

Intusoft Newsletter

Personal Computer Circuit & System Design Tools



Copyright © Intusoft, All Rights Reserved

Issue #46 May 1996

Tel. (310) 833-0710

Fax (310) 833-9658

New Magnetix Designer

Intusoft has released a new software program for transformer and inductor design. The program is called Magnetix Designer. **Magnetix Designer synthesizes and analyzes various types of transformers, inductors, and chokes.** It is optimized to design layer wound magnetix using cores that have rectangular or circular winding areas. Typical designs may include ferrite pot cores, EI, EE, EP, cut C, and RM cores, toroids, and EI laminated cores. Magnetix Designer is capable of designing single phase multi-winding transformers and inductors operating from 10 to well above 1MHz. Typical applications include 60Hz line transformers, switching regulator transformers, and output filter inductors.

Continued on pg. 2

In This Issue

2 New Magnetix Designer Released

5 Generating SPICE Models For Magnetix

8 New IsSPICE4 Has New BSIM3v3 MOS Model

10 Pspice Abandons Users Again

11 Modeling Mechatronics: Synchro Modeling

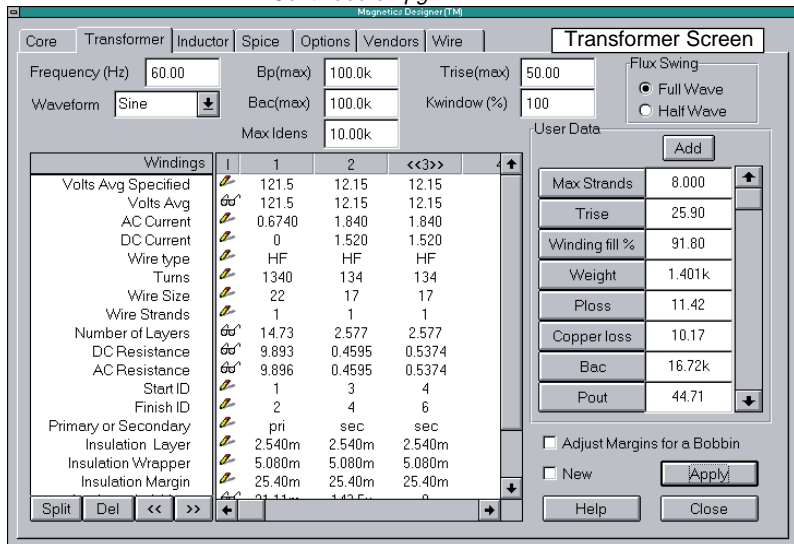


Figure 1, The new Magnetix Designer program makes transformer and inductor design easy. A demo and FAQ are available at <http://www.intusoft.com>.

New Magnetics Design Software

A database with thousands of cores and a wide variety of materials is included with the program. They are accessed through the Core screen shown in Figure 2. Vendors like TDK, Magnetics, Philips, Thomson, Micrometals, and Ferrite International are represented. However, the database is extensible. **You can add your own core and material information using a supplied Excel spreadsheet template.**

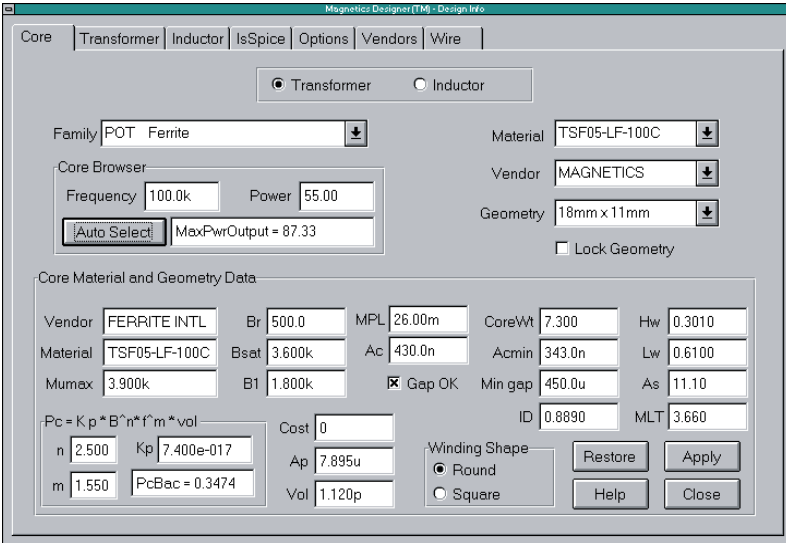


Figure 2, The Core Screen allows you to browse through and change the extensive core database included with Magnetics Designer.

