inductor design features of Magnetics Designer to create a flyback transformer.

The first step is to determine the specifications. A flyback regulator is designed to operate in the discontinuous mode. That is, the flux in the flyback transformer (inductor) returns to 0 on every cycle, as evidenced by the fact that the current on both the primary and secondary are equal to zero during a part of the switching cycle. During the “on” time of the transistor, energy is stored in the inductor. During the “off” time of the transistor, energy is released to the output.

The required parameters for the (inductor) flyback design are: core type and material, operating frequency, E_{dt} (volt-seconds), primary and secondary (AC and DC) currents, required inductance, and the peak primary current.

The type of core we will use is the EI Ferrite from TDK. Magnetics Designer includes a database of over 7,000 cores and dozens of materials including many new exotic materials.



The Edt across the primary winding is found from the equation:

$$Edt = \frac{DV_{pk}}{f_{sw}} = \frac{(.5)(29.7)}{50kHz} = 2.97 \times 10^{-4} V - sec$$

Although the primary winding peak current (see Figure 5) is 1.5 Apk, the switching regulator usually has a current limit which is 10 to 20% greater than the maximum load condition. Therefore, we choose $I_{pk} = 1.8 \text{ Apk}$. Since the turns ratio is 1/5, the peak current reflected to the secondary is 9 Apk. The average (DC component of current) on the primary is:

$$I_{dc}(pri) = \frac{D(I_{pk} + I_{min})}{2} = \frac{.5(.5 + 1.5)}{2} = .5A_{dc}$$

On the secondary, the average current is

$$I_{dc}(sec) = \frac{(1 - D)(I_{pk} + I_{min})}{2} = \frac{.5(7.5 + 2.5)}{2} = 2.5A_{dc}$$

It can be shown that the RMS value of a trapezoidal waveform is found from the following equation:

$$I_{rms} = \sqrt{D \left[(I_{pk} \cdot I_{min}) + \frac{1}{3} (I_{pk} - I_{min})^2 \right]}$$

Likewise, using principles derived from Fourier analysis,

$$I^2_{rms} = I^2_{dc} + I^2_{ac}$$

where I_{ac} is the RMS value of the AC component of current (i.e. the RMS value of $I(t) - I_{dc}$).

Using the I_{rms} equations above, the RMS current for the primary and secondary windings is calculated to be .746 and 3.68 Arms, respectively. Solving for I_{ac} , the AC currents were calculated to be .54 and 2.7 Arms, respectively.

The inductance on the primary was originally specified as .297 mH. Also, the turns ratio was defined to be 1/5. Therefore, the inductance on the secondary is 11.9 μ H.

The requirements were entered on the Inductor screen (Figure 6). The ambient temperature was set to 50°C in the Options tab. The maximum allowed temperature rise is set 30 degrees C. Litz wire was chosen due to the high operating frequency.

After the Apply button is clicked, the program performs thousands of calculations. It tries different stranding arrangements, different wire sizes, and so forth, in an effort to generate a design with the minimum power losses in the smallest core possible to still meet the fill and temperature rise specifications. Magnetics Designer provides a list of the cores it tried, along with their respective fills and temperature rises. The last geometry is the one that the program finally settled

