

# Proposed SPICE model for core loss

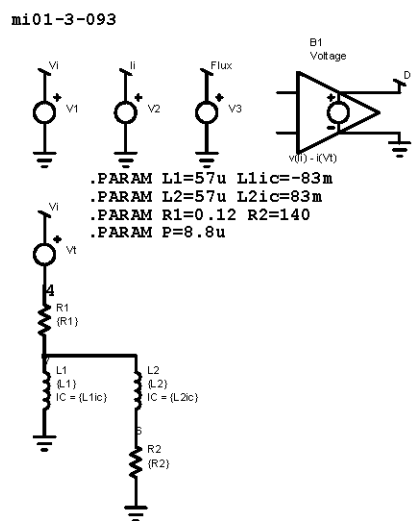
Edward Herbert  
Co-Chairman, PSMA Magnetics Committee

February 18, 2012  
Revised December 6, 2014

This write-up describes a method for making SPICE models. As an example, we show how to make simple SPICE model of a magnetic core for estimating its core loss.

## A simple SPICE model

The very simple model below is reasonably good at simulating the hysteresis loops of the various waveforms used in core loss studies sponsored by PSMA at Dartmouth that examined the influence of increasing off-time. The actual model of the core is the two resistors and the two inductors. The other components provide the test environment and measurements.



## Background

I have long been dissatisfied with existing SPICE modeling of magnetic components, but the solution has been elusive.

Most methods of estimating core losses use the maximum flux density  $B_{\max}$ , a parameter that cannot be determined instantaneously, and thus is not suitable for use in a SPICE model. A way of using instantaneous parameters such as voltage and current and parameters easily derived from them, such as instantaneous flux, had to be developed.

There is no need to use the unfamiliar magnetic parameters to simulate magnetic component behavior. For a component, there is no need to use dimensional factors, or even the turns ratio.

Two studies of core loss were sponsored by PSMA at Dartmouth, and they produced a large volume of very good data, data that was formulated as voltage and current vs. time.

### ***SPICE model construction—theory***

Test data from a real sample, the voltage and the current, are imported into SPICE. In this case, the voltage is the independent variable, the excitation detected using a test winding on the magnetic core. Applying that excitation to the magnetic core resulted in the measured input current, the independent variable.

In building the SPICE model, the same excitation is applied to the SPICE model and the simulated current is compared to the measured current. If the simulated current matches the measured current, then the SPICE model is valid for that exact excitation. If that is the only excitation that will be used, then the model is done. Usually, however, the model must be tested with a variety of excitation voltages and waveforms that represent the extremes of practical applications, and the model must be improved iteratively until there is an acceptable match under all excitation conditions.

This discussion is about magnetic components, but similar techniques can be used to model other components such as capacitors.

### ***SPICE model construction—automated***

Automated equipment for constructing a SPICE model will fixture a sample component and apply a systematic series of stimuli, in the process building a table of response to the stimuli. In the case of a magnetic component, the stimuli may be various excitation waveforms and voltages as measured on a test winding and the response is the current through the drive winding.

The same stimuli are then applied to a SPICE model template. Under software control, the component values of the SPICE model are adjusted iteratively until the simulated current of the model matches the measured data to the required accuracy. In a sophisticated model, the components may have dependencies on voltage, flux, temperature and other derived parameters, all easily implemented in SPICE.

A simple SPICE model for estimating core losses is derived with ideal passive components in the example that follows. With four components, two resistors and two inductors, the simple SPICE model matches the core loss of the measured data very well when excited by the measured stimuli. That is to be expected, as, after all, the exercise is really an elaborate curve fitting scheme. More promising is that the curve fit is reasonably good for other voltages and waveforms.

### ***Equipment***

In automatically generating SPICE models, the most important equipment is a computer such as a PC or workstation having a suitable program installed.

As part of generating the SPICE model, the computer commands test equipment to apply stimuli to the article being simulated. In general, this will be a programmed series of voltages or currents that are appropriate to the article (transformer, inductor, capacitor, etc.). Most modern test equipment can be controlled by a computer over a bus such as USB, as an example, not a limitation.

As the article is stimulated, its response is measured by data acquisition equipment such as an oscilloscope, wave-form analyzer, etc. Most modern oscilloscopes can sample and store data at a high rate and then upload it to a processor over a bus such as USB, as an example.

In most instances, it will be preferred to have a test fixture with suitable terminals for the article. It may include clamps or other holding means. A test fixture can ensure that the interface is well designed and is consistent from test to test.

### ***SPICE model template***

Usually, the general nature of an article will be known and a model template can be used. The template may be a collection of model components interconnected in a network with tentative values. The tentative values may be generic or they may be refined with some knowledge of the article such as its expected voltage and frequency range, and maybe even anticipated component values, such as a starting point inductance or capacitance estimate.

Regardless, the article is stimulated and a database of the response is assembled. Then the same stimuli is applied to the SPICE model and the software compares the response to the response of the test article. The model is then improved iteratively until the model response is an acceptable match to the actual response.

Although it would be possible to program software that would assemble a model from a library of components, it is more efficient to have a starting template with an arrangement that is known to be suitable for the type of article being tested and with reasonable tentative values for the components.

The template may include representative temperature coefficients for the various components derived from a library of SPICE circuits that are known to be representative of like articles. The model generator may then generate a SPICE model from data taken at one temperature, and the model can be expected to predict to a reasonable accuracy the behavior over the whole temperature range. Obviously, the model will be superior if actual data taken over a temperature range is used, but this method should give good results for components that are variations of the baseline components.

As an example, the temperature coefficient of an inductor is largely determined by the material of which it is made. Other inductors made of the same material, though of different size and value, would be expected to have similar temperature coefficients.

## The SPICE model preparation

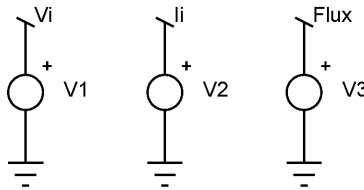
It assumes that there is measured data available in a suitable format to reconstruct the waveforms of the measured data in the SPICE model. A table with two columns, time and the measured parameter, can be imported into the PWL table of a SPICE voltage or current source. A short example is shown below. For good resolution, a real data table may have thousands of lines.

0.00000000E+00	7.57318193E-02
1.42800000E-08	7.73022123E-02
2.69000000E-08	7.55057087E-02
3.95200000E-08	7.65876657E-02
5.21400000E-08	7.72300922E-02
6.47600000E-08	7.86718761E-02
7.73800000E-08	7.83250494E-02
9.00000000E-08	7.62792205E-02
1.02620000E-07	7.80319258E-02
1.15240000E-07	7.91575263E-02
1.27860000E-07	8.02883888E-02
1.40480000E-07	7.93554698E-02
1.53100000E-07	7.95096151E-02
1.65720000E-07	8.04502723E-02
1.78340000E-07	8.01340889E-02
1.90960000E-07	8.13677148E-02
2.03580000E-07	8.11903550E-02
2.16200000E-07	8.16478381E-02
2.28820000E-07	8.13985129E-02
2.41440000E-07	8.21591793E-02
2.54060000E-07	8.31846473E-02

**Note:** This is an example, not part of the data used below.

## Building the SPICE mode—the input sources

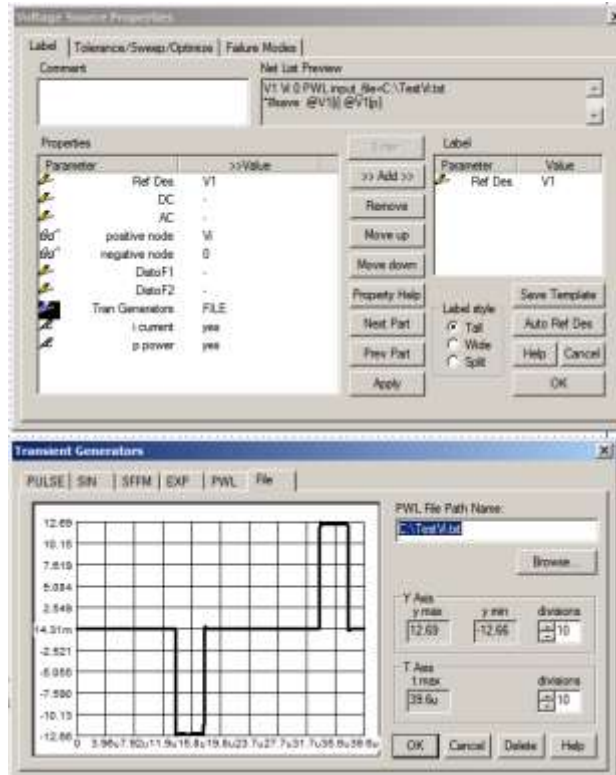
Start with the three voltage sources with terminals "Vi", "Ii" and "Flux," as shown below. In each, set up the PWL file for each voltage source and import the data. A convenient way of doing this is by establishing three text files with the data. The PWL for each source can be programmed to import the text files when the model is run. In this way, new data is easily imported by updating the text files or by selecting different text files.