Magnetics Designer allows you to choose (Figure 1) from a wide variety of wire types: **heavy formvar**, **small formvar**, **square**, **double-square** and **Litz** magnet wire (European wire sizes are included); **pcb traces** specified by the copper thickness, minimum trace width, minimum trace spacing, and PWB thickness and **foil** specified in various thicknesses. You can create your own wire tables with different wire sizes. For example, the square wire table can be edited to describe virtually any rectangular wire format, or you could describe a table with a sparse set of gauges (i.e., even gauges only).

Three insulation options allow the user to specify layer insulation thickness, wrapper thickness, and end margin lengths. If there are multilayer foil windings, layer insulation may be included between each layer of foil. There is also an option to account for the margin required for inner layer windings to exit the bobbin.

How Magnetics Designer Works

To start a transformer design, you first select the core family and material on the core screen (Figure 2). Then the operating frequency and electrical specifications (voltage and current per winding) are entered on the transformer screen (Figure 1). You may optionally enter one or more design constraints like temperature rise, peak flux density, and/or window fill.

Magnetics Designer will optimize the design using a powerful set of state-of-the-art algorithms. The program will select the core geometry, turns ratio, wire sizes, and number of strands to achieve your design goals, all while minimizing the total power dissipation for a given temperature rise in the smallest possible core.

Once finished, Magnetics Designer will provide a summary of the cores it tried to optimize, and suggestions on how to possibly improve the design if the design constraints were not met. You may browse through the core database and choose an alternate core geometry. If desired, you can lock your choice and optimize the design around your selection.

As shown in the transformer screen, you can manipulate over 20 parameters such as the wire size and type, number of strands, gap length, insulation thickness and more. Magnetics Designer will predict characteristics like peak flux density, AC flux density, DC and high frequency AC resistance, core and copper losses, leakage inductance, winding capacitance, weight, efficiency, temperature rise, and window fill %.

Program Sports Unique Options

Magnetics Designer has several unique features. The program exposes the mathematical equations and algorithms it uses to calculate the transformer or inductor performance (Figure 3). For example, you can access a parameter such as temperature rise or leakage inductance and view the equation used to

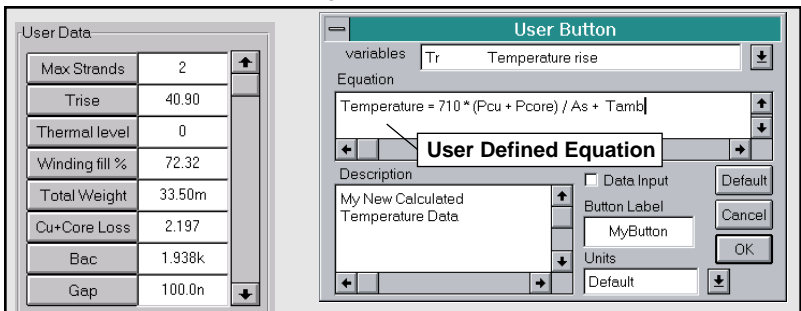


Figure 3, Magnetics Designer outputs a variety of data including temperature rise, weight, and fill in the “User Data” Fields (left & Figure 1). The User Button dialog allows you to edit the equation(s) which defines each field.

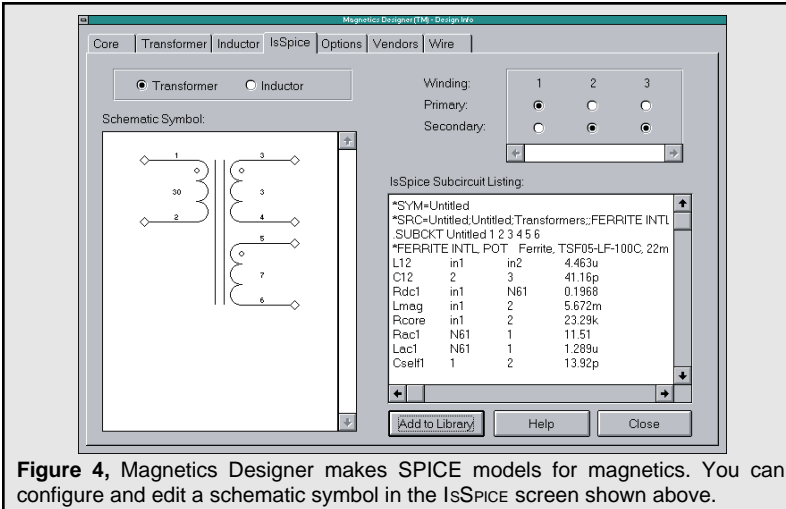


Figure 4, Magnetics Designer makes SPICE models for magnetics. You can configure and edit a schematic symbol in the IsSPICE screen shown above.

calculate its value. You can then substitute one or more expressions in place of the default equation. Equations can be set up using data from the program to derive your own performance merits. This feature allows Magnetics Designer to be customized for each application or design.

