IEEE Transactions on Nuclear Science, Vol. NS-33, No. 6, December 1986

NDRO CORE MEMORY SIMULATION USING RADSPICE™

J.R. Florian, S.P. Hardy and J.P. Retzler

SCIENCE APPLICATIONS INTERNATIONAL CORPORATION
San Diego, California

J.F. Bruder, QUADRI CORPORATION
Tempe, Arizona

ABSTRACT

Simulation of magnetic memories has been limited by the availability of effective nonlinear magnetic simulation techniques. Synthesis of an effective nonlinear magnetics model, its installation in RADSPICE™ and application to NDRO Magnetic Core Memories are described. The new tool allows simultaneous magnetic and semiconductor circuit simulation in the radiation inclusive code.

INTRODUCTION

Magnetic Core Memories compete with plated wire memories and special semiconductor memories (e.g. MNOS, CROSSTIE) for non-volatile, non-destructive readout (NDRO) applications requiring fast access times. Various versions of the SPICE electronics simulation code have been extensively used by integrated circuit (IC) designers and RADSPICE combines radiation inclusive device models with the efficient HSPICE version of the code [1]. Simulation of magnetic memories has been limited by the availability of effective simulation techniques for nonlinear magnetics. This paper describes the synthesis of such a model, its installation in RADSPICE and application to NDRO Magnetic Core Memories. Core Memory Technology (both NDRO and DRO) is a mature technology with two-decade-old patents and many existing, as well as potential, system applications. The model and methods are demonstrated through accurate simulation of an NDRO memory cell.

Since core memory technology marries nonlinear magnetics with semiconductor support electronics, a simulation code that effectively simulates both technologies is a valuable tool for the memory designer. The inherent radiation hardness of magnetic core storage media makes them candidates for applications requiring radiation hardness. In these cases the radiation effects models in RADSPICE will allow the designer to simulate the response of the entire memory element circuit including photocurrent, SGEMP and permanent degradation electronic responses. Such a tool will be valuable in characterizing NDRO Core Memory responses in nuclear and particle beam threat environments for advanced systems applications.

THE MODEL

The magnetics model implemented in RADSPICE is an improved version of the "Pheno" model [2]. The model provides a clean way of representing the hysteresis of B-H loop characteristics of toroidal magnetic cores by considering the rotation of magnetic dipoles into and out of the six directions of easy magnetization in the cubic structure of iron. The latching action of hysteresis is accomplished in the computer algorithm by keeping track of previous values of dipole orientations.

The main improvement on the original model involves the initialization procedure. Figure 1 illustrates the idea of the B-H curve found in transformer and choke data sheets from which the analyst abstracts data for computer input. In the original model implementation, the analyst had to input eight B-H values. Since all eight values overdetermines the model, the analyst frequently cannot get a consistant representation of the B-H curve he is trying to model. The problem is aggravated by the fact that the point identified as BM, HM is quite ambiguous on many data sheet B-H curves. Since this point represents the situation where all magnetic dipoles have been rotated to the easy direction (parallel to one edge of the iron cubic lattice) but not yet elastically rotated to the direction of the applied magnetizing force (saturation), the correct determination BM, HM is essential. The new model circumvents this problem by calculating the proper values of BM, HM in the initializing phase of the model implementation. Also eliminated is the need to input BCR. Hence the analyst may just enter five values from the B-H curve (the values without parenthesis in Figure 1). This initialization procedure in RADSPICE has proven to be trouble-free despite the variety of B-H curve shapes encountered in the workaday application of RADSPICE. The use of TC (core time constant), AC (core area), LC (core length) and LG (gap length) is the same as in the original model. The new model is still limited by the knowledge of the magnetic properties of the materials and reproducability of their B-H curve characteristics. For Core Memories individual core B-H properties are tested on individual cores prior to assembly.