In this example, the flux data is available, but alternatively, it can be generated in the SPICE model by importing the voltage data into a SPICE current source. The current source then charges a capacitor. If the PWL data in the current source is the measured time and voltage, and the capacitor is one microfarad, the voltage on the capacitor corresponds to the flux in volt-seconds. (For other units, the value of the capacitor can be chosen as a scale factor. If there is an initial flux, the initial condition of the SPICE capacitor can be set to the corresponding voltage.)

The examples below are using Intusoft's ICAP/4. It is expected that other SPICE programs would have similar capabilities. While SPICE is used for the example and is the preferred modeling program, that is partly because of its familiarity and wide use. The same general methods can be used with other circuit modeling software.

### ***Importing the data***

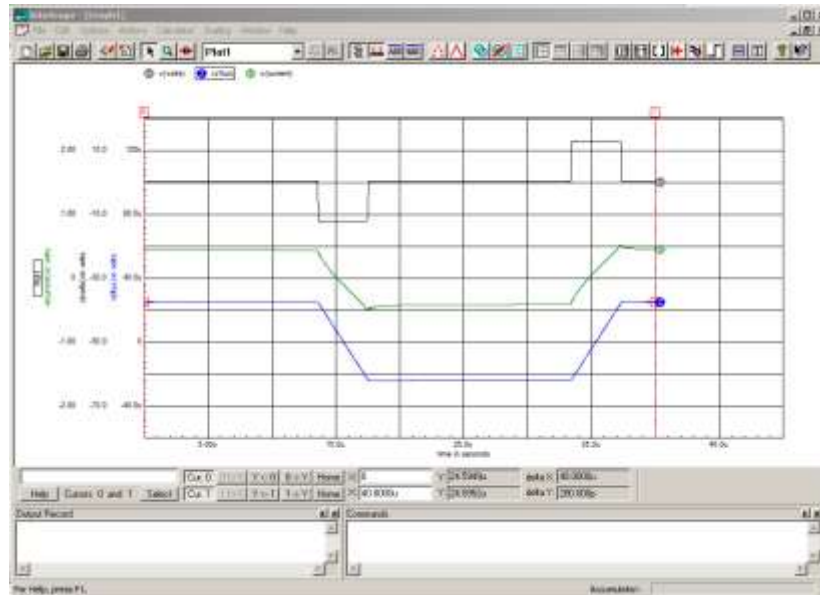


Open the voltage sources, each in turn, select Trans, then File and select the file to which the data has been saved. (Alternatively, open PWL and copy and paste the data.)

Once the file is selected, its wave shape is shown. Do the same for the current and flux files. Name and save the model file.

If the voltages are displayed using the SPICE scope utility, we can see the input voltage  $V_i$ , the input current  $I_i$ , and the flux in volt-seconds, as shown in the figure below.

**Note:** The illustrations in this write-up are not all from the same SPICE model and simulations. While that would be preferred, the illustrations were selected to emphasize points that are made in the text. The differences are small but noticeable, especially in the amount of noise seen.



## ***Importing drawings into documents***

The pictures above were imported using print-screen in Windows. The screen shots were then imported into Photoshop Elements, cropped, and saved as .tif files. The .tif files are then imported into Word as pictures. For other purposes, such as a web page, the .jpg file format might be preferred.

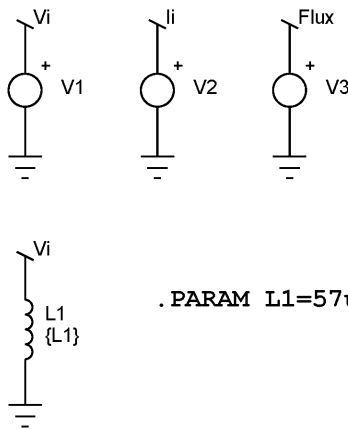
Some pictures can be imported just by copying and pasting them, but the results often are disappointing. Many can be imported into a CAD program as an alternative to Photoshop, and editing for appearance is more flexible in CAD if the drawing is vectors and text. Text can be added or edited, and the color and weight of lines can be modified. The drawing can be cropped and scaled, and features from several drawings can be combined if attention is given to the scale. Bit maps and screen shots do not import well into CAD.

Copying and pasting from CAD to a document may be possible, but again the results may be disappointing. It usually is better to print to a .pdf file, crop as necessary, then save as a .tif file and import that into the document.

## The SPICE model, getting started

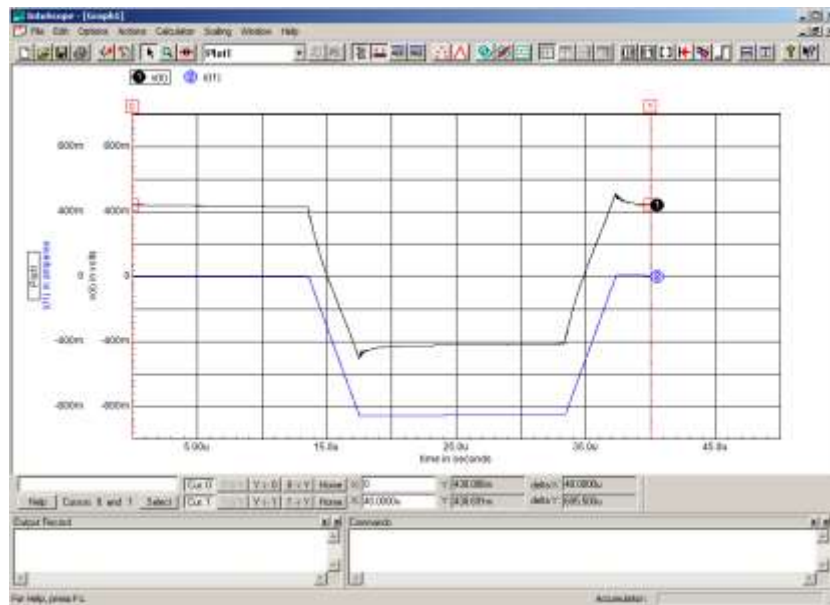
To begin building the SPICE model, add an inductor, and connect it to  $V_i$  using a terminal.

mi01-3-093



Note that I set the value of the inductor  $L_1$  to  $\{L_1\}$ , and created a `.PARAM` statement to set its value. This is unimportant, just my preference. The value of  $L_1$  was set by trial and error. Putting the `PARAM` statement on the schematic does nothing except display it. It must be entered into the SPICE setup, along with other functions such as the run time and so forth.

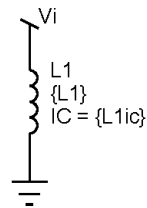
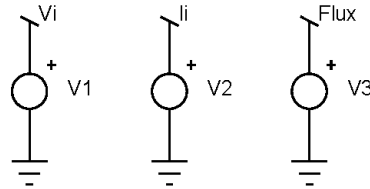
Run the SPICE model, and display the currents  $I_i$  and  $i(L_1)$



## Setting initial conditions

Note the current offset. The exact value can be read from the cursor at the bottom of the screen. This must be added as an "initial condition" for the inductor L1.

