Neural Networks Based Approach to Enhance Space Hardware Reliability

Ricardo S. Zebulum Anilkumar Thakoor Thomas Lu

Lauro Franco Tsung Han Lin S. S. McClure

Jet Propulsion Laboratory
California Institute of Technology
4800 Oak Grove Drive Pasadena, CA 91101
e-mail: rzebulum@jpl.nasa.gov

Key words: Neural Networks, device modeling.

Abstract: This paper demonstrates the use of Neural Networks as a device modeling tool to increase the reliability analysis accuracy of circuits targeted for space applications. The paper tackles a number of case studies of relevance to the design of Flight hardware. The results show that the proposed technique generates more accurate models than the ones regularly used to model circuits.

1. Introduction

This paper provides a novel approach to the problem of reliability in circuit design, which specifically addresses the challenges of hardware design for space applications or Flight hardware. Flight hardware design cycle is deeply impacted by the analysis of electronic circuit behavior across the specific Mission's "environmental extremes". Particularly, as a critical step in the design process, the Worst Case Analysis (WCA) of electronic hardware provides a quantitative assessment of circuit performance, accounting for the manufacturing process variability and the unavoidable dimensional tolerances as well as the mission-specific environmental, aging, and radiation effects. The WCA ascertains flight hardware quality and reliability, while determining its operational limits and margins. Essential for the WCA is the availability of accurate SPICE models for electronic components used in the circuits.

Device modeling requires an understanding of device physics in conjunction with a thorough knowledge of the circuit. However, detailed device models are rarely available from the manufacturer, and usually have limited accuracy. An accurate and detailed SPICE model development for a device from generally a limited set of available (measured) characteristic/behavior of the device under a set of controlled environmental conditions is an involved, labor-intensive process, and typically results in a cost of several thousand dollars per component. The cost of the comprehensively modeled electronics to a mission therefore would become enormous considering that hundreds of different components are used on each circuit board, and a large number of different boards are typically required for each mission. Furthermore, SPICE models cannot be easily re-used across different missions due to differences in mission environmental requirements. Due to cost and time constraints, a full suite of accurate models are rarely generated during reliability analysis effort, which often results in an extended design time due to design changes from Prototype to Flight.

This paper presents an Artificial Neural Network (ANN) [1] autonomous modeling tool for electronic components that for the first time produces and delivers accurate SPICE models of electronic devices at a variety of environmental conditions from limited or incomplete test data, as needed by a given mission scenario. This method was applied in the modeling of a variety of devices typically used in flight circuits, such as diodes zeners, voltage regulators and bipolar transistors. As opposed to built-in/regular or manufacturer provided models based on device physics, ANNs produce behavioral model consisting of non-linear equations. The results show that ANNs produce a more accurate model than the ones conventionally used: the error between simulation and actual behavior decreasing from approximately 30 % or more to less than 5% in some cases. Moreover, ANN modeling also allowed the capture of critical circuit limitations that were ignored when using conventional or manufacturer provided SPICE models.

2. ANN Approach for Device Modeling

Artificial Neural Networks (ANNs) are employed as a modeling tool to generate SPICE simulation models for electronic components. ANNs can be trained to learn non-linear relationships from corresponding measured or simulated data, even if they are incomplete. This paper particularly focuses on the reliability challenges of circuits for space applications, where ANNs generate simulation models that account for variables not accurately modeled conventionally, mainly temperature, initial device tolerance and radiation effects of Total Ionization Dose (TID). Previous work [2, 3] also proposed the use of ANNs to generate SPICE models for electronic devices; however, this article is the first to include environmental variables as inputs to the modeling tool; and also to use ANN modeling to specifically tackle circuit reliability issues.

During a mission, circuit components will be subject to different levels of radiation and temperatures; the component response will also be probabilistically affected by aging and random factors. ANNs can efficiently capture the device response under different conditions from real measurements, Worst Case Database (WCDB) specific for the particular Mission environment; and screening data from manufacturer. Moreover, this paper shows examples where ANN produces tunable models that allow the designer/analyst to simulate a device for Nominal, Worst Case Minimum and Worst Case Maximum conditions.

After successful training, the ANN will produce a device model that can be incorporated into SPICE netlist using specific built-in functions for behavioral modeling: voltage controlled and current controlled sources. The ANN derived model is defined as a set of non-linear equations with real-valued coefficients or weights. These mathematical models are seamlessly used to define the controlled sources. Figure 1 provides an example of a typical ANN model equations as written in SPICE format.

```
.subckt ann_diode 100 200

Gdiode1 100 200 VALUE =  {(V(95) )}

E1 1 0 VALUE = {((V(100,200) + 5)/10 )}
V2 2 0 0.0V
E21 21 0  function 0.757871 +(-10.335355 *
V(1)) +(-0.801717 * V(2))
E22 22 0  function -15.304038 +(27.702725 *
V(1)) +(0.794741 * V(2))
E81 81 0  function -17.333016 +(-8.673347 *
(1/(1 + exp(-V(21))))) +(10.524507 * (1/(1
+ exp(-V(22)))))
.ends ann diode
```

Figure 1: Example of a typical ANN model transported into a SPICE controlled-source.