Another unique function is the SPICE modeling capability. Magnetics Designer produces a Berkeley SPICE model of your transformer or inductor. The program also allows you to produce and configure (Figure 4) schematic symbols. The symbols can be used with schematic entry packages from OrCAD (CAPTURE[®]), Protel (Schematic³) or Intusoft (SPICENET).

Within the general constraints described above, you can see that Magnetics Designer has a great deal of flexibility and can

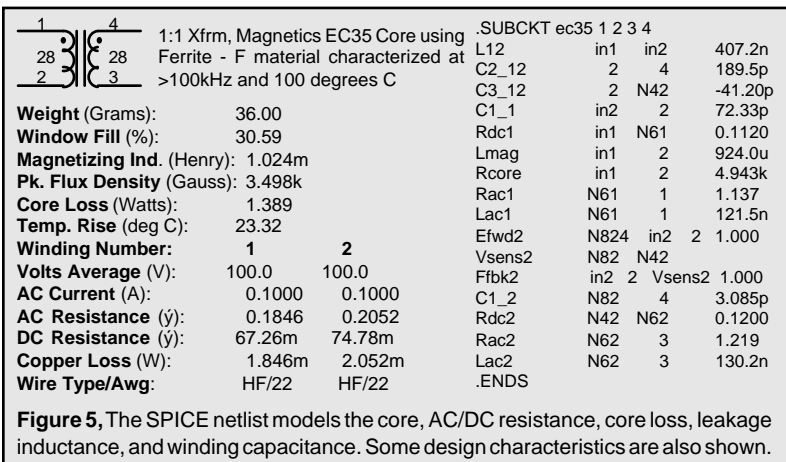


Figure 5, The SPICE netlist models the core, AC/DC resistance, core loss, leakage inductance, and winding capacitance. Some design characteristics are also shown.

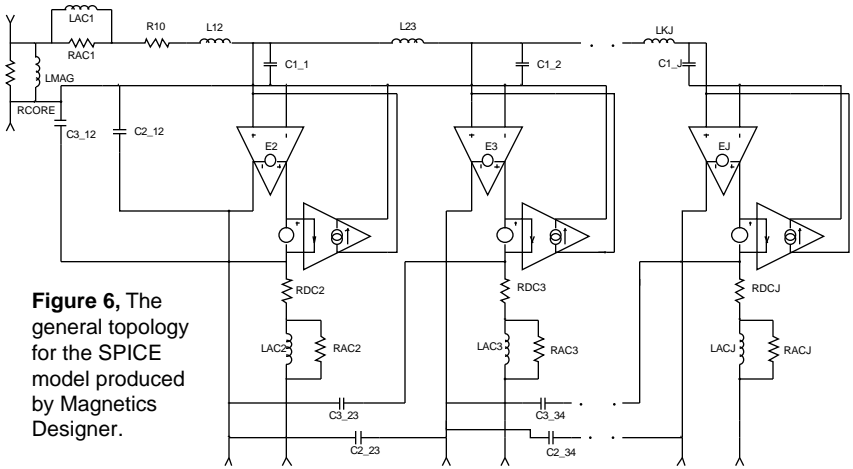


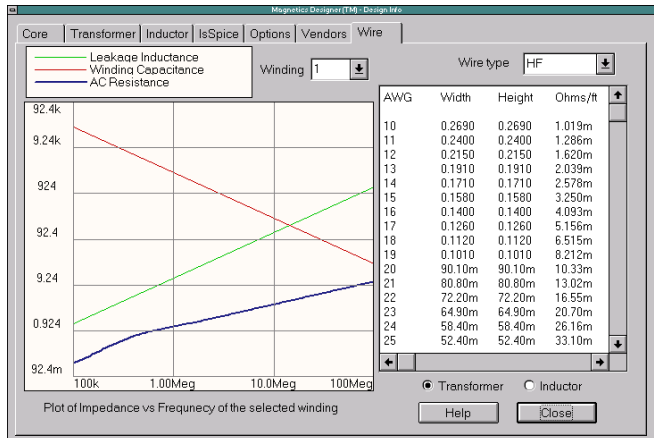
Figure 6, The general topology for the SPICE model produced by Magnetics Designer.

handle many different winding configurations, wire types, and core configurations.