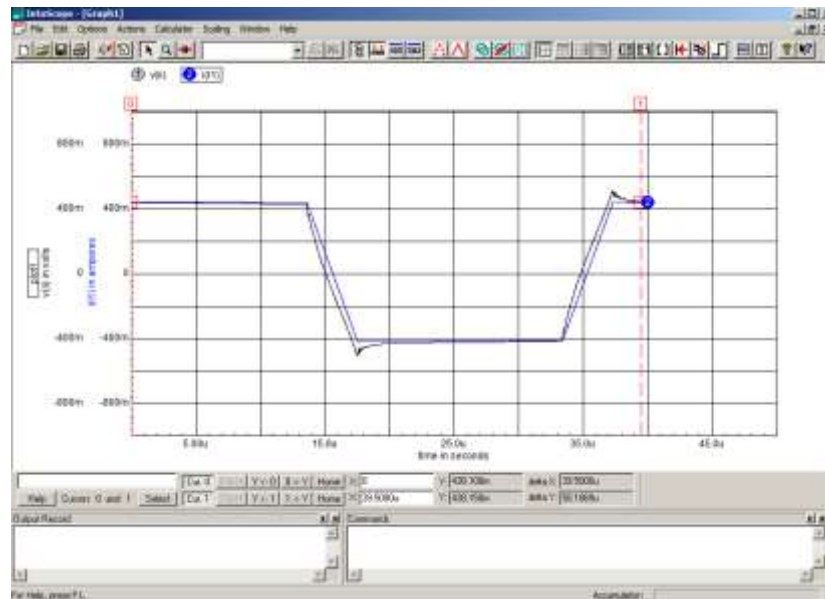
mi01-3-093



```
.PARAM L1=57u L1ic=438.1m  
.PARAM P=39.5u
```

I added a parameter `.PARAM {L1ic} = 438.1 m`. I also added a `.PARAM P=39.5u`. The parameter `{P}` is the period, and can be used to set the run time in the Simulation Setup.

Now the currents look like this:

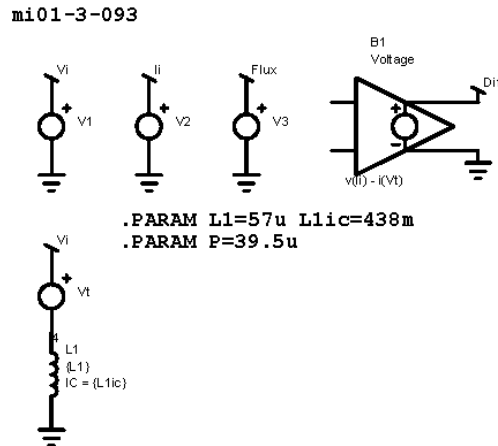




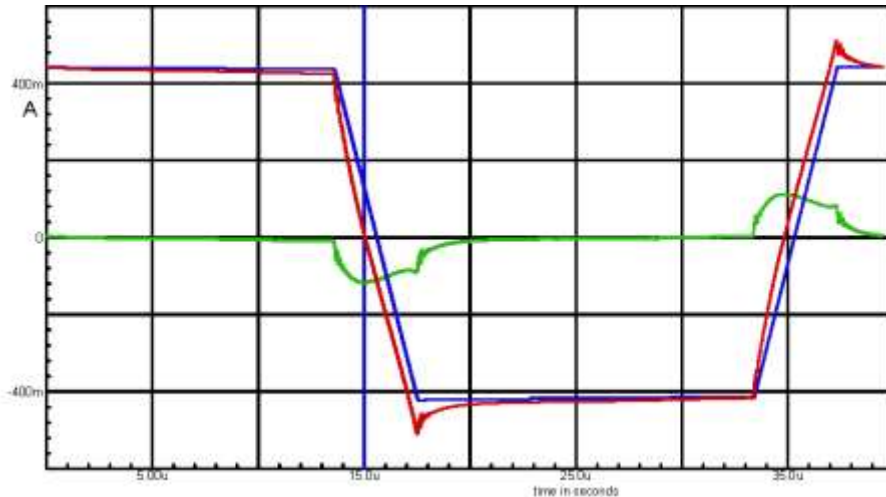
## Improving the SPICE model measurement and visualization

Next, I added another voltage source set at 0 V and named it "Vt," to measure current, a SPICE recommended procedure. I also added a behavioral voltage source with a ground and a terminal "Dif". The formula for the behavioral voltage source is  $v(i) - i(Vt)$ . SPICE does not care that I am subtracting a current from a voltage; it just uses the values of each. This allows me to see how good the model is as I add components.

The objective is to get the model current  $i(Vt)$  close to the measured input current  $v(i)$  using the same excitation voltage  $v(Vi)$ .



Running it, I can see the following;



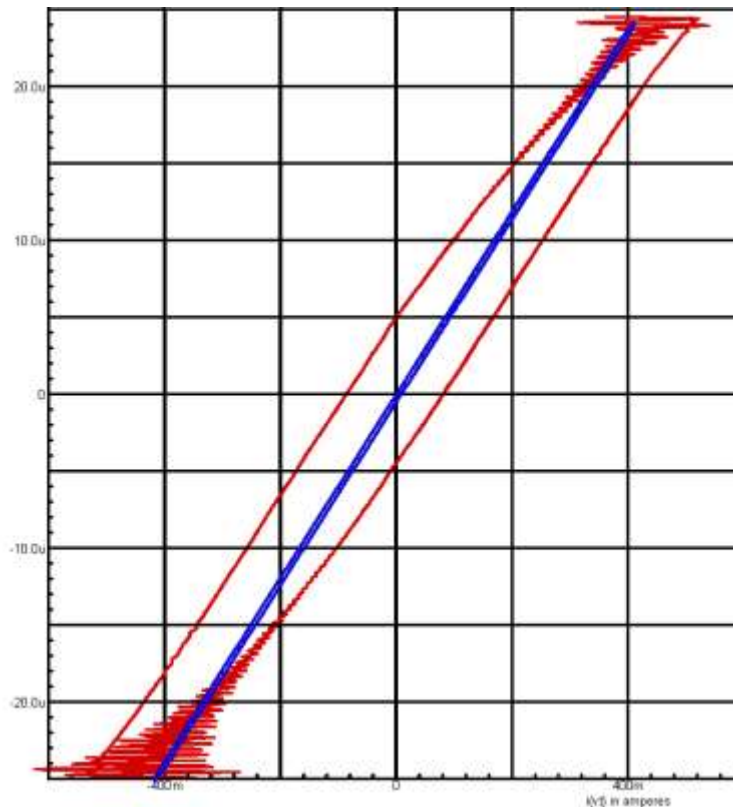
The scale factors of the traces can be set in the SPICE oscilloscope. The Dif function makes the current difference easier to see. In particular, the vertical distance between the sloped lines is very hard to visualize. Dif shows a much greater error there than is apparent to the eye.

## ***Importing drawings to CAD***

The drawing above and most of the drawings in the following discussion are imported into CAD using the SPICE scope utility print function, and in particular the option to “print to clipboard.” The drawing can then be copied into CAD as a vector and text drawing. The drawings can be cleaned up for appearance by removing artifacts, changing color, changing line weight and style, all without altering the data. Most important, data from different simulations can be superposed if care is taken that the scale is consistent.

## ***Displaying the hysteresis loop***

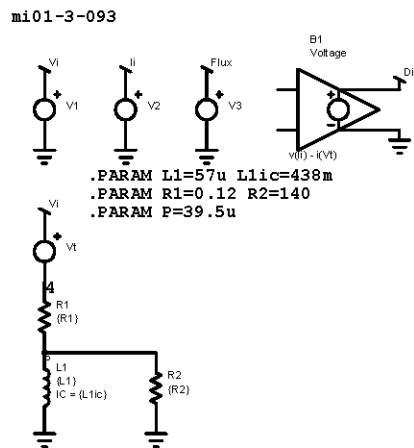
The hysteresis loops for Flux vs.  $i_i$  (the measured data, red) and Flux vs.  $i(Vt)$  (the simulated current, blue), are superposed. With only an inductor, the model hysteresis is a straight line.



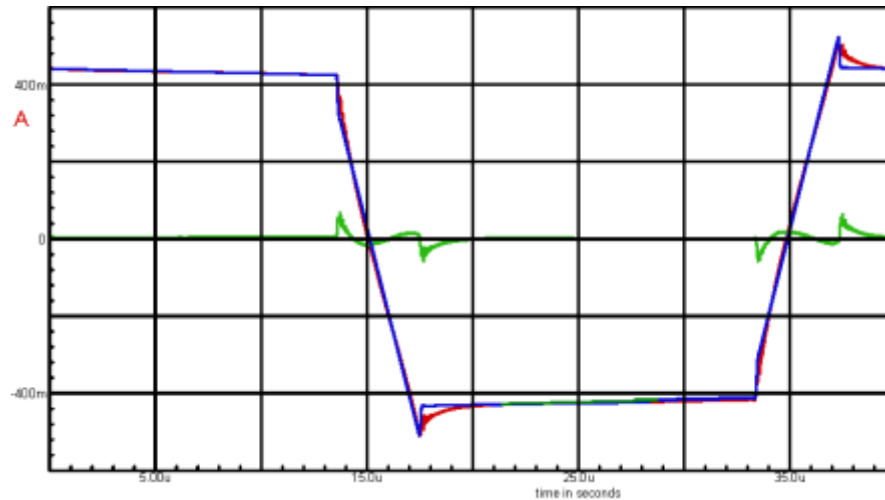
It is difficult to display superposed hysteresis loops in the SPICE scope utility. Individually, they are displayed by using the current for the x-axis and the flux for the y-axis. Because the currents are different for the two hysteresis loops, the only way to show them superposed in SPICE is to use the flux as the x-axis, because it is the same for both, and use the y-axis for the two currents. This looks really strange. It is preferred to import the curves individually into CAD and superpose them in CAD.