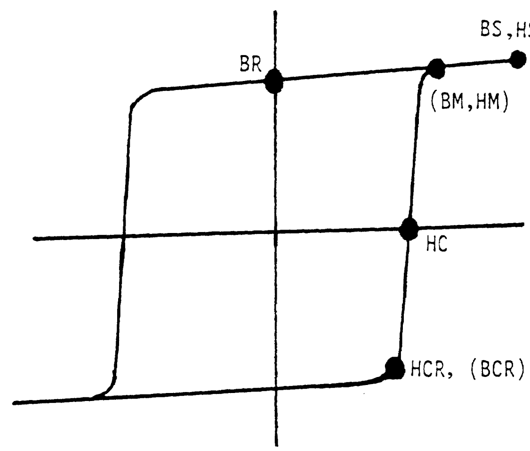

FIGURE 1. B-H Curve with RADSPICE™ Model Parameters

0018-9499/86/1200-1515$01.00 © 1986 IEEE

INSTALLATION IN RADSPICE

The model is simply a way of generating a value of B given a value of H by keeping track of the history of B and H. However, the model had to be interfaced to the circuit analysis algorithms inherent in RADSPICE. Since the code already had a linear magnetics model based on using the inductance L of the coils involved, the new nonlinear magnetics model was interfaced to RADSPICE by providing a determination of L (u) from the value of u (the instantaneous permeability) using u = dB/dH (the instantaneous slope of the B-H curve). This relation is:

$$L = uN^2A/l$$

where A is identified with AC, l with LC, and N is the number of turns on the core. The mutual inductance effects between coupled windings on the transformer have to be taken into account in the numerical implementation of the model into SPICE. Mutual inductance tables have to be built which can be used to calculate mutual inductance and flux terms between all coupled windings. Since RADSPICE poses no upper limit on the number of windings which can be placed on a transformer, these tables are built "on the fly" during simulation to suit the specific transformer configuration that is being simulated. Extreme care was exercised with the numerics in the incorporation of the model into RADSPICE. Since a SPICE style simulator uses a discrete linear approximation to linearize inductors and capacitors (thereby reducing a system of differential equations to a system of algebraic linear equations) there are several numerical pitfalls to avoid. The simulator time step controller has been modified so that transformer characteristics do not change too much from one time point to another. If this were to happen, it would destroy the local linearity approximation that is implicit in the SPICE algorithm. It is usually difficult for any circuit simulator to obtain convergence on circuits that have positive feedback. This is true even for circuits that are relatively linear. Since other principal uses for a transformer model are for circuits such as switching power oscillators and converters, special care had to be exercised in the numerics to preserve convergence. We also found that magnetic circuits are best solved using GEAR integration.

NDRO CORE MEMORY SIMULATION

RADSPICE has been used to effectively simulate various kinds of magnetic circuits, including the NDRO Core Memory Cell shown in Figure 2. As can be seen from the schematic diagram, a single two wire