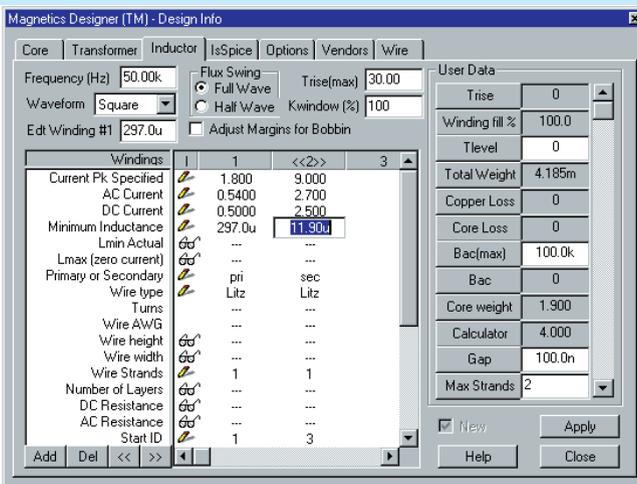
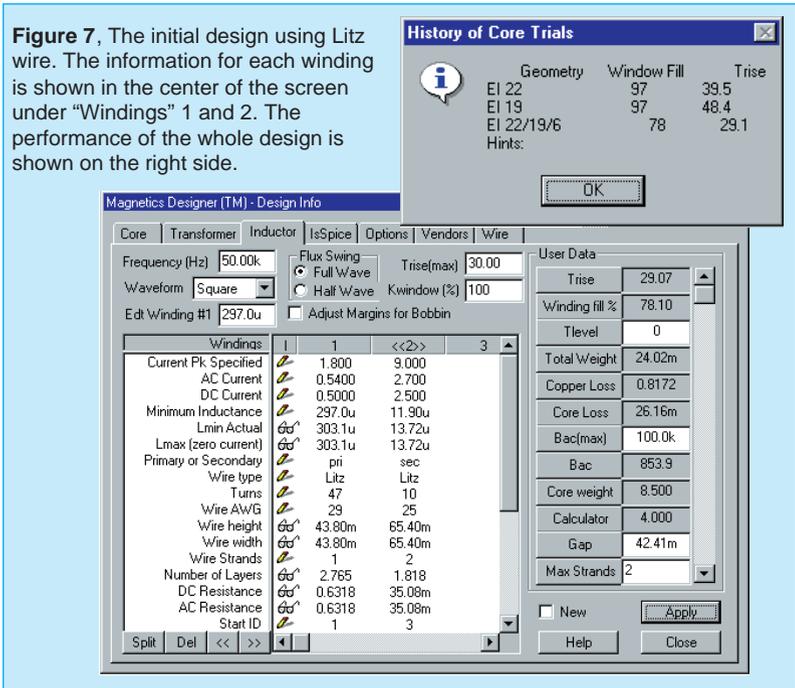


Figure 6, The inductor screen with frequency, Edt, maximum allowable temperature rise, current peak, AC/DC current, and minimum inductance entered. The Apply button generates the rest of the flyback design.

Figure 7, The initial design using Litz wire. The information for each winding is shown in the center of the screen under “Windings” 1 and 2. The performance of the whole design is shown on the right side.



upon. The total time to calculate the full design specifications on a Pentium 133 is under 1 second! The initial design, shown in Figure 7, generally met the design requirements with the exception that the turns ratio was not exactly 1/5.

The window fill algorithm calculates the build assuming that two windings will not share space on a single layer. That is, if a winding has 1.1 layers, the build algorithm calculates the height of that winding as if it consumed two full layers. Since the primary winding occupies 2.765 layers and the secondary occupies 1.818 layers, the window is not utilized to its fullest efficiency.

Turns are always rounded to the nearest integer by the program. Therefore, some slight adjustments are required to achieve exact turns ratios. For example, if the program calculates that 46.5 turns are required on the primary and 9.6 turns are required on the secondary, the primary will have 47 turns and the secondary will have 10 turns. The designer would then have to adjust the turns on the primary to achieve an exact turns ratio of 5/1.

Several design changes were tried in order to improve the inductor design. It was originally assumed that Litz wire was

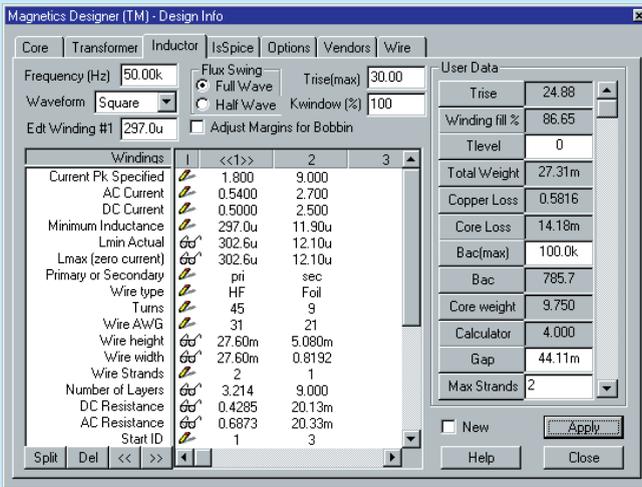


Figure 8, The flyback design using heavy formvar (HF) and foil windings. Here, the Windings window shows the turns, wire size, strands, layers, and AC/DC resistance. The User Data fields show the temperature rise, fill, weight, losses, and flux density.

necessary to avoid excessive losses due to potential high frequency AC resistance problems (skin and proximity effects). To test this assumption, heavy formvar (HF) wire was substituted for the Litz wire. This resulted in a significant temperature rise increase. After closely reviewing the results, it was determined that this problem was primarily due to increased AC resistance on the secondary side. Therefore, foil windings were substituted for the magnet wire on the