## Improving the SPICE model

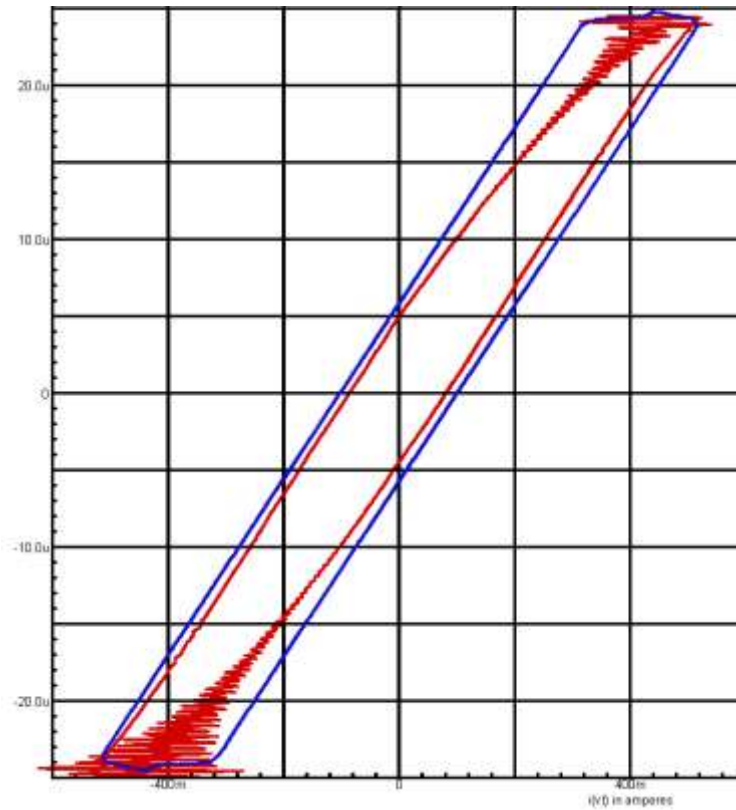
To move more quickly and combine some steps, note the gentle slope of Dif on the left. This is current decay during the off time, and suggests some series resistance. By trial and error, I added  $\{R1\} = 0.12$ . Note also the steep sides of the Dif. This suggests a parallel resistance, and I added  $\{R2\} = 140$ , again by trial and error.



The model is significantly improved. The Dif error is much less than it was. Note that the slight slope error on the left is now gone. The contribution of R1 is small, but is important for simulating longer off-times. The contribution of R2 is greater. The amplitudes of the currents are now quite well matched, but the time constants are mismatched.

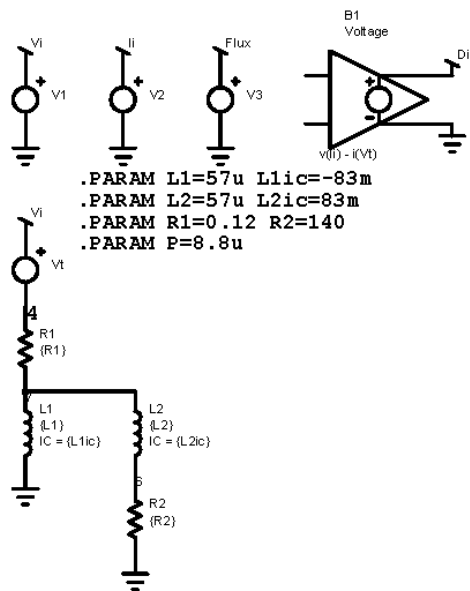


By adding the resistors, the SPICE model now has losses, which show up as the area in the simulated hysteresis loop, blue. The hysteresis loop of a resistor alone is a rectangle, centered about the y-axis

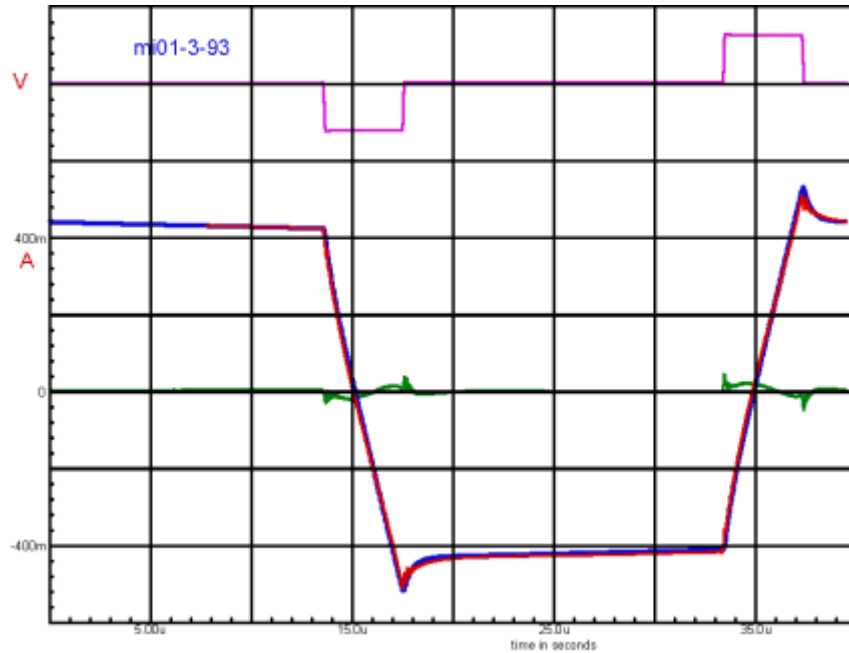


There is an exponential current increase as the excitation voltage is applied, and an exponential current decrease after the voltage is removed. This can be seen in the  $v(i)$  trace, but much more clearly in the  $Dif$  trace. A series inductor is added.

mi01-3-093

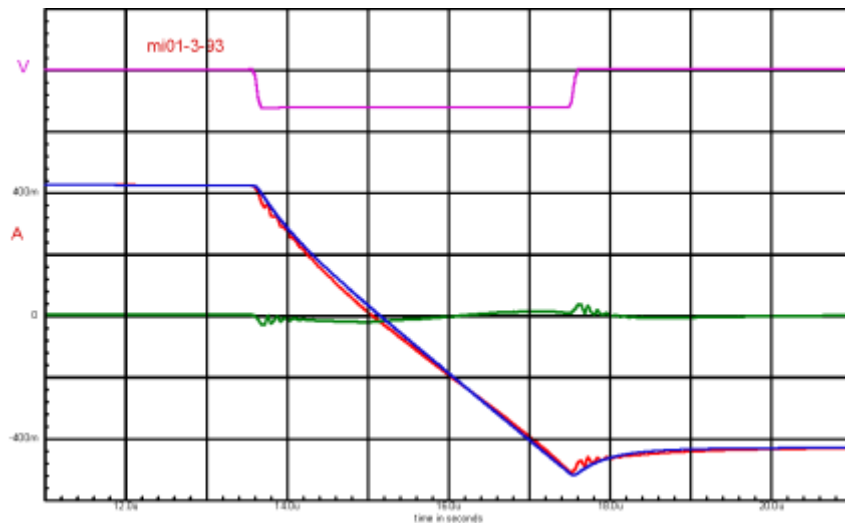


Now the currents can be compared again:



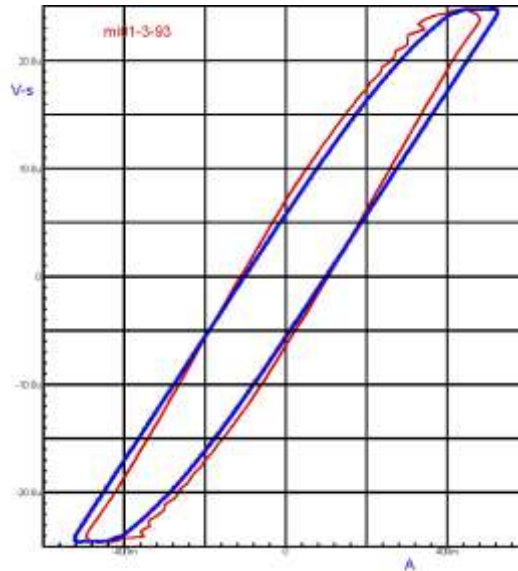
The current match between the data and the simulation is now fairly good. Notice that the error in the Dif current is not zero, but has nearly equal area above and below 0. There will be a slight error in the shape of the hysteresis loop, but the area within it should be quite close.