How Magnetic Models Are Generated

The complete SPICE model for a layered winding is shown in Figure 6. Notice that the AC resistance is approximated using a single L-C parallel section. The impedance is fit to the AC resistance curve in the neighborhood of resonance. A plot of AC resistance is shown in Figure 7. For all cases we have observed, the leakage inductance produces a much higher impedance than the AC resistance. This leads us to conclude that the effect of the AC resistance will be to damp the resonance of the transformer, hence the importance of making the model accurate in the neighborhood of the expected resonance. The resonant frequency using leakage inductance (L12, etc.) and winding capacitance represents a theoretical maximum. We reduce this frequency slightly by summing the

Figure 7, The wire screen provides insight into the expected losses due to harmonics with a plot of impedance vs. frequency for each winding.



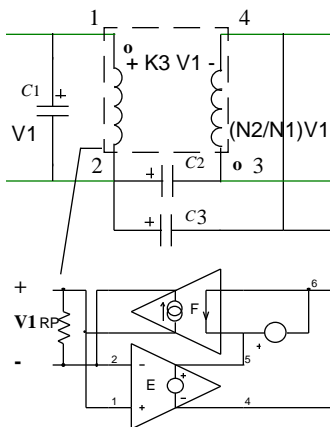


Figure 8, Capacitance and Ideal transformer topologies.

previous leakage inductance and inter-winding capacitance in order to align our AC resistance calculation more closely with expected circuit operation.

Three capacitors (Figure 8) are sufficient to account for the switching energy for different interconnection schemes. Certain turns ratios and winding directions result in negative capacitor values. IsSPICE handles this properly, as long as the negative capacitance does not “see” a real valued resistor. Therefore, the model topology is restricted to placing the inter-winding capacitances “inside” of the winding resist-

ance and leakage inductance models. As with most power electronics simulations, Gear integration works best. IsSPICE users can switch to Gear integration by issuing the .Options parameter Method=Gear. Many other SPICE programs, such as Pspice and Electronic Workbench, do not let you force Gear integration, therefore, greatly limiting their effectiveness for power electronics simulation.

The ideal transformer accounts for the forward voltage transfer and reverse current transfer, and is modeled using a pair of IsSPICE controlled sources (E & F). In the forward direction, the “primary” voltage is transferred to the secondary using a voltage controlled voltage source with its gain set to the turns ratio. In the reverse direction, a voltage controlled current source assures that current is reflected from the secondary to the primary by the turns ratio. The voltage source after the first winding is used to sense current for the feedback term in the ideal transformer model. The DC and AC resistance equivalent circuits are each attached to their respective windings.

The core is modeled as a simple resistor in parallel with the magnetizing inductance. The resistance is set by the core loss equations at the operating frequency. A linear core model is used. In the June 1995 *Intusoft Newsletter*, we illustrated techniques for making a nonlinear saturable core model which can be added. The eddy current break is not sufficiently described in the program’s core database, so its effect isn’t modeled. This effect can be modeled by adding a series R-C network across the magnetizing inductance, set to match manufacturer’s data. This could add extra damping to the ringing observed when switching devices turn off while still conducting current. Modeling added effects requires considerable computational resources, and shouldn’t be done unless you depend on the nonlinearities for an aspect of your design.



The Magnetics Designer software was originally developed by **Analytic Artistry**, a leader in the field of nonlinear magnetics design software. In



1995, Intusoft acquired the rights to the "Transformer and Inductor Spreadsheets", two separate DOS based programs. A large number of changes have been made, including combining of the DOS programs into one unified solution. Intusoft is now the sole owner and developer of the Magnetics Designer software. A brief list of enhancements over the DOS version is shown next:

- The core selection and transformer/inductor design algorithms have been changed. Magnetics Designer now automatically tries the suggestions previously shown in the documentation. You can switch cores, families, geometries and materials at any time and lock your choice to play "what-if" investigations.
- IS_{SPICE} models and schematic symbols are automatically generated.
- Planar Magnetics (Printed circuit board cores and wire traces) are included.
- You can define your own core materials, geometries, and wire tables. Wire tables may have sparse gauges, i.e., you can have an even gauge table.
- Leakage inductance and winding capacitance values are calculated.
- The split winding feature is automatic. You can clone and move windings around and the proximity fields are recalculated based on the interleaved configuration.
- AC resistance is computed for square wave harmonics (up to the 100th harmonic) or for a pure sine wave.
- You can set the minimum number of turns for the winding with the smallest turn count. You can control the accuracy of the secondary voltage outputs.
- Three thermal models are supported. The thermal model allows user entry of the convection coefficient as well as calculation of the hotspot temperature in the winding stack, or core, based upon heat flow through the windings or insulation to the core. Other models may also be implemented.
- The program supports alternate sets of units and European wire sizes.

