

# Intusoft Newsletter



Personal Computer Circuit & System Design Tools

Copyright © Intusoft, All Rights Reserved

Issue #61 November 2000  
Tel. (310) 833-0710  
Fax (310) 833-9658

## Using a Fault Tree

**B**ack in 1996, Intusoft began the development of Test Designer. The final product doesn't reflect the methods that were discarded. Some of the false starts and mistakes in strategy that we discovered here at Intusoft are showing up in products being proposed even today. The fault tree approach to diagnostics and testing originated when the cost of making a single test was itself expensive. Today, many tests can be made before applying inference logic. It appears, on the surface, that the terminal leaves

of a fault tree are simple intersections (see the sidebar for definitions). One could find all such intersections that reach the same conclusion and perform an "Ask the Audience?" type of vote. That method is actually available in Test Designer; however, after discovering the value of a guard band and proper sequencing, it has been virtually abandoned.

The problem with such a voting method is that it considers each test to be equal in value. Making a fault dictionary is not exact. Let's backtrack a bit and define a "Fault Dictionary". A fault dictionary contains the fault

### In This Issue

- 1 Using a Fault Tree
- 3 Handy Test Designer Definitions
- 5 Using Magnetics Designer Buttons
- 8 Using Equations in Intusoft's Units Converter
- 9 What's New
- 9 Library Manager
- 14 Modeling Bifilar Windings

signature for each failure mode or fault. It basically gives the measured response of the unit being tested for each defined fault. The ability to accurately predict these fault signatures is no where nearly as good as our ability to characterize a good unit because the unit under test is placed in a mode that is not as well understood by the test engineer, the design engineer or the simulator. That is, it's much harder to accurately predict a failed signature than it is to predict the behavior of a good device.

If one were to include some uncertainty in the fault dictionary; then the intersections provided in a fault tree would probably contain multiple fault detections even though the fault tree decisions are correct. How can this be? The sequential nature of the fault tree logic prevents you from reaching spurious detections. The sequential nature of the fault tree is NECESSARY; making measurements in parallel isn't sufficient reason to do the fault logic in parallel. Here is an example showing how an unreliable test becomes more accurate if it's performed later on in the fault tree logic.

### Guard Band Example

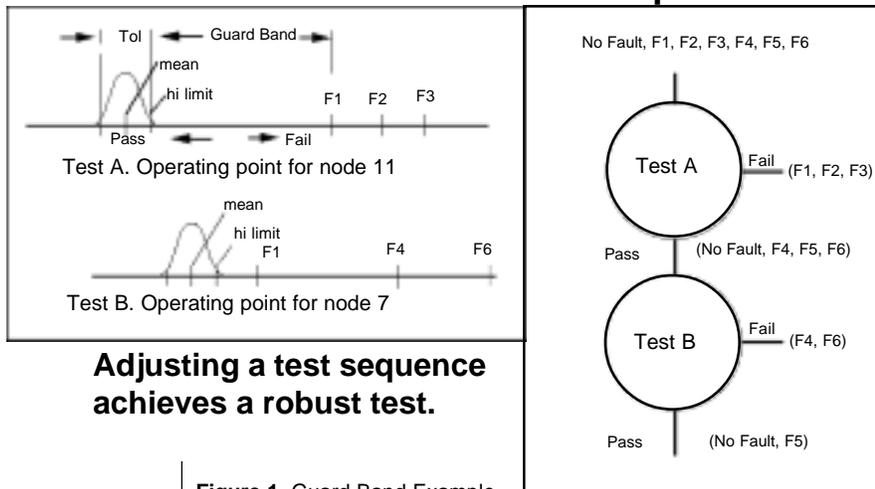


Figure 1. Guard Band Example.

Figure1 illustrates the importance of a proper test sequence; if test B was performed first, there is a high likelihood that failure F1 would be incorrectly diagnosed.

When you pick measurements that give solid fault detections, you must make that decision with respect to the faults that are being **input to that particular test**, not with respect to all faults in the dictionary. It is for this reason that the sequential processing of test intersections using a fault tree gives better fault diagnostics than you get by selecting tests without regard to the guard band or distance between the good signature and the nearest fault signature.

## Handy Test Designer Definitions

### Test Definition

- ♦ A test consists of a comparison of a resultant value against limits that classify the unit under test (UUT) as good (pass) or bad (fail).
- ♦ The resultant value can be the combination of one or more measurements and calculations.
- ♦ Interactive Command Language (ICL) allows you to do automated measurements in IsSpice. Each test has an associated ambiguity group.