The graph below shows the same curves as above, but it is zoomed in to highlight the time of the negative excitation pulse. The difference between the current traces is somewhat easier to see, but the difference current display is useful nonetheless.



The difference current display is particularly valuable in showing that the error (the difference current) is about the same when excitation is applied as when it is removed. This suggests that the off-time core loss phenomenon is mis-named. It is there during the on time as well. This suggests that transition time core loss is a better name.

By eye, the areas of the hysteresis loops appear to be very close, so the core loss estimate will be as good. This can be measured more exactly in CAD using the function to measure the area enclosed by a line. Unless scaled rigorously or converted, the area units in CAD will be nonsense, but their relative values are useful for comparison.



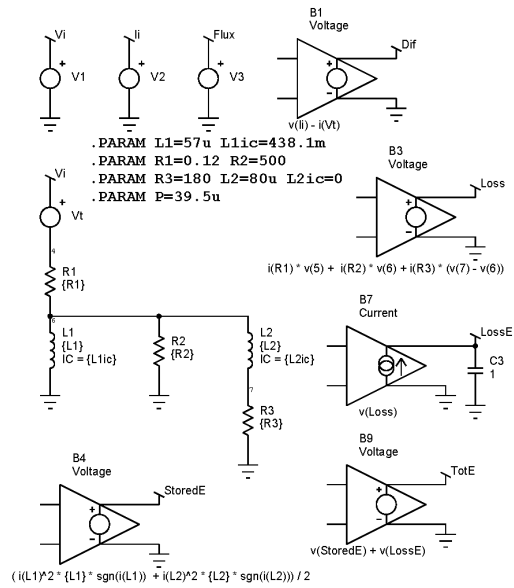
Improving the SPICE model further requires running different waveforms on the model as will be shown below. The parameters can be modified iteratively and new parallel branches can be added to refine the model as much as desired. Manually, this becomes an exercise like the game "Whac-A-Mole," but it is expected that a viable calibration algorithm will be found with further testing and experience. The temptation to over-refine the model should be avoided—at some point the added accuracy will have diminishing returns and may be dominated by other variables such as lot-to-lot variations.

Real magnetic cores have a significant temperature coefficient. With temperature dependent data, an algorithm for modeling the temperature coefficients should be straightforward. The effects of dc bias and load currents in transformer circuits should be added as well. There is suggestive evidence that the circuit influences core losses. If confirmed, an impedance model in Spice will be much more accurate when used in a circuit model.

### ***Added functions for analysis***

Some additions to the SPICE model are worthwhile. A behavioral voltage source B3 calculates the loss as the total of the  $I^2R$  losses of the resistors. A behavioral voltage source B4 calculates the total stored energy as  $\frac{1}{2} I^2L$  for the inductors. The energy lost is calculated in the behavioral current source B7 and the capacitor C3 as the integral of the loss. The total energy is calculated in the behavioral voltage source B9 as the sum of the stored energy and the lost energy.

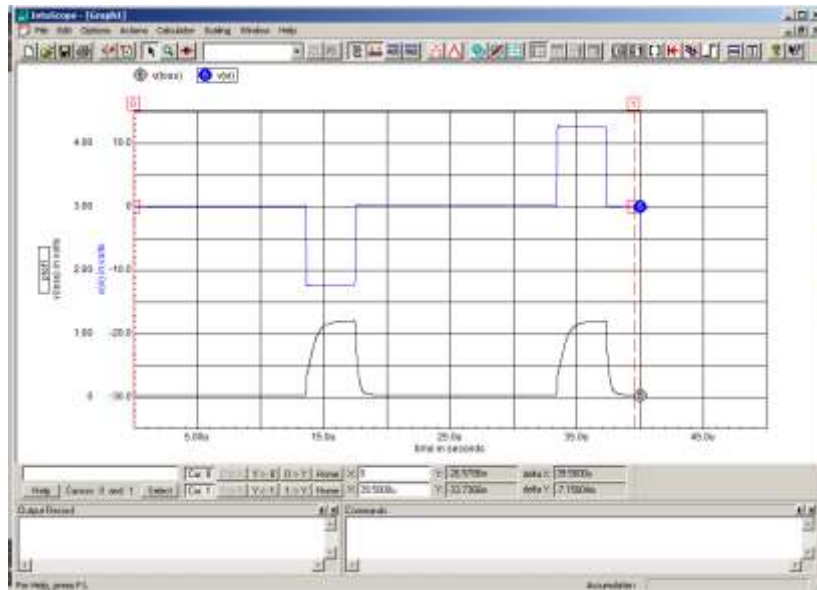
mi01-3-093



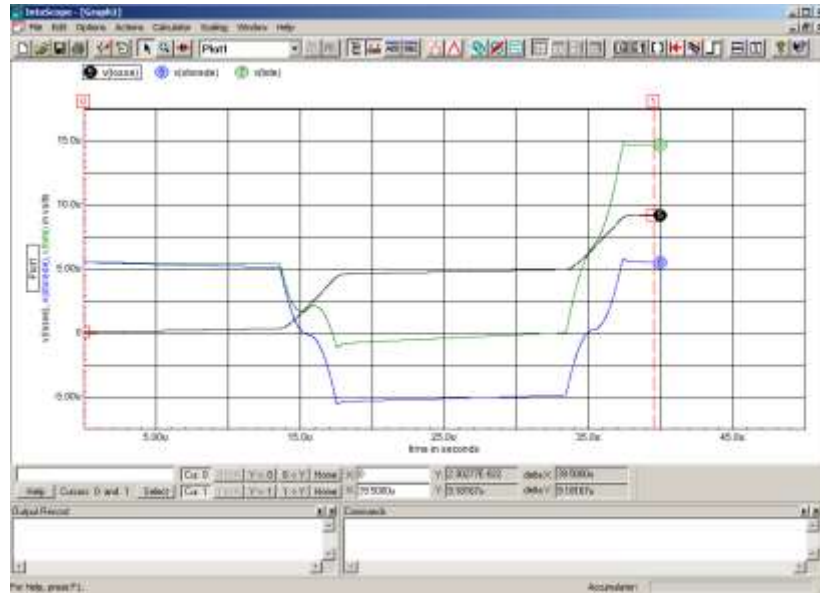
**Note:** The core loss SPICE model is different. This illustration is to show the added functions. The simulation examples that follow use this model, and it is not as good as the simple mode.

### The losses in the SPICE model

The first picture below shows the input voltage  $V_i$  and the loss. Note that this is not the same as the instantaneous input power as might be calculated by multiplying the input voltage  $V_i$  and the input current  $I_i$ . The loss shown below is the loss calculated in the model resistances. The loss never goes to zero and persists after the voltage is removed.



The picture below shows the energy loss, the energy stored and the total energy over the cycle in the SPICE model. The energy loss in the SPICE model should equal the energy calculated from the measured data, and it is quite good. The value from the graph can be seen at the bottom as Cur 1 Y = 9.18u. The calculated energy from the data is 9.16E-6 J. This close match exists because the model is running with the same excitation conditions that were used to make the model.



The energy stored must be equal at the beginning and the end of the cycle if conditions model steady state conditions with a repetitive waveform. The initial stored energy is determined by the inductors and the initial currents specified for them.

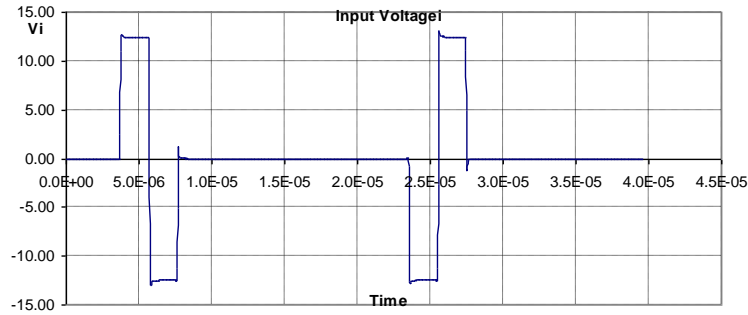
However, remember that the model is just a curve fitting technique, with fudge factors as the component values. Drawing any conclusions about the underlying physics is risky.