- Magnetics Designer is compatible with Windows 3.1x, 95, or NT.

Documentation, Pricing, and Availability

A detailed applications manual is included with the program. It provides information for the engineer who wants to learn more about magnetics design. It describes the design equations and algorithms, discusses core materials and geometries, and presents detailed examples. Virtually all of the program's documentation is also available on-line.

As an Introductory Offer, Magnetics Designer is SPECIALLY priced until June 28, 1996. After that date, the price will increase. More information, including an application note and a demonstration version of Magnetics Designer, is available on the Intusoft Web Site <http://www.intusoft.com> and CompuServe (CADD/CAM/CAE Vendor forum, Library 21).

IsSPICE4 Gets New BSIM3 Model

By Farhad Aryom

MOS IC technology has made tremendous progress in recent years. The desire to increase circuit speed and reduce circuit area has continually reduced the feature size of MOS ICs. As size decreases, short channel effects and high field effects, which did not exist in large devices, become more and more significant, causing device characteristics to deviate substantially from long channel device behavior. In 1983, the Berkeley Short-Channel IGFET Model (BSIM) version 1 was developed. BSIM1 works reasonably well for devices with channel lengths of $1.0\mu\text{m}$ or larger. In 1989, an enhanced version, BSIM2, was developed. BSIM2 can fit devices down to $0.25\mu\text{m}$ for both current, as well as small signal parameters such as output resistance.

The parameter extraction strategy used for BSIM1 and BSIM2 is relatively simple and accurate. But it introduces some inherent problems. First, a single process file does not work well when a wide range of device dimensions are used. Multiple process files are needed to obtain good accuracy over the full range of device dimensions. Second, the existence of redundant parameters makes statistical circuit performance prediction difficult, if not impossible. BSIM3v3 is the latest deep-submicron MOSFET model from the University of California at Berkeley, and it is designed to address these problems.

BSIM3 is a physical model and is based on a coherent quasi two-dimensional analysis of the MOSFET device structure, taking into account the effects of device geometry and process parameters. In other words, dependencies of important geometry and process parameters such as channel length, channel width, gate oxide thickness, junction depth, substrate doping concentration, etc. are built into the model. BSIM3 allows users to accurately model MOSFET behavior over a wide range of existing technologies and predict the behavior for future technologies.

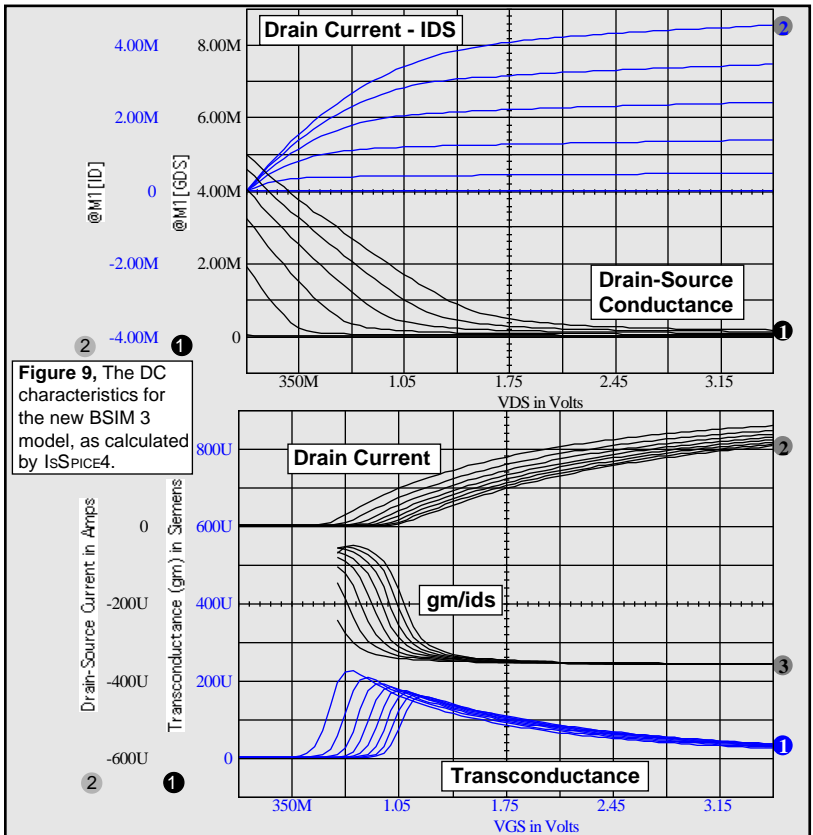
The BSIM3v3 (version 3) model has been extensively modified from previous releases and is the recommended version. Some of the modifications that are not found in version 2 are:

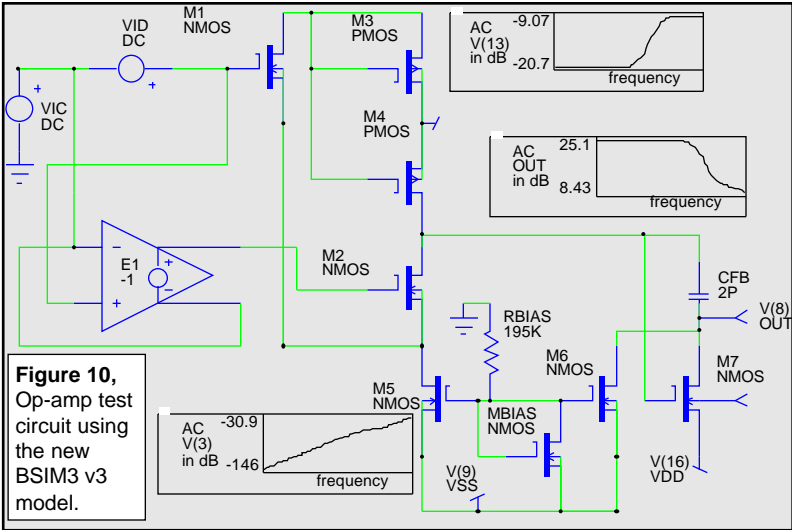
- A single I-V expression is used to model current and output conductance characteristics for subthreshold, strong inversion, linear, and saturation regions. This formulation guarantees continuity for **I_{ds}**, **G_{ds}**, **G_m** and their derivatives for all **V_{gs}** and **V_{ds}** bias conditions.

- There are new width dependencies for bulk charge and source/drain resistance, **Rds**. This greatly enhances the accuracy of the model for narrow width devices.
- **dw** and **dl** dependencies are available for different W_{drawn} and L_{drawn} devices. This improves the model's ability to fit a variety of **W/L** ratios with a single set of parameters.
- New capacitance equations improve the modeling of short and narrow geometry devices.
- New relaxation time model for characterizing the non-quasi-static effect of MOS circuits for improved transient behavior.

Simulation Characteristics

The various DC characteristics of a typical short channel BSIM3 model are shown in Figure 9. The op-amp circuit (Figure 10) uses a BSIM3 model consisting of all the default parameter values. The frequency response for this circuit is shown below. A transient simulation of a one-bit comparator, along with the previous simulations, is also included on the *Intusoft Newsletter* subscriber's disk, in addition to several other test circuits.





To obtain test circuits and more documentation about BSIM3V3 and join the BSIM3 users group mailing list, you can contact the BSIM3 homepage at: <http://rely.EECS.Berkeley.EDU:8080/bsim3www/>. **The new BSIM3v3 model is included in the new IsSPICE4 version 7.6 which is available now.**

BSIM3 Included in New 7.6 Release

In addition to the BSIM3 model, the 7.6 release includes new behavioral modeling constructs and functions, a lossy transmission line with frequency dependence (skin effect/dielectric loss), and now R, L, C, and B elements can have expressions that use frequency, time and temperature. More analog and digital models are included (now at over 8000). The basic price of the ICAP/4Windows package remains the same. Contact Intusoft or your local dealer for information on upgrade pricing for different versions.

Pspice Abandons DOS Users

In keeping with their policy on the Macintosh SPICE product line, Microsim™, the makers of the Pspice®, has stated that they will no longer offer DOS based products. Intusoft welcomes all DOS users who still need support, and pledges not to force you to upgrade to Windows. We have no plans to discontinue our DOS based products.

Intusoft does not force you to purchase maintenance in order to get full trade-up value for your existing software. Our upgrade policy is that upgrades are always the price difference or less!

Modeling Mechatronics

In previous *Intusoft Newsletters* [1], we have frequently addressed how to extend analog simulation into other domains. In the next few issues, we will explore how to model and simulate systems consisting of electrical, mechanical, and hydraulic elements. You might currently be using a mathematical tool like Mathcad or Matlab, or another non-SPICE simulator like Saber to analyze “mechatronic” problems, but as we’ll see, IsSPICE4 solves these problems just as efficiently with the added advantage that analog and digital electronics can be mixed in. Much of this work is adapted from J. Scholliers [2].