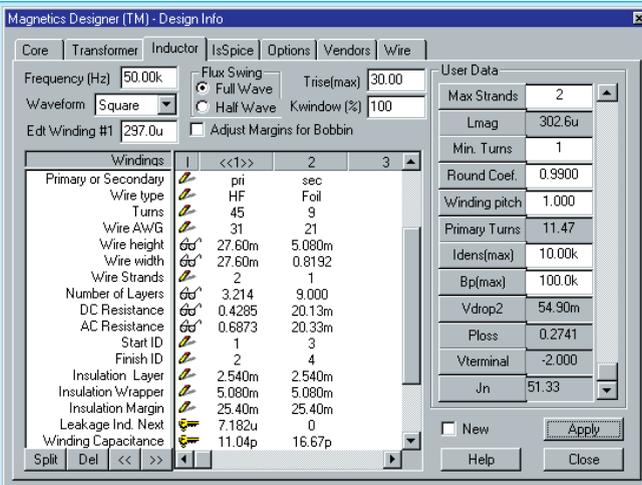
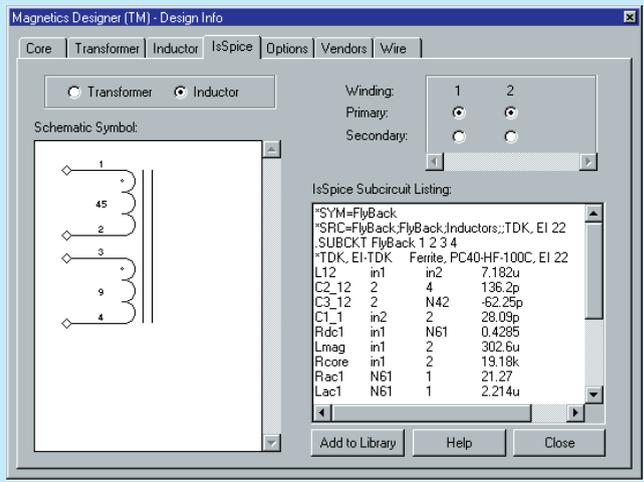


Figure 9, Both the Winding and User Data fields scroll, revealing more information such as leakage inductance, winding capacitance, weight per winding, magnetizing inductance and more.

Figure 10, A SPICE model (SPICE 2G syntax) can be produced, along with a reconfigurable schematic symbol.



secondary. Foil windings typically have very low AC resistance since the thickness required to achieve the desired cross-sectional area is usually much thinner than the skin depth of corresponding wire. Since a foil winding was used, the program adds layer insulation, wrappers, and end margins to prevent shorts between turns and other windings.

Next, the number of turns on the primary was set to 45 so that the turns ratio from primary to secondary would be exactly 5/1. To maintain the inductance at approximately .3 mh, the Gap was increased to .044 cm. The result of these changes are shown in Figure 8. The final design meeting all specifications is shown in Figures 8 and 9. It has a slightly lower peak flux, lower temperature rise, lower AC and DC resistance, and uses a smaller geometry than the initial design in Figure 7.

At this point, you can explore the design further by splitting windings or trying different materials. A winding sheet and design summary report can be printed out or copied to Microsoft Excel. As shown in Figure 10, Magnetics Designer can also produce a SPICE model and schematic symbol for the design. This allows you to simulate the device while it is being made. The SPICE model can be used with ANY SPICE program.

Magnetics Designer provides a simple and fast way to design custom transformers and inductors, a key part of virtually all electronic systems and many EMI filtering applications. The software even designs planar magnetics, an important part of many new high frequency applications. Magnetic design is an art that is slowly being lost as experienced designers retire. Products like Magnetics Designer help fill the void by providing intelligence and design expertise in a software tool.