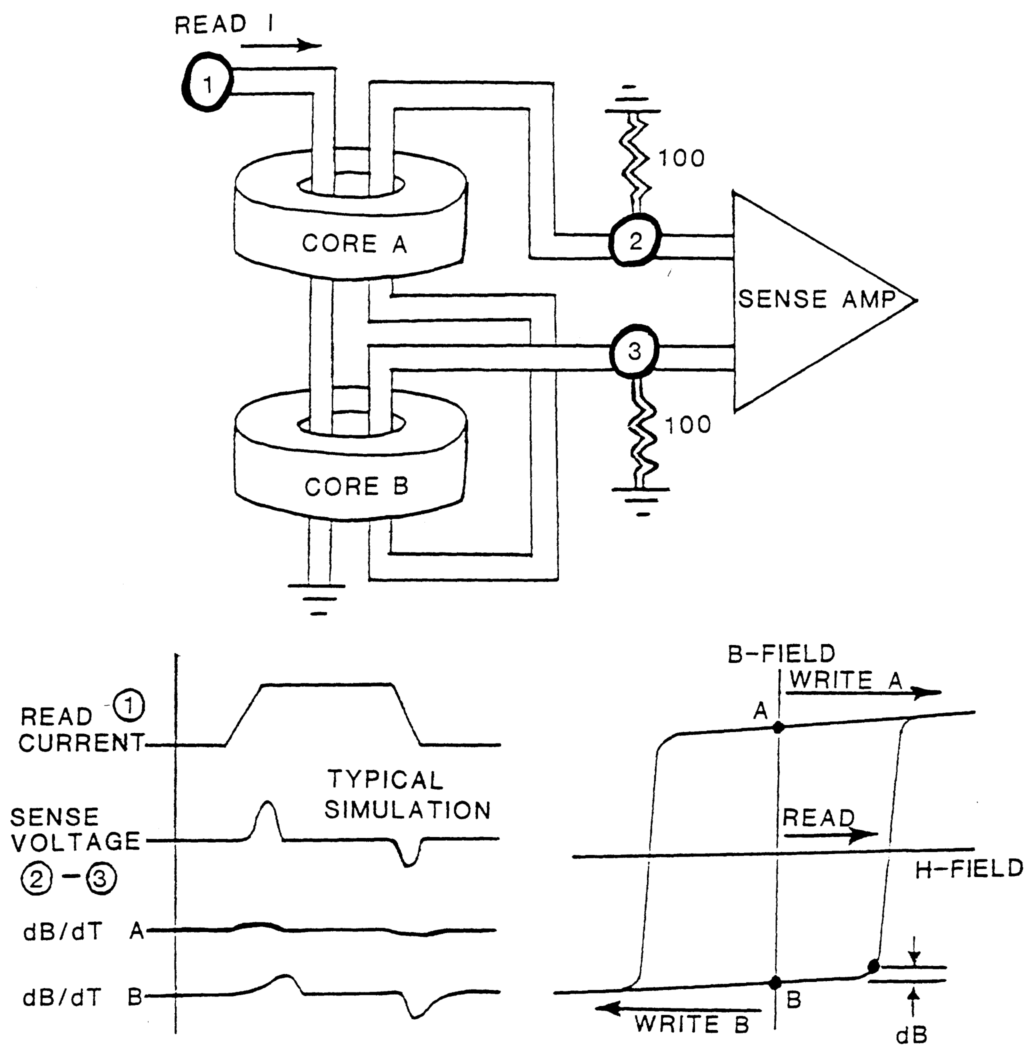

FIGURE 2. NDRO CORE MEMORY RADSPICE™ SIMULATION

NDRO Memory Cell is modeled. The pair of cores is written into the "1" or "0" state by driving current (using support circuits not shown) between Nodes 2 and 3, saturating the cores in opposite directions. This leaves Core A at point A and Core B at point B of the B-H curve in the storage mode. During a Read operation both cores are driven in the same direction (by current injected into Node 1) to the knee of the B-H curve for Core B and back toward saturation for Core A. The drive current is divided by Pi when there is not a complete transformer wire turn. The result is a differential voltage induced by transformer action on the sense line between Nodes 2 and 3 across the terminating resistors for reading by the sense amplifier. Pairs of cores per bit are typically used in a flux bucking arrangement to compensate for common mode noise sources in the core mat. These and other effects in a large core mat array can be modeled by an array of several cells with distributed capacitance and inductance on the drive and sense lines. More wires, cores or turns and the actual supporting semiconductor circuits can be added to model other configurations or radiation effects unique to a particular application or environmental specification. Complex circuit simulation capability including radiation effects is not new nor unique to RADSPICE and has been reported previously by others.

RESULTS

Figure 3 presents the results of RADSPICE simulation and actual measurement of the response of a 4-pair 13 mil NDRO Core Memory Cell. The particular core measured is a common core used in quantities of tens of billions in many applications and the simulation was scaled from a 1-pair run. There are a wide variety of available cores each with similar but unique magnetic properties, and the simulation parameters can be adjusted to match empirical measurement of the parameters.