3. Case Studies

This section describes three relevant case studies for circuits and devices used in space applications. A Multi-Layered Perceptron (MLP) using Backpropagation [1] training method was used in all applications. The training data was prepared by a combination of real device measurements and calculations from Mission specific Worst Case Database [5], which considers the exact environmental conditions (temperature, radiation, duration) of a particular project.

3.1 Zener Modeling

Using the correct SPICE model of a Zener diode is critical for a circuit worst case analysis. This section provides a real example where an ANN model is used to improve the simulation fidelity and detect a reliability issue not identified when using regular SPICE model. A simplified diagram of the circuit is shown in Figure 2. The Figure shows a power switching circuit used in a Flight circuit design that employs the 3.9V (V_Z) Zener.

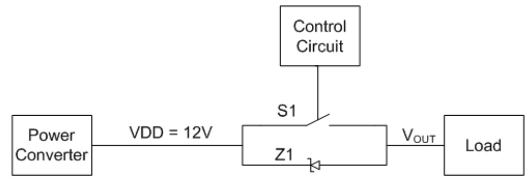

Figure 2: Simplified diagram of power switching circuit using 3.9V Zener.

Depending upon a control circuit, a load will be powered either by the switch S1 or by the Zener Z1 signal paths. The value of the voltage V_{OUT} delivered to the load is critical to the circuit overall functionality. When S1 is in OFF state, V_{OUT} = 12V – V_Z. Figure 3 depicts an actual measurement of the I x V transfer curve for the 3.9V Zener, which shows that the device presents a "soft knee" in the neighborhood of V_Z. For the particular application in Figure 2, the device operating current is approximately 30 mA, which corresponds to an operating voltage around 3 V as of graph in Figure 3. However, the SPICE Zener regular model generates a "hard knee" I x V in the neighborhood of V_Z, as also shown in Figure 3.

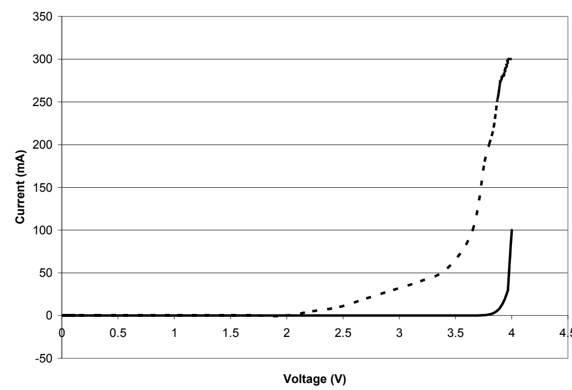

Figure 3: Measured (traces) and SPICE model (full line) I xV transfer curve for 3.9V Zener.

The SPICE model shown in Figure 3 produces the expected voltage output of approximately 8.1V (12V – 3.9V) to the load represented in Figure 2, whereas the measured value is 9V, since the actual value of V_Z is 3V instead of 3.9V. This modeling inaccuracy caused an erroneous behavior of the circuit in the second stage.

A 3-layer backpropagation ANN was employed to replace the SPICE physical model. The ANN topology is shown in Figure 4. It consists of 2 x 4 x 1 topology, i.e., 2 input neurons, 4 hidden neurons and 1 output neuron. As shown in the Figure below, this ANN models the Zener current I_Z as a function of two inputs, temperature and voltage applied to the Zener V_{IN}. The ANN training data was collected both by measurements and also by data from a Mission specific Worst Case Database, the latter being used to provide accurate data on the diode temperature dependency.

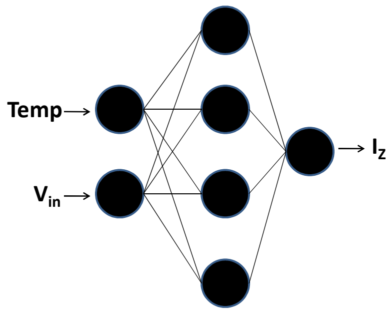

Figure 4: ANN Topology employed for 3.9V Zener Modeling

The ANN model captures the actual device transfer function and the "soft knee" very accurately, as shown in Figure 5 at 25C.