### ***Using the model with other waveforms***

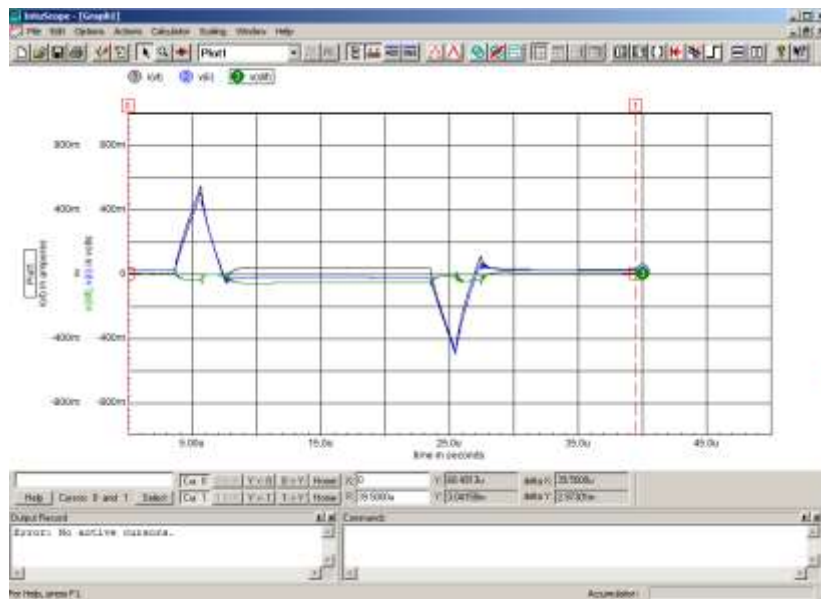
Ideally, the model would accurately predict the current with other excitation levels and waveforms.

To compare the model using other waveforms, the source voltages are updated with the data of a different waveform. However, the period {P} and the initial currents {L1ic} and L2ic} have to be set for the new curve. The data selected is for a "Hippo curve" with an off-time of 15.8  $\mu$ s and  $V_i$  of 12.5 V. The graph below is copied and pasted from the a graph in a data spread sheet.

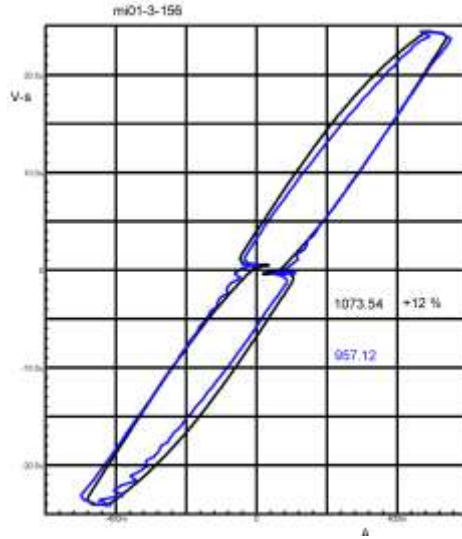




The input current  $v(I_i)$  from the measured data and the model current  $i(V_t)$  are shown with the Dif error current. The errors are significant.

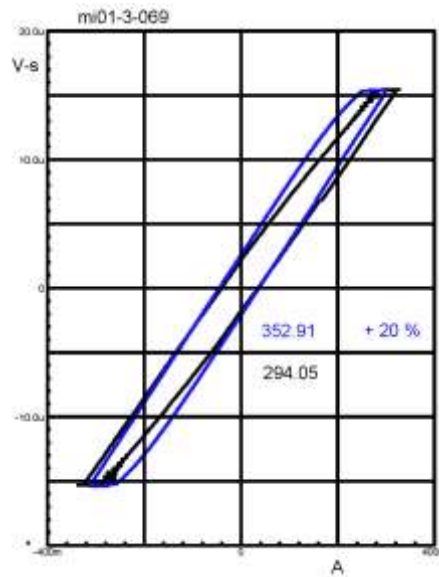
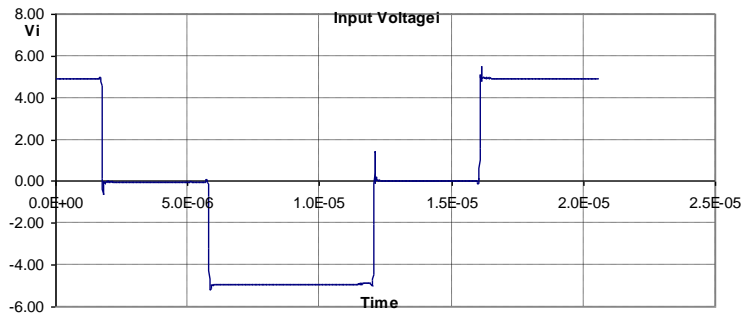


The hysteresis curve for the model vs the data is shown below. The blue hysteresis loop is from the SPICE model and the black hysteresis loop is using the .csv data input current, both with the .csv input flux as the vertical axis. By taking the ratio of the areas inside the hysteresis loops, we find that the model estimates the loss about 12 percent low. The qualitative behavior is quite well matched.

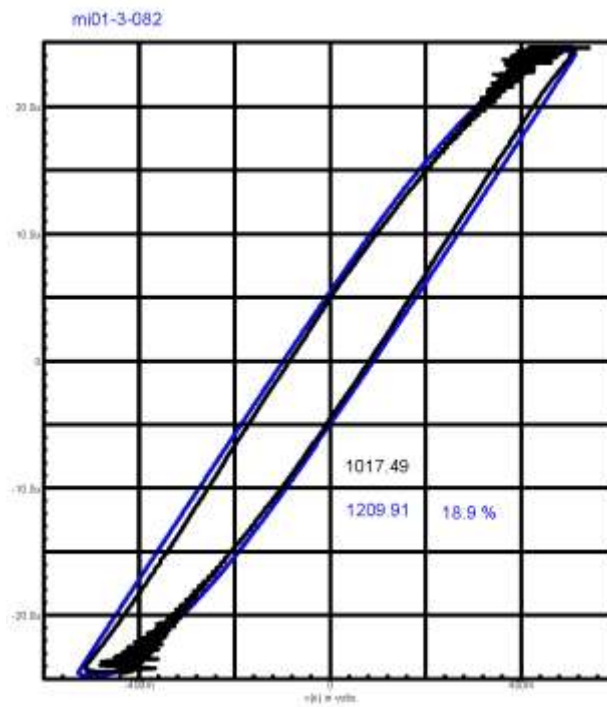
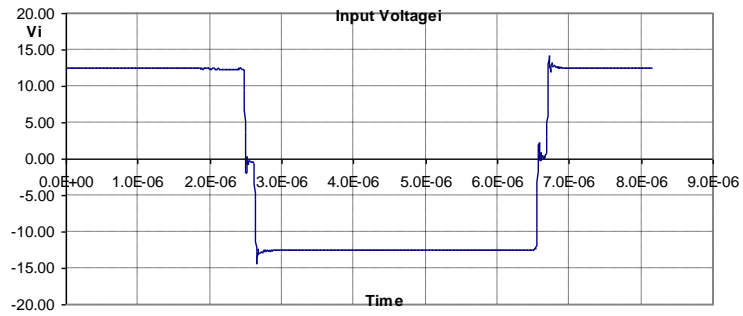


**Three more examples are shown below.**

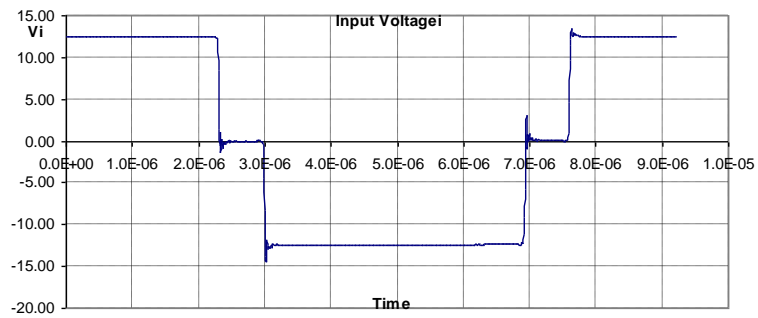
Expanded excitation, with an off-time of 4.2  $\mu$ s and  $V_i$  of 5 V.