SPICE solves sets of nonlinear differential equations, and any problem that can be put into that form can be solved by SPICE. The added features of parametric and behavioral modeling and graphical input make IsSPICE4 the MOST powerful and affordable multitechnical tool on the market today. It should be noted that IsSPICE4 has a full AHDL capability, but in the case of mechatronics, this added power is not required.

Table 1 shows the mapping of “through” and “across” variables as they relate to their electrical counterparts. The energy conservation principle behind the relationships allows you to convert different systems to a single (electrical) domain. By translating the variables in the different domains to the electrical domain (voltage/current) any one or combination of the systems can be simulated with IsSPICE. Table 2 shows what quantities the electrical elements represent. From the user’s point of view, once the subcircuit and symbol have been constructed, the electrical analog remains hidden, and the simulation proceeds from a graphical perspective.

As an example of a mechatronic element, let’s look at a model for a synchro. Synchros operate as rotating transformers, and

	through	across	Power (P)
Electrical	current [A] i	e.m.f. [V] v	$i*v$
Magnetic	flux rate [Wb/s=V] $d\phi/dt$	m.m.f. [A] \mathcal{I}	$\mathcal{I}*d\phi/dt$
Mechanical Translational	force [N] F	velocity [m/s] $v=dx/dt$	$F*v$
Mechanical Rotation	torque [Nm] T	rot. velocity [rad/s] $\omega=d\theta/dt$	$\tau*\omega$
Fluidic	volume flow [m ³ /s] Q	pressure [Pa] p	$Q*p$

Table 1,
Power
variables
in
different
energy
domains.

Table 2, Electrical primitives in different energy domains. The units are S.I. (i.e. kg for mass, meter for length)	Electrical	Element Equation	Mechanical Translational	Mechanical Rotational	Fluidic
	voltage source (V)	$v=f(t)$	velocity source (v)	velocity source (ω)	pressure source (p)
	current source (I)	$i=f(t)$	force source (F)	torque source (T)	flow source (Q)
	resistor (R)	$v=Ri$	damping (b)	rot. damping (b)	fluidic resistance
	capacitor (C)	$i = C dv/dt$	mass (m)	inertia (J)	tank
	inductor (L)	$v = L di/dt$	spring (k)	rot. spring (k)	fluidic inertia

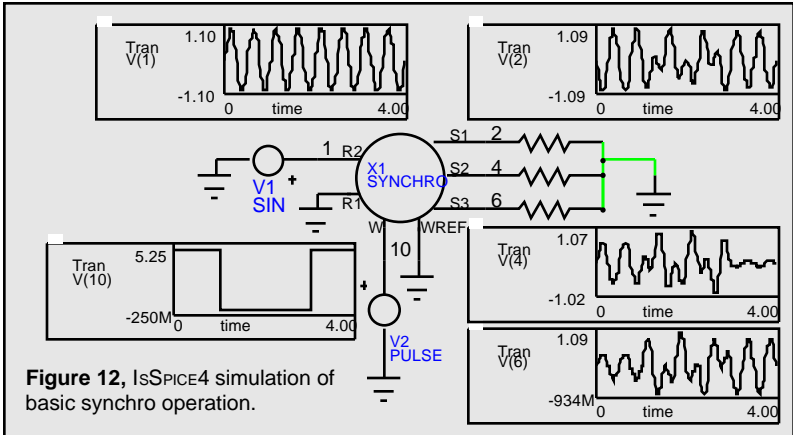
are used in sensitive control applications where a mechanical output is required. The synchro is modeled as three transformers whose gains are dependent upon position. The gain of each transformer is derived from the position of the stator with respect to the rotor, which is equal to $K*\sin(\alpha)$, $K*\sin(\alpha+120)$, $K*\sin(\alpha-120)$ for the synchro (3 windings at 120 degrees).