### Ambiguity Group

- ♦ A list of failure modes and their failure weights.

### Failure Weight

- ♦ An arbitrary number proportional to the failure rate; it is set to 1.0 for each default failure mode.

Failure Mode	Weight
No Fault	100
C1::Short	2
R2::Open	1
Q2::OpenC	3

### No Fault Weight

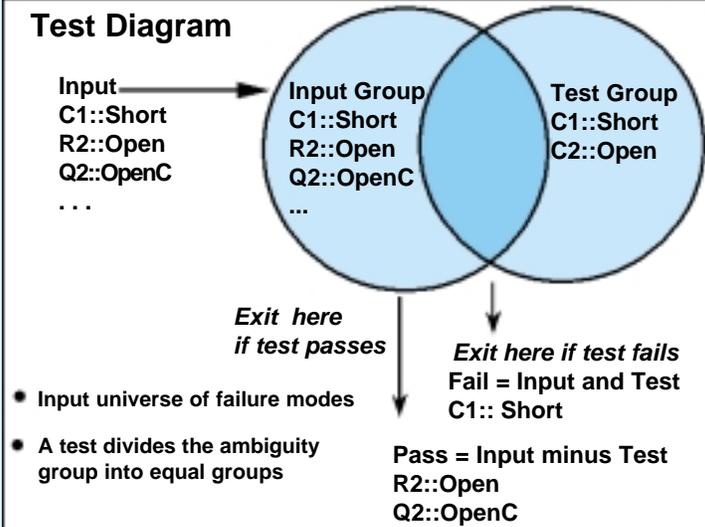
- ♦ Used to reduce the number of tests needed to arrive at the expected conclusion.
- ♦ High if UUT is expected to be good (Acceptance Test)
- ♦ Low if UUT is expected to be bad (Field Test)

### The Best Test

- ♦ Without performing an exhaustive search, the best test tends to be the test with the highest entropy. Entropy is a measure of the information resulting from a test.

## Handy Test Designer Definitions (Con't.)

### Test Diagram



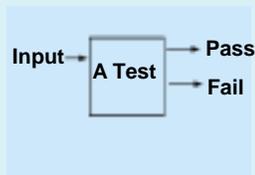
### The Best Test (con't.)

- ♦ Variations on the exhaustive search technique, such as looking ahead one test rarely produce better results.
- ♦ Exhaustive search has been shown to consume computational resources so rapidly that it is not a viable method.

### Sequencing Algorithm

- ♦ Sequencing algorithm tries to find tests that makes the probability of pass (p) equal to the probability of fail (q).
- ♦ Probability of the outcome is equal to the multiplication of the probabilities of each outcome

Let  $p$  = probability of a pass outcome  
 And  $q$  = probability of a fail outcome  
 Then  $p + q = 1.0$  and  
 $p = \frac{\sum \text{Pass weights}}{\sum \text{Input weights}}$   
 $q = \frac{\sum \text{Fail weights}}{\sum \text{Input weights}}$   
 Entropy =  $-p \log(p) - q \log(q)$



## Using Magnetics Designer Buttons

Buttons in the Magnetics Designer Transformer and Inductor tabs are fully programmable. Using <Add Buttons> in the <Edit> menu, you can define input data and use equations to calculate output data. The buttons are saved with your design so you can use the equations in other new designs. Here's an example we did for a customer at PCIM2000 so that he could convert his customers data into the input for the MD spread sheet. He wanted to have power, frequency and duty ratio as inputs for a flyback transformer operating in a discontinuous-mode switching regulator. Here are the button equations we made for him (as shown in Table 1).

After entering data shown in the buttons here, press Apply and the results are calculated and displayed in the buttons shown in Table 2. We saved the button template in DCMinductor.mag; which is posted on the web. The actual wave shape for

nearly all switching power supplies deviates from the ideal; for example, the triangular current and square voltage in this case. The data MD uses for its calculation is nearly waveform independent. To correct for these differences, you should run a simulation and measure the resulting data and feed these corrected numbers back into MD. One iteration is generally sufficient. Table 2 shows what MD does with your inputs.

The only waveform dependency of the MD data is resolved by selecting the waveform type and its duty ratio. This information is used to determine the spectral components of the AC current in calculating the copper loss.