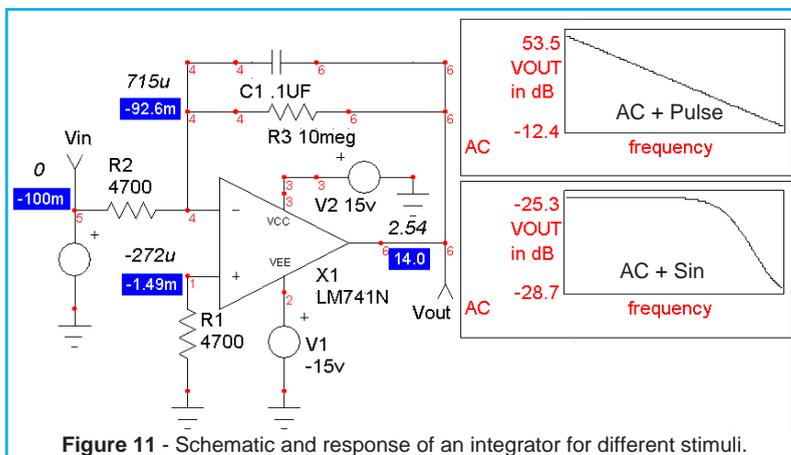
AC CIRCUIT BIAS PROBLEMS

From: Jeffrey P. San Jose, CA

Dear Intusoft Tech Support: I was tutoring a friend in basic EE and showing him how an integrator works. Using SpiceNet, I drew a simple op-amp circuit using a 741 as an integrator, and was demonstrating the effect of a resistor in the feedback path shunting the integrating capacitor. By increasing its value, I could effectively eliminate it without deleting it. I found that if I made the input source a combination of a simple AC (frequency stimulus) source plus a transient PULSE source (Pulse -100m 100m 0 1m 1m...) and I was only looking at the AC response (AC Bode plot), I would get a completely erroneous result. When I made the input source AC only, the result was correct. In addition, I tried the AC in combination with a Sin transient generator and the circuit worked okay.

Why does the AC + Pulse source give bad results?

The answer is simple, but may not be obvious. When the AC + Sin source is used, the DC value of the Sin source (Sin 0 ...) is zero because the DC offset (initial value of the source) is zero. In the AC+Pulse combination, the DC value is -100m (the initial value of the pulse source) so the op-amp saturates (Figure 11). You can see this if we look at the operating point solution. In fact, this is the correct answer for the given bias condition. But that's why the AC+Pulse simulation looks different. The AC analysis solution is affected by the biasing of the circuit. If you put an additional DC=0V value in the AC+Pulse source, then IsSpice4 will use the DC value instead of the initial pulse value for the operating bias point calculation of the circuit. The result is then the same as the AC+Sin simulation.



FREE IBIS to SPICE Converter

*Continued
from pg. 5*

Version 1.0 was first introduced in June, 1993. While IBIS Version 2.1 remains the official standard, IBIS Version 3.0 containing some technical extensions was ratified on June 12, 1997 by the EIA/IBIS Open Forum. Work will continue on refining this version for eventual submission for national and international standardization consideration. Updates to the IBIS-to-SPICE converter will be posted as necessary.

Official IBIS Related Web Sites

EDA Industry Working Groups - www.eda.org/ibis/

IBIS Directories and Files - <ftp://ftp.eda.org/pub/ibis/>

Official EIA/IBIS Home Page - www.eia.org/eig/ibis/ibis.htm

RAIL Directories and Files of a related development activity - <ftp://ftp.eda.org/pub/rail/>

An additional bonus article, "A Mercury Arc Lamp Model", extra SPICE models, and links to more details about ICAP/4 8.x.6 are available on Intusoft's web site.

ICAP 7.x, 8.1.0 & DOS Users Save A Bundle

*Continued
from pg. 1*

All of the new version 8 features can be yours. Version 8 is required if you are using the Windows 95, 98, or NT operating systems. In addition, the price includes maintenance for **one full year** from the purchase date. **The reduced prices give you a savings of between 25% and 30%.**

And if you own version 8.1.0 of ICAP/4Windows or ICAP/4Rx, you can also save over 25% when you upgrade to the latest version. This offer will no longer be available after Dec. 31, 1998, so take advantage today! For information on prices and other upgrades, please contact Intusoft or your local Intusoft dealer.

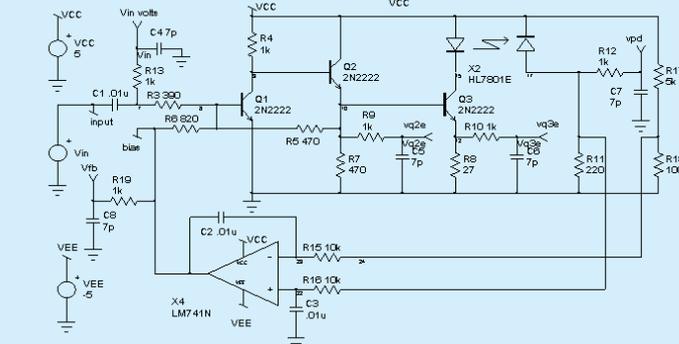
ICAP/4Rx For Sale On The Web

You can now purchase and download ICAP/4Rx from the Web. By checking into 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 put aside that evaluation version or powerless workbench tool and step up to professional simulation power. ICAP/4Rx is both easy-to-use and powerful, and now it's available instantly.