The rotor is modeled with a general purpose load model consisting of inertia, damping and Coulomb friction. This model

```
* SYNCHRO - Connections: s1,s2, s3: 3 nodes in star on the stator, r1, r2: 2 nodes for the rotor,
* w, wref: mechanical nodes. Parameters: K - gain, x0 - initial angle in radians, Jm - load inertia, b - load drag,
* w - load friction, v0 - initial mass velocity in m/s, d120 - 120 degrees in radians, K - gain

.Subckt Synchro s1 s2 s3 r1 r2 w wref {d120=2.094395}
xload w wref load {Jm=1f b=1f w=1f v0=0}
xint w wref theta posrel {x0=0}
beqsin1 qsin1 0 v=sin(v(theta))
beqsin2 qsin2 0 v=sin(v(theta) + {d120})
beqsin3 qsin3 0 v=sin(v(theta) - {d120})
x1 r1 r2 s1 n qsin1 0 transfov {k=1}
x2 r1 r2 s2 n qsin2 0 transfov {k=1}
x3 r1 r2 s3 n qsin3 0 transfov {k=1}
rn n 0 1T
.Ends
* LOAD (i.e. mass with connection to ground)
* In 101, Out 102, Gravity in m/s^2
.Subckt Load 1 2 {Jm=1 b=1 n w=0 v0=0 g=0}
xm 1 massg {Jm=1f v0=0}
xd 1 2 damper {b=1f}
xw 1 2 frict1 {w=1f}
.Ends
* MASSG: mass with gravity
* Connections: 1 - X-component of velocity/force
.Subckt Massg 1 {Jm=1 u g=-9.81 v0=0}
v 1 101
r 1 1 0 1T
r 101 0 1T
c 101 0 {Jm} ic={v0}
ig 0 1 {Jm*g}
.Ends
* DAMPER : Viscous damping, In 101, Ref 102
.Subckt DAMPER 101 102 {B=1}
r 101 102 {1/B}
r1 101 0 1T
r2 102 0 1T
.Ends
* FRICT: Constant friction force
* 1 2 : nodes between which force has to be added
.Subckt Frict1 1 2 {W=0}
rin 1 2 1T
befric 111 0 V=V(1,2)*{W}^0 < -.1m ? -1 :
+ V(1,2)*{W}^0 > .1m ? 1 : V(1,2)*{W}^0 /1M
bgw 1 2 l = {W} * V(111)
.ENDS
* TRANSFOV: Ideal Transformer with variable ratio
* In-pri 101 102, Outsec 201 202, Control 301 302
.Subckt Transfov 101 102 201 202 301 302
+ {K=1 effi=1.00 effv=1.00}
rin 101 111 1n
be 205 202 v={K*effv} * v(301,302) * v(101,102)
bg 111 102 i={K} * v(301,302) * i(vm) / {effi}
vm 205 206
rp 101 102 1T
rs 206 201 1n
r3 301 302 1T
rout 201 202 1T
.Ends
* POSREL: Relative Position, Input 101(vel. of point)
* 102(ref. vel.), Output 201 (relative position of point)
* Parameters: x0 - initial position, K - gain
.Subckt Posrel 101 102 201 {x0=0 K=1}
Rin 101 102 1T
Bw1 103 0 v={K} * v(101,102)
Rtime 103 104 1MEG
Ctime 104 105 1U ic={x0}
R2 104 0 1G
Rf 104 105 1G
E2 0 105 104 0 1G
Rout 106 201 1N
Eout 106 0 0 105 1
R3 201 0 1T
.Ends
```

Figure 11, IsSPICE4 netlist for a synchro model.



can be used in other mechanical components (e.g. the rotor in a motor, the piston in a hydraulic cylinder). The synchro has 5 electrical nodes (3 for stator windings, 2 for rotor windings), and 2 mechanical nodes. The parameters for the model are the gain (K) of the transformer, the initial angle (x_0) of the rotor, and the rotor parameters (inertia- J_m , damping- b , friction- w and initial velocity- v_0).

The rotor winding is excited by an AC reference voltage at frequencies ranging up to a few kHz. The magnitude of the voltage induced in any stator winding is proportional to the sine and cosine of the angle between the rotor-coil axis and the stator-coil axis. The voltage induced across a pair of stator terminals is the vector sum of the voltages across the two connected coils. If $V \sin(\omega t)$ is the reference voltage in the rotor of a synchro, the stator voltages are: $V_1 = V \sin(\omega t) \sin(\alpha)$, $V_2 = V \sin(\omega t) \sin(\alpha + 120)$ and $V_3 = V \sin(\omega t) \sin(\alpha + 240)$, where α is the shaft angle.

When an AC reference signal drives the rotor winding, the amplitude of the voltage induced in the stator windings varies according to rotor angular position. Figure 12 shows the result of the step change in the mechanical input along with the resulting synchro output. A resolver model is also included on the newsletter subscriber's floppy disk.

In future newsletters, models for a variety of building blocks will be developed including DC and AC motors, hydraulics, control elements, and sensors. This research will culminate with a future release of a complete mechatronic library.

[1] "Modeling Motors", Intusoft Newsletter #19/20, October 1989

[2] Johan Scholliers, "Design and Simulation of Mechatronic Systems", VTT Automation January 1995