**Table 1**  
Magnetics Designer  
Button Equations

Button Name	Button Equation	Action
power	power	input
freq	freq	input
Vinput	Vinput	input
Voutput	Voutput	input
Duty Ratio	DR	input

**Table 2.**  
Magnetic Designer's  
Programmable Inductor and Transformer Tabs

Button Name	Button Equation	Action
Edt	$Edt + DR * V_{input} / freq$	Paste into Edt Winding #1 data field
Ipeak1	$I_{peak1} = power / V_{input} / DR * 2$	Temporary data
Ipeak1entry	$I_{peak1entry} = I_{peak1} * 1.5$	Paste into Current Pk specified for Winding #1. Note, the core may saturate slightly above this value. The 1.5 factor gives some pad; you may wish to use a different pad.
IAC1	$IAC1 = DR * I_{peak1} * (4 - 3DR / (12 * DR))^{.5}$	Paste into AC Current for Winding #1
Iavg1	$I_{avg1} = power / V_{input}$	Paste into DC Current for Winding #1
L1	$L1 = Edt / I_{peak1}$	Paste into Minimum Inductance for Winding #1
The following equations hold for designs with only one secondary on Winding #2		
Ipeak2entry	$I_{peak2entry} = I_{peak1} * V_{input} / V_{output} * 1.5$	Paste into Current Pk specified for Winding #2
IAC2	$IAC2 = IAC1 * V_{input} / V_{output}$	Paste into AC Current for Winding #2
Iavg2	$I_{avg2} = V_{input} / V_{output} * I_{avg1}$	Paste into DC Current for Winding #2
L2	$L2 = L1 * (V_{output} / V_{input})^2$	Paste in Minimum Inductance for Winding #2

## Magnetics Designer Button Templates

**Current Pk specified:** Adjusts the core gap to prevent core saturation below this value. Note that the zero current inductance may be higher because MD does allow the B-H curve to become non-linear.

**AC Current:** Accounts for copper losses that are frequency dependent. The AC current is the square root of the rms current squared minus the dc current squared.

**DC Current:** Accounts for the DC copper losses.

**Minimum Inductance:** The winding inductance, at the Pk Current. Used in setting the gap and number of turns. These are particularly nasty calculations to make by hand because inductance decreases with increased turns near saturation and increases before saturation.

**Edt Winding #1:** The volt-second product that is responsible for core losses.

If you have more than one secondary, then repeat the entry for each winding and don't use the "winding 2" button results.

We have also made a set of button equations for a continuous mode flyback design in CCMinductor.mag, which can be found on the web.

Note that the discharge time is a free parameter, which was chosen to be the same as the charging time. This works well for offline converters where the breakdown voltage of the switching transistor is of concern; you need to rewrite the button equations if you want to make the discharge time different.

We have added our Fair-Rite demo to our web site. This demo was done as a special promotion for Fair-Rite Corporation. It uses a limited set of real cores and limits your transformer or inductor to 2 windings. Our regular demo uses cores that have better performance than you can actually buy; however, you can have as many windings as you like. You can make button equations in

both of these demos. This user equation capability can be quite useful for performing a number of repetitive tasks. Notice that you can make your input in a different set of units than your output; for example, use inches for input and centimeters for output. Your input data is not saved, only the spreadsheet entries.

## Using Equations in Intusoft's Units Converter

The Intusoft Units Converter and Calculator Program includes 3 main features: a scientific calculator, a comprehensive engineering units converter with constant storage, and a mathematical/logical expressions evaluation section. Software includes Help and is customizable.

The Intusoft Units Converter acts like a single button in Magnetics Designer. You can combine buttons into a multi-line series of steps. The disadvantage here is that you will have to manually revise certain lines for input. For the discontinuous mode flyback transformer, you can paste the following into the equation field, fill out the question marks and press enter to get L1. Remove one line at a time from the end and press evaluate to get the other parameters.

```
power= ???  
freq= ???  
Vinput= ???  
Voutput= ???  
DR= ???  
Edt = DR*Vinput/freq  
Ipeak1 = power/Vinput/DR*2  
Ipeak1entry = Ipeak1*1.5  
IAC1 =DR*Ipeak1*(4-3*DR/(12*DR))^.5  
Iavg1 = power/VinputL1  
L1 = Edt/Ipeak1
```

The Intusoft Units Converter and Calculator is available for free from <http://www.intusoft.com/utilities.htm>