New ICAP/4 8.x.6 Features: RSS & EVA

Figure 12, A simple circuit and the output from an RSS (root square sum) analysis.



**** RSS Analysis data for V(10)

COMPONENT	PARAMETER	NOMINAL	TOLERANCE	SENSITIVITY	CONTRIBUTION
r3	resistance	3.90e+002	2.00e-002	0.00e+000	0.00e+000
q1	bf	1.05e+002	3.50e-001	1.28e-026	4.48e-027
q1	is	1.52e-014	2.00e-001	3.26e+002	6.52e+001
r4	resistance	1.00e+003	2.00e-002	-7.76e-005	-1.55e-006
q2	bf	1.05e+002	3.50e-001	6.93e-004	2.43e-004
q2	is	1.52e-014	2.00e-001	9.26e+011	1.85e+011
r5	resistance	4.70e+002	2.00e-002	1.71e-005	3.41e-007
r6	resistance	8.20e+002	2.00e-002	1.71e-005	3.41e-007
q3	bf	1.05e+002	3.50e-001	6.55e-006	2.29e-006
q3	is	1.52e-014	2.00e-001	-8.83e+011	-1.77e+011
r7	resistance	4.70e+002	2.00e-002	2.96e-005	5.93e-007
r8	resistance	2.70e+001	2.00e-002	1.74e-003	3.48e-005
VECTOR	OP-VALUE	RSS-VALUE	TOLERANCE%	HI-VALUE	LO-VALUE
V(10)	1.11e+000	2.58e-002	2.33e+000	1.13e+000	1.08e+000

*Continued
from pg. 3*

The ability to measure key performance behavior and use the measured results within other analyses such as stress and sensitivity provides a powerful means to automate performance analysis and design optimization.

Tight integration of the Simulation Templates into the IsSpice4 mixed-signal simulator greatly increases processing speed because the simulator is not exited or reloaded, and netlists don't need to be re-compiled. You can automatically run multiple simulations in either batch or interactive mode while varying toleranced parameters at each pass. The results are automatically extracted and displayed against user-defined limits. "Stress alarms" are automatically displayed when a performance parameter is out of tolerance.

Figure 12 shows an example amplifier circuit and the basic output RSS data in tabular form. The data shows the individual contributions from each component and model parameter as well as the final RSS value at node 10. Output from the EVA, sensitivity, and worst case analyses is similar.

The Design Validator™ feature, which has been added to ICAP/4Windows, keeps track of multiple circuit configurations, analysis setups and test measurements. It then runs an entire

suite of tests automatically and creates a report that summarizes the measurements and pass-fail grades (Figure 2). This powerful capability extends SPICE well beyond its current single simulation oriented boundaries. The measurements can be made on any number of nodes, currents or devices. This is all done without the need for any user interaction.

The feature additions in particular, the design validation and Simulation Templates, put ICAP/4Windows in a class by itself. No other SPICE tool has this feature set at any price. For more details on version 8.x.6, visit www.intusoft.com.

WHAT ABOUT LINUX?

In the last few years Linux has become a viable platform for both client and server based computing. IsSpice4 is available on Unix under Solaris as the ViewAnalog package from Viewlogic. Intusoft would like to know your opinion on Linux. Please write or email Intusoft (info@intusoft.com) if you would like to see IsSpice4 and possibly ICAP/4Windows ported to Linux.

The Biggest & Best SPICE Libraries

What vendor has the most model types? What vendor has published 3 books on SPICE and modeling? What vendor has the only Windows based AHDL language? What vendor provides their own SPICE training? What vendor has the most behavioral modeling features? What vendor has a FREE modeling service for its customers?

No Contest - The only answer is Intusoft!

No SPICE Vendor can match the variety and quality of Intusoft.

- Total Library Size **Now Over 14,000 parts**, hundreds of model types
- User Defined C Code Models for analog, digital, array, and sampled-data elements
- AHDL Models using a Windows C based language
- Power Library (500+ models from Unitrode, Cherry, Linear Tech, Siliconix)
- RF Device Library (600+ models from H-P, NEC, Philips, Motorola)
- Vendor supplied IC/Op-Amp Models (1700 models from Analog Devices, TI, APEX, etc.)