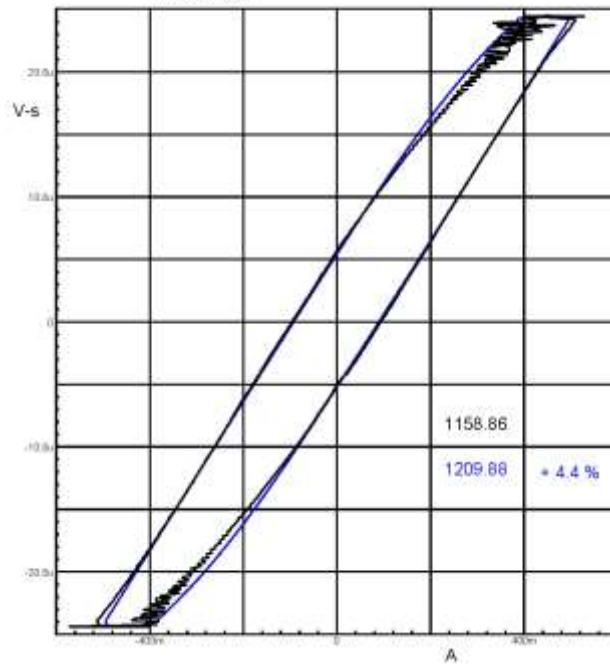
Expanded excitation, with an off-time of 0.125  $\mu\text{s}$ .

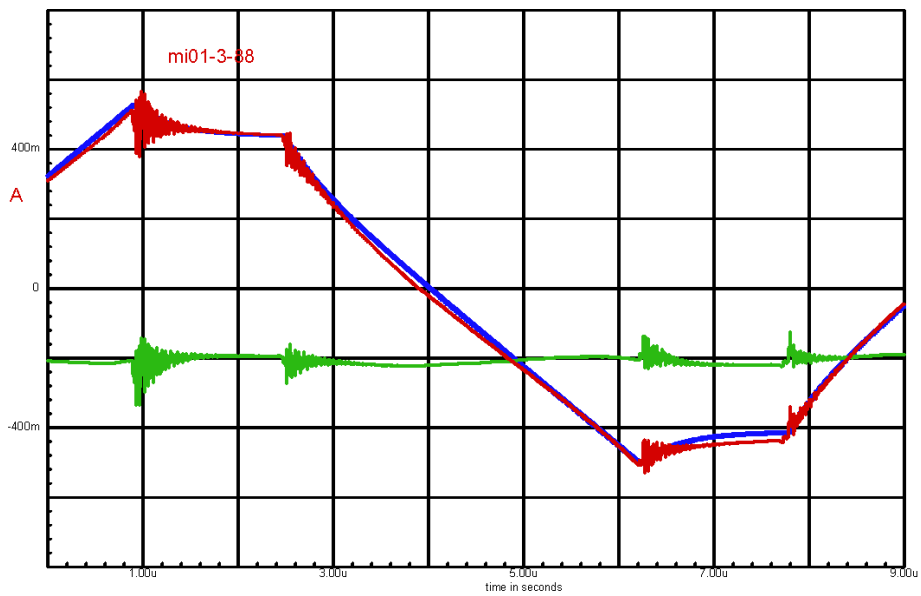
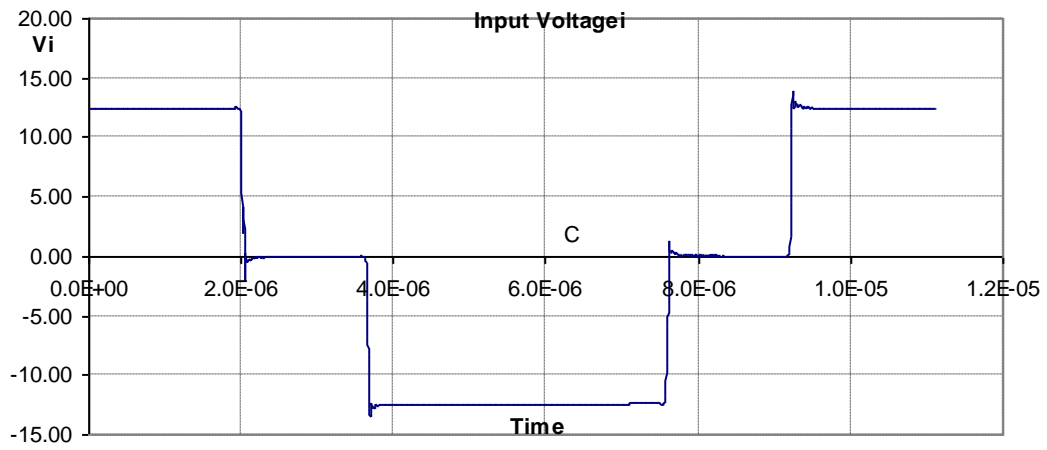


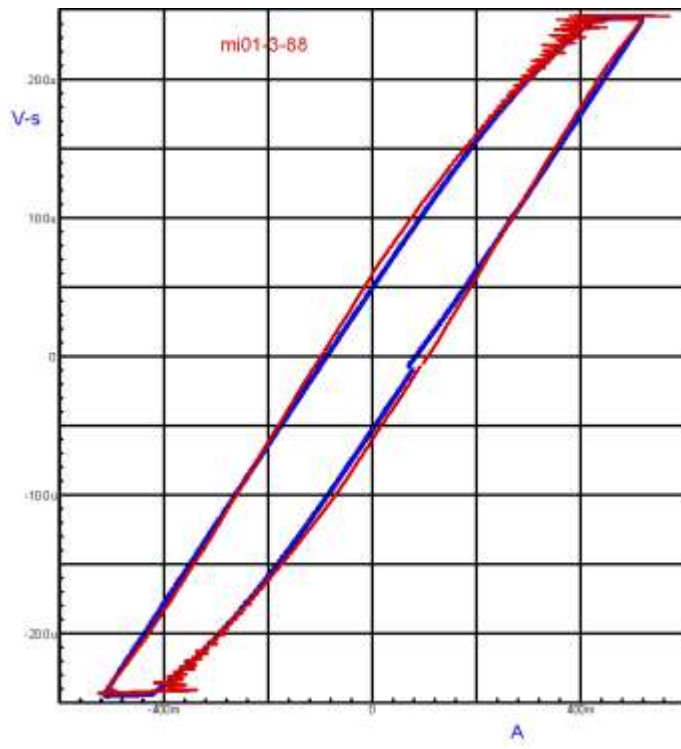
Expanded excitation, with an off-time of 0.67  $\mu\text{s}$ .

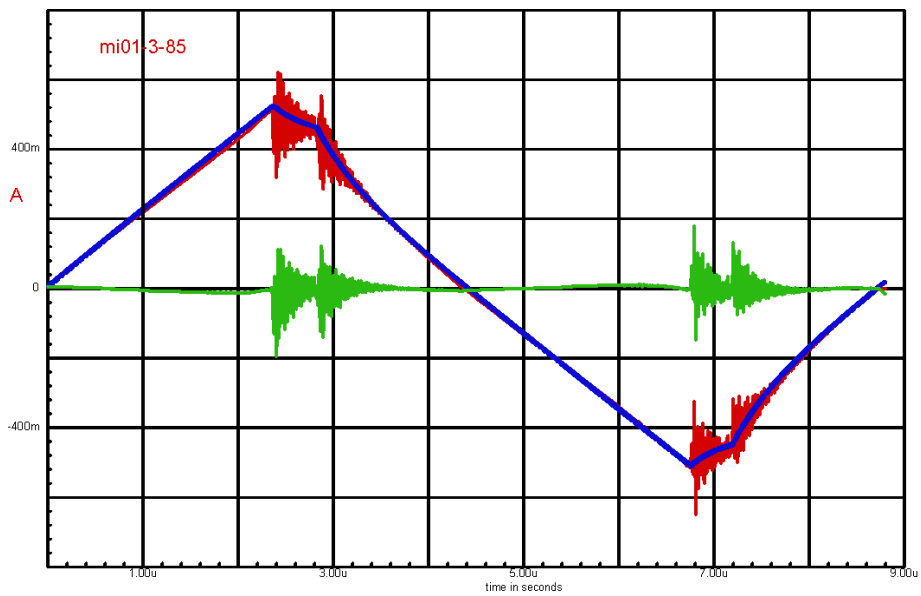
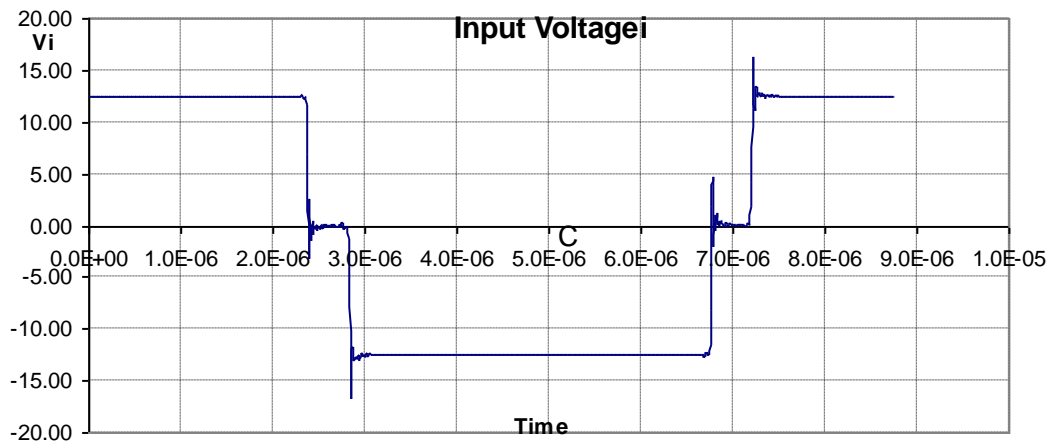