## What's New

Scope5 will be released in our next update in late November. In the meantime, the final Beta is posted on the web. The main addition in this release is the finalization of the Update function in the <Actions> menu. You can save a graph as a template for a given analysis. When it's recalled and there is an appropriate IsSpice simulation that's active, you can press <Ctrl> + u to update the waveforms. You have 3 choices to define the behavior of the update. First, you can replace traces on the template plot, secondly you can draw the new traces on the same plot each with the same scale and thirdly you can make a new plot for the updated traces. You select the desired operation from the <Actions> Update menu before pressing <Ctrl> + u. We have also defined a set of alias waveforms so that you can use the shortcut w1, w2, ... when you type IntuScope script equations. For example, suppose w1 is current and w2 voltage, which already appears on the graph:

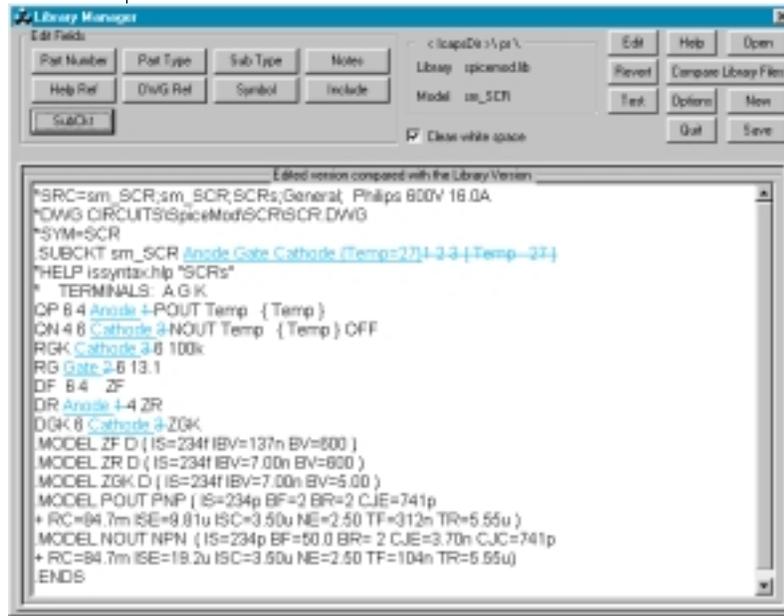
```
power = w1 * w2
plot power
newplot
plot w1 w2
```

Along with the final release of Scope5, we will have the 8.x.8 update to our ICAP/4 software. This release will serialize (Save) the "vsectol" option and integrate the new library manager test operation. The Library manager, when used in conjunction with Spicemod, provides the methods needed to easily test and add your own models to your library so that the parts can be placed on the schematic using the Part Browser dialog.

### Library Manager

Intusoft libraries are saved in ASCII files. You can edit these files with a text editor; however, many of the formats have a fairly rigid syntax. The library manager takes care of reading and applying this syntax so that the ICAP/4 Windows software can read them. Among the features in the Library Manager is the ability to change subcircuit pin

names. You can change the pin numbers into meaningful names like “anode,” “gate” and “cathode” as shown in Figure 2.



**Figure 2.** Library Manager can be used to assign names to pins, replacing previously assigned pin numbers.

If the Library Manager is present, pressing edit model in the Part Browser will bring up the library manager instead of IsEd5, the text editor. IsEd5 can still open and edit libraries using <Ctrl> + d to open a selected file.

## Common File Dialog

The Microsoft common file dialog uses part of the Windows Explorer to display the file name space (Folder, desktop, network layout, etc.). We’ve been waiting for Microsoft to add a recent folder list for a number of years. Symantec added this feature to Apple’s Macintosh computers 7 or 8 years ago. Navigation between various libraries, symbols and test circuitry has become so burdensome, we have gone ahead and made the modification ourselves. You will begin to see a new drop down list at the top of our file dialogs that is titled “Recent”. This will contain our “standard” folder locations along with the most recent places you have visited (at least the ones that are still around).