The new magnetic model in RADSPICE has been also used to simulate magnetic oscillators, driven converters (linear and nonlinear) and various other magnetic circuits. In addition to printing and plotting normal voltages, currents, power, etc., RADSPICE allows the user to plot magnetic state variables such as flux, B-field, H-field, dB/dT, d(flux)/dT, incremental inductance, etc. The new model, in conjunction with RADSPICE, has been tested on various configurations involving combinations of chokes and transformers and found to produce accurate simulations of commercial circuits not seen in codes using the older models. Figure 4 illustrates a Magnetic Oscillator Simulation.

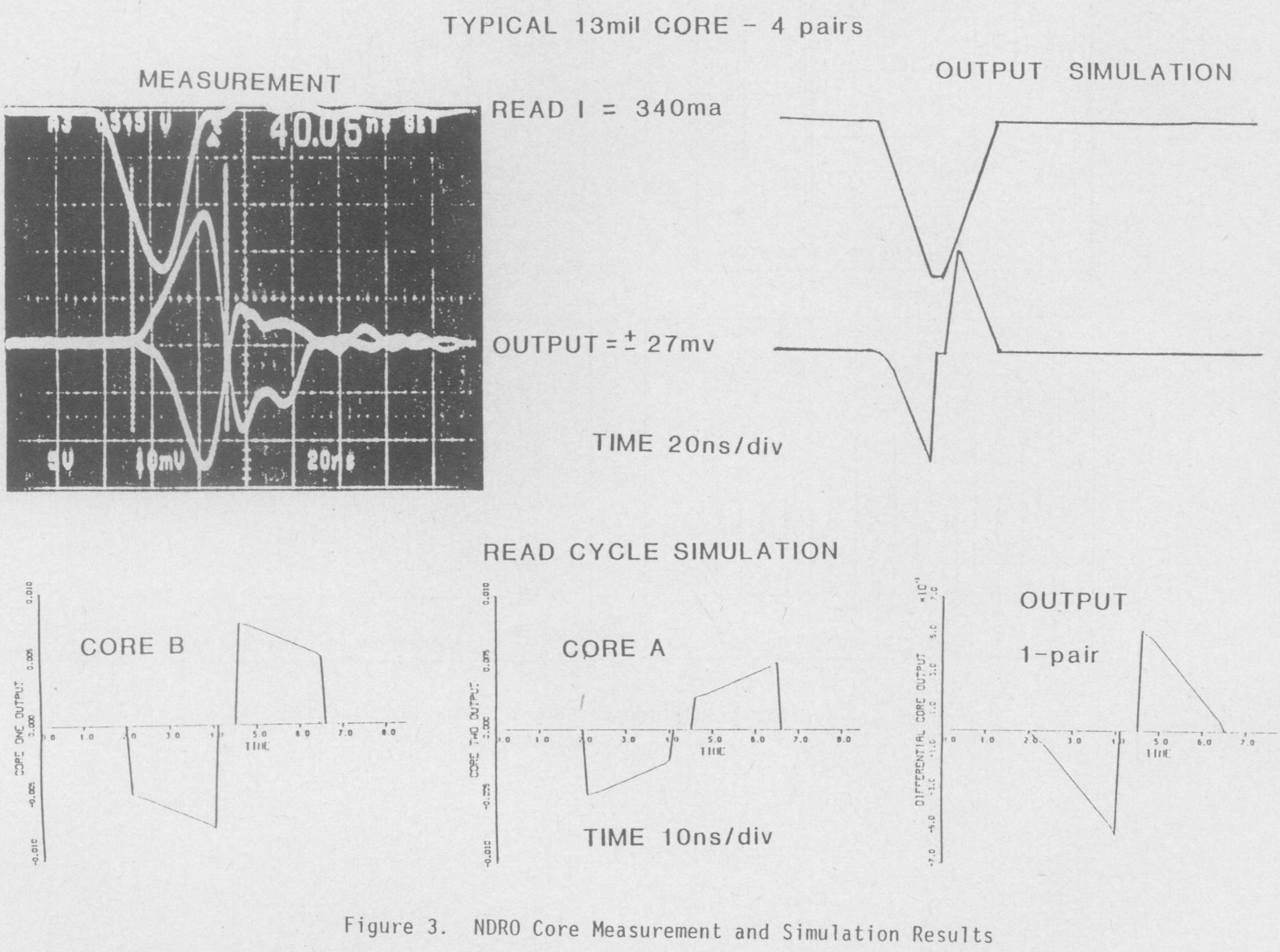

Figure 3. NDRO Core Measurement and Simulation Results