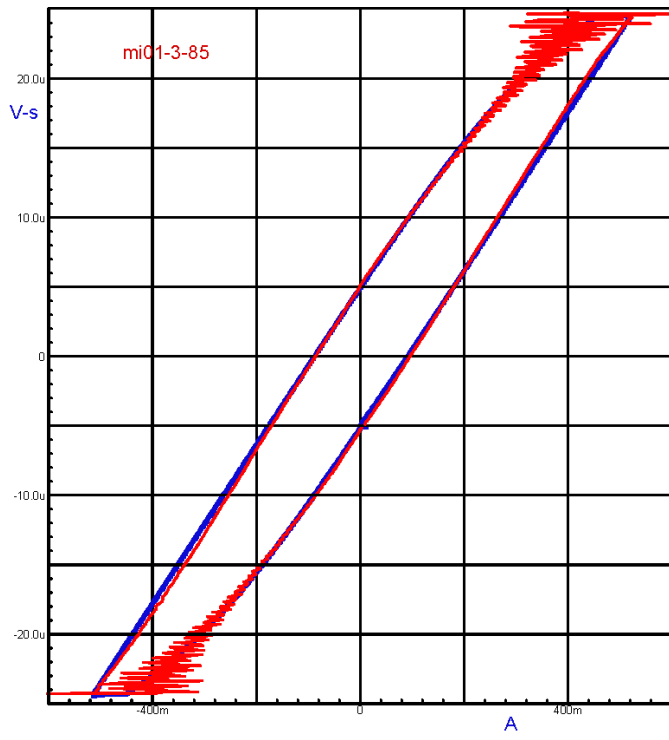
m01-3-086



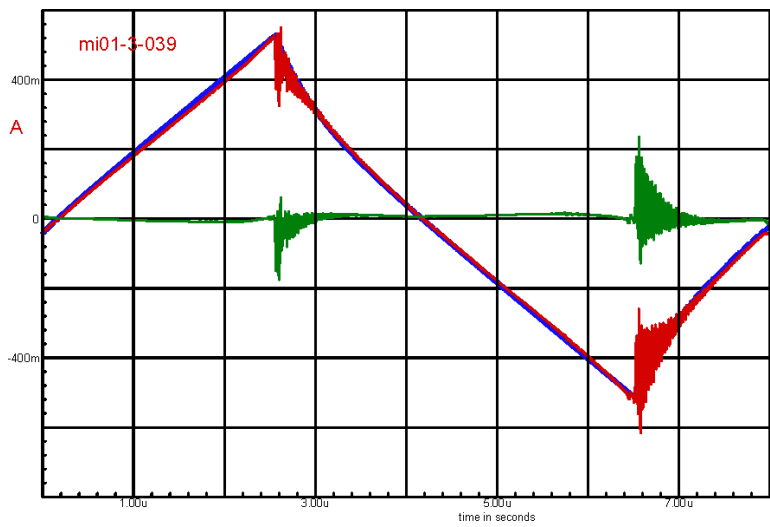
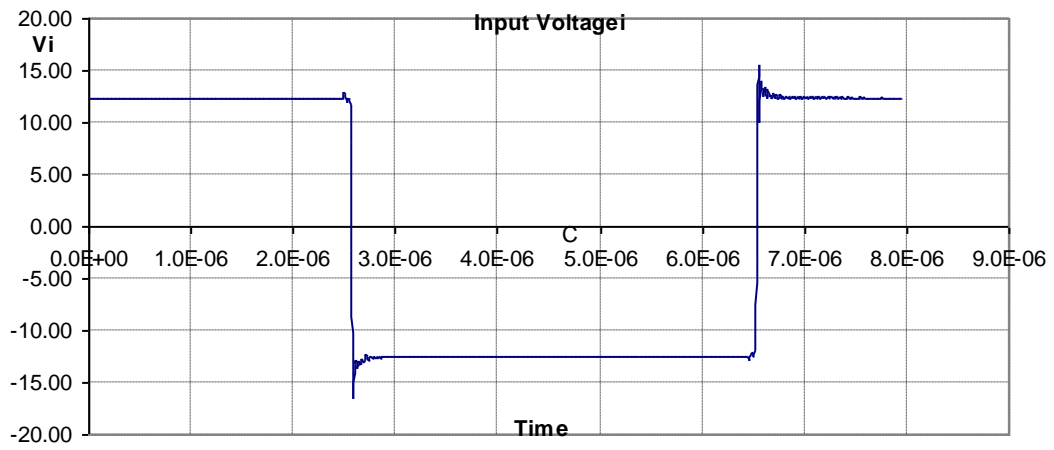


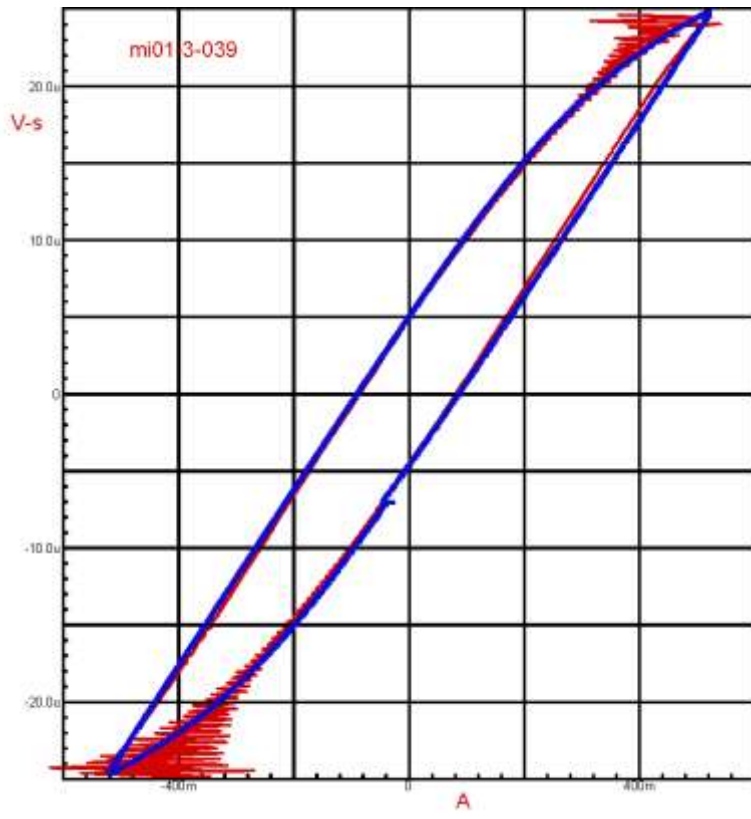












## ***Conclusions***

A simple SPICE model made with just four ideal components provides a reasonably accurate estimation of core losses for a variety of excitation waveforms. There is reason to believe that a more sophisticated model would match more closely with a wider range of inputs.

Building a SPICE model, reduced to basics, entails building an impedance model of the magnetic component. Impedance measurement and synthesis are familiar turf to power supply designers. Circuit response analysis really determining the impedance in both the time and frequency design. Feedback compensation and filter design both require impedance synthesis to specification.

The impedance model of a magnetic component is very likely to have components with voltage and flux dependencies; these are easily simulated using SPICE components.

Magnetic components also have a temperature dependence. If the input data includes a temperature measurement table, then the temperature coefficients can be simulated as well.

The reader is cautioned against trying to use the components of a SPICE model, no matter how good, to explain the physics of core loss or other phenomenon. It is, after all, only an elaborate curve fitting exercise.