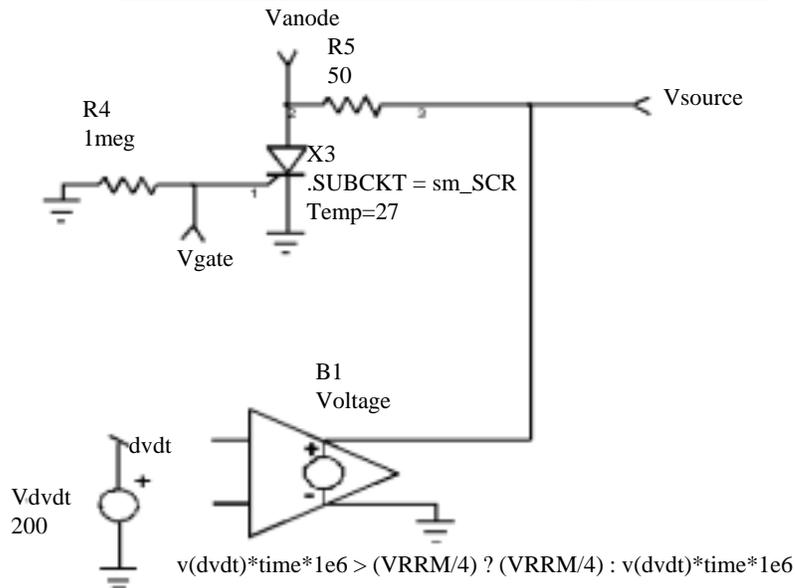
## SpiceMod5

SpiceMod5, the windows version will also be released with ICAP/4 8.x.8 in late November. It is available for ICAP/4Windows Deluxe, Professional and Test Designer users. It will feature linkage to the library manager along with test circuits and measurement scripts to evaluate each model. Shown here is the re-vamped SCR tab. Table 3 compares simulation test results with the SpiceMod5 model. Figure 3. shows the test circuit for dvbydt.

**Table3.**

Comparison of SpiceMod5 spread sheet entry with its simulated model.

Parameter	SpiceMod5	Simulation
IDRM( 27DegC)	7u	7.003u
IDRM(125DegC)	2.72m	2.709m
IRRM( 27DegC)	9.81u	9.812u
IRRM(125DegC)	.944m	.995m
dvbydt	50	48.63
IGT	17.6m	17.77m
IH	74.9m	74.76m
Tr	2u	2.151u
Ts	35u	35.02u



**Figure 3.** Test circuit for dvbydt.

SpiceNet knows the SpiceMod parameters because SpiceMod makes a file, params.txt, that SpiceNet includes in its netlist, assigning numerical values to the symbols. Here's params.txt for this sm\_SCR model: The params.txt file, shown as Table 4, assigns values for SpiceMod5 symbols.

**Table 4.**  
Params.txt File

```
.param dv_dt=50meg IDRM=0.007m IGT=17.6m IH=74.9m  
+ IRRM=0.00981m ITM=32 ITrms=16.0 Toff=35u  
+ Ton=2u VDRM=600 VGT=0.750 VRGM=5.00  
+ VRRM=600 VTM=2.00 RGK=100k
```

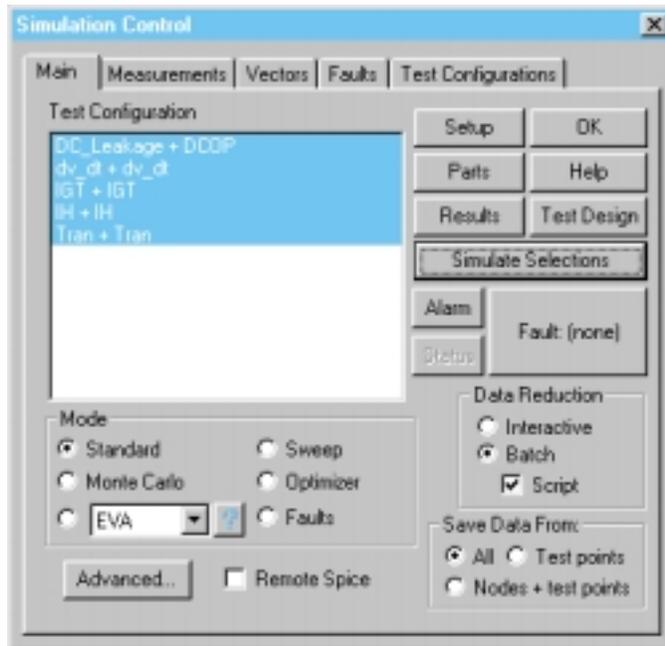
Scripts were used to get data from the IsSpice4 Simulator into the SpiceNet schematic database. The script technology is unique to the Berkeley Spice3 simulators. Many vendors who supply variations of Berkeley Spice3 left this feature out because of many bugs in the original code. Intusoft has taken the time to make the scripting language work as it was intended to work in Spice3 and has extended this powerful syntax to perform many more functions; for example, adding the cursors used in the SCR script. Below, in Table 5, is the script used to measure dvbydt for the SCR.