CONCLUSIONS

The non-linear magnetic simulation capability of RADSPICE adds to available repertoire of tools for the designer and analyst to assess electrical performance of NDRO Core Magnetic storage media. This capability coupled with radiation effects and complex circuit capability already resident in the SPICE based code provides a significant analytic assessment asset to augment empirical methods.

REFERENCES

1. RADSPICE™ Users Guide, SAIC and Meta Software, 1985.
2. W. H. Dierking and C. T. Kleiner, "Phenomenological Magnetic Core Model for Circuit Analysis Programs", IEEE Transactions on Magnetics, September 1972.

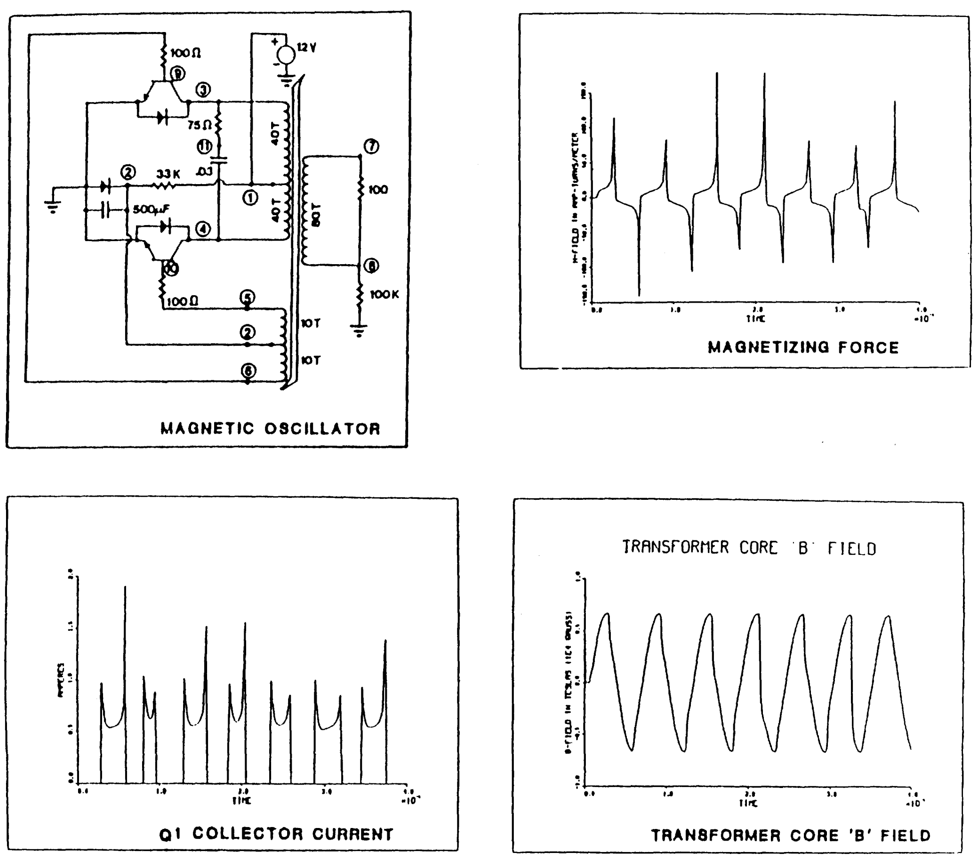

Figure 4. Typical Simulation of Complex Magnetic Circuit