**Table 5.**  
dv/dt script

```
constants.error = 100  
repeat 10  
condition = getcursorY1(vanode)  
if condition > 10  
    gain = 1  
else  
    gain = -1  
end  
alter @vdvdt[dc] = @vdvdt[dc] + error * gain  
constants.error = constants.error / 2  
tran 1u 100u  
homecursors  
end  
dvbydt = @vdvdt[dc]  
set rewind  
echo tran analysis measurements  
echo tran analysis measurements  
echo  
echo test 1 dvbydt  
print dvbydt
```

This script gets to the final dvbydt by successive approximation. It increases or decreases dvbydt depending on whether or not the SCR turned on in the previous simulation. The change in dvbydt error is reduced in half each time.

The trick in making a test circuit for SpiceMod5 is to insert a model with a common name into Spicemod.lib. Basically we prefix these test models with “sm\_”. The SCR model then becomes “sm\_SCR”. When SpiceNet is directed to open a drawing with the open\_sm action, SpiceNet will prompt you with the “Update Cache” dialog. If you agree, press <Apply> and SpiceNet will load in the new model description. Use the <Actions> ICAPS menu to bring up the simulation control dialog. You can run all of the tests at once by shift selecting all of the “Test Configurations”, Standard Mode, check the script box in data reduction and press <Simulate Selections> as shown in Figure 4.



**Figure 4.** The Simulation Control Dialog lets you evaluate your SpiceMod5 model.

Then press the <Results> button to compare the simulation results with the SpiceMod spreadsheet.

A major new feature in SpiceMod5 is the capability to link external DLL's so that they become model tabs. This is the way the Diac tab was made. We will be releasing the SpiceMod Software Development Kit, SMSDK, in November for owners with up-to-date maintenance of ICAP/4 Windows Professional, Power Supply Designer and Test Designer. Anyone that wants to make add-in models for free or for profit can get them listed on our web site if they provide Intusoft with evaluation copies.

SpiceMod5 now has a restricted license controlling use of models made using the software. This shouldn't affect end users, but it prevents our competitors from making and selling models using our software.

## Modeling Bifilar Windings

The geometrical view described by Dauhajre<sup>1</sup> and used in Magnetics Designer yields no leakage inductance for bifilar windings. Bifilar windings reduce leakage inductance significantly; however, using zero may not be a good idea. We can instead treat the winding as a transmission line. The common mode inductance is preserved using the IsSpice model for a perfect transformer. The bifilar coupling is then introduced using a transmission line as shown in Figure 5.

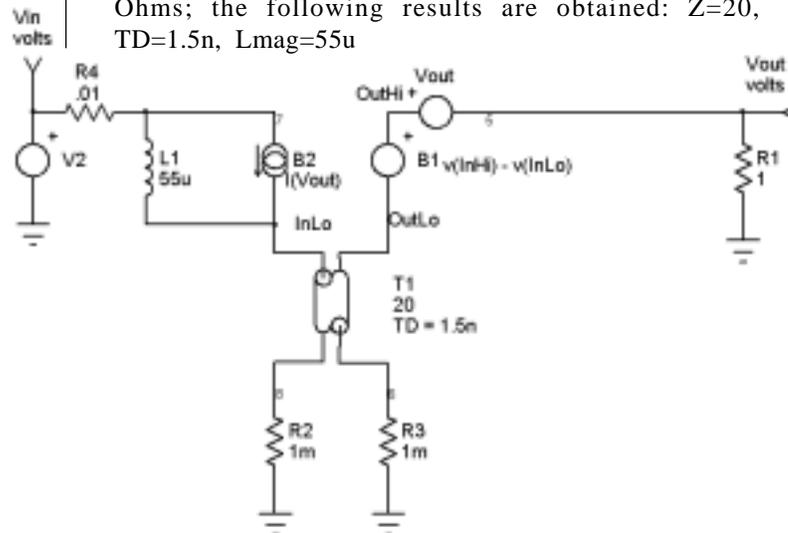
Determining the impedance of the transmission line is tricky. As the distance between conductors goes to zero, the impedance approaches zero. For transformers, the control of the wire separation is poor so that experimental measurements may be required. Here are a few values estimated from a graph in the ITT handbook:

Let  $D$  be the distance between the center of the wires and  $d$  be the wire diameter; then:

Low impedance must go with large diameter wires because the insulation thickness remains constant. Large diameter wires aren't good at high frequency so we must conclude that an impedance

$D/d$	Impedance (Ohms)
1.0	0
1.05	40
1.1	55
1.2	75

ranging from 10 to 50 ohms is about as good as you'll get. Here's the result for a bifilar wound 4-turn #13 winding on a ETD-34 ferrite core. The winding length is  $4 \times 2.2/12$  ft; if we estimate propagation delay of about 1.5 nsec to account for dielectric properties and an impedance of 20 Ohms; the following results are obtained:  $Z=20$ ,  $TD=1.5n$ ,  $Lmag=55u$



**Figure 5.** An IsSpice Model for bifilar windings.

If we load the winding with 1.0 Ohms and input an AC voltage from a .01 Ohm source, the resulting transfer function is shown below. We could further refine (complicate?) the model by using a lossy transmission line; however, we would need to get additional model parameters from experimental measurements.

The results show that the transmission line model isn't required at frequencies commonly used for power supplies (below 1 MegHz). On the other hand, for RF operation, at frequencies above 10MegHz, the transmission line model is needed. The added complexity when using the transformer below 500KHz isn't really warranted. The transmission line effects at these lower frequencies are small enough to ignore; the external interconnection is likely to be more important than the internal "leakage" inductance. Check the html version of our newsletter <http://www.intusoft.com/nl61.htm> for Figure 6, a graph of the results.

<sup>1</sup>Modeling and Estimation of Leakage Phenomena in Magnetic Circuits by Dauhajre, Abrahm Arnold 1986 UMI Dissertation Services.

## 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**

**Denmark, Spain, Portugal, 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. Müller**

**Germany, Austria, Croatia, Slovenia, Czech 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, Southeast Asia**

Tele: 886-2-2847-9888-9

FAX: 886-2-2289-0151

email - chochieh@ms2.hinet.net

Web: <http://www.chochieh.com.tw>

### **Cadforce Systems, Inc - Reg Harrison**

**Montreal, Ottawa, Quebec Canada**

Tele: (800)249-7332 or (514) 286-0347

FAX: (514) 286-4861

e-mail: reg@cadforce.ca

Web: <http://www.cadforce.ca>

### **Cadforce Systems, Inc - Monique Masserey**

**Calgary, Toronto, Vancouver Canada**

Tele: (888) 464-8552 /FAX (905)275-5546

e-mail: monique@cadforce.ca

### **Elbo - Bogdan Olejarczyk**

#### **Poland**

Tele/FAX: (48-22) 838-23-91

email - elbo\_@astercity.net

### **Trident Techlabs PVT Ltd.**

#### **India**

Tele: 91-11-593-9907 or 593-9960

Fax: 91-11-593-9960

email - tritech@ndb.vsnl.net.in

### **Trident INFOSOL PVT LTD.**

#### **India**

Tele: 91-80-224-5037, 223-3579, 223-6304

Fax: 91-80-229-1566, 22-4574

email - trident@bgl.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 Wannemacher**

#### **Switzerland**

Tele: 41-1-308-66-66/FAX: 41-1-308-66-55

email - peterw@ccontrols.ch

Web: <http://www.ccontrols.ch>

### **IVIS Co., Ltd. - Hiro Nagano**

#### **Japan**

Tele: 81-45-332-5381/FAX: 81-45-332-5391

email - sales@i-vis.co.jp

Web: <http://www.i-vis.co.jp>

### **EDA Software - John Meltezos**

#### **Greece**

Tele: 301-8256258-9/FAX: 301-884-1016

email - sales@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 - sales@intsys-europe.fr

Web: <http://www.intsys-europe.fr>

### **DFM - Tuvia Liran**

#### **Israel**

Tele: 972-4-9533059/FAX: 972-4-9533057

email - dfm@netvision.net.il

Web: <http://dfm4vlsi.com>

### **TECH 5 - Geert Mosterdijk**

**Netherlands, Belgium, Luxemburg**

Tele: 31-184-6155-51/FAX: 31-184-6154-51

email - info@tech5.nl

Web: <http://www.tech5.nl>