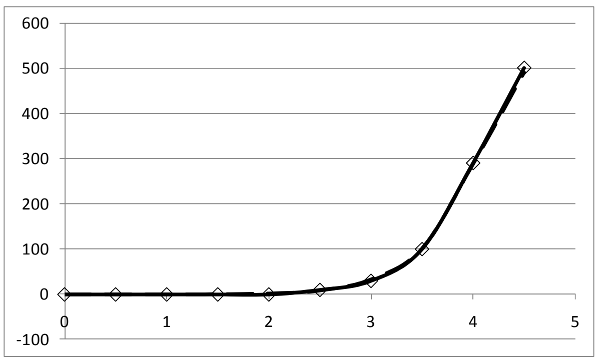

Figure 5: Comparison between ANN (trace) and actual (full line) device responses at 25°C. Y axis is current in mA; X axis is the Zener input voltage.

As the authors included the ANN behavioral model in the circuit netlist depicted in Figure 2, the SPICE simulation produced the correct output voltage of 9V, i.e., V_Z equal to 3V at 30mA load. The Zener was later replaced to fix this issue.

3.2 2.5 V Voltage Regulator Modeling

Voltage regulators are commonly used in space applications. Since this device is susceptible to Total Ionization Dose (TID), it is critical that the reliability analyst accounts for this effect: even small deviations from the nominal value of 2.5V may cause the circuit to fail unexpectedly. An ANN was trained using actual measured data for this device for different radiation levels (up to 1Mrad of TID); and also different regulator currents I_{REG} (from 200 uA to 10 mA). An extra binary input was also applied to the ANN, representing the state, powered or unpowered, of the device during irradiation. Figure 6 depicts the ANN topology.

Figure 7 compares the response between the ANN and the measured regulator output. Each training sample as indicated in the X axis consists of a different combination of regulator current and TID level. Four devices were used for ANN training, serial numbers SN2, SN3, SN4 and SN5. The graph in Figure 7 clearly shows a difference in behavior between SN2/SN3 and SN4/SN5: SN1 and SN2 were unpowered during irradiation (*Bias* = 0); and SN4 and SN5 were powered (*Bias* = 1).

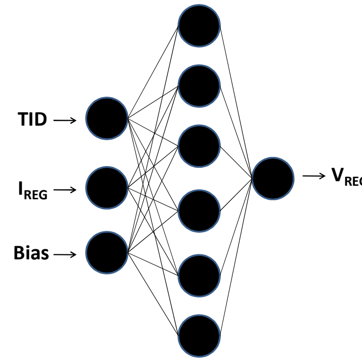

Figure 6: ANN Topology for 2.5 Regulator Modeling.

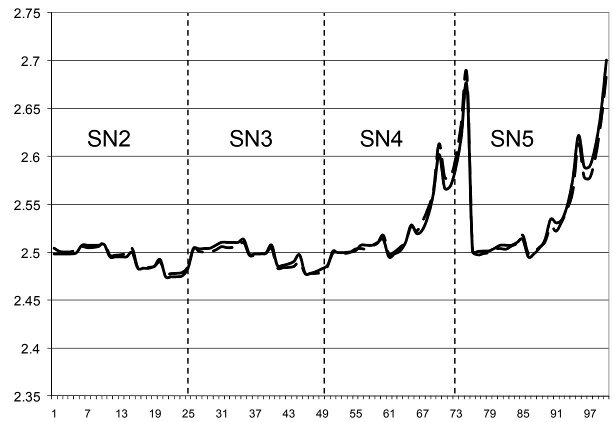

Figure 7: ANN Outputs (trace) against actual device output (full line). Y axis shows the regulator voltage output in Volts; X axis is the sample index.

After training, the maximum error between ANN output and each training sample was 0.58 %; and the average error was 0.15 %. The ANN model was later incorporated into a netlist representing an actual Flight circuit, the simulation running successfully.

3.3 Bipolar Transistor

Bipolar transistors are also widely used in space related applications. Regular or manufacturer provided SPICE models do not capture variations of some relevant transistor parameters with temperature and radiation. These models also do not incorporate initial tolerance and End-of Life (EOL) effects. The following parameters have been selected as the most critical for the circuits investigated: *Base-emitter junction voltage, V_{BE}*; and t*ransistor gain, β*.

Instead of using a single ANN to model a transistor, this paper approach uses two different ANNs to model each of the above parameters. This is accomplished by using the Ebers-Moll transistor model [4]. The base-emitter junction characteristics is modeled through diode D1 using this model; and the transistor gain β is modeled through the controlled current source from collector to the base. The impact of these parameters is discussed in the next sections.

3.3.1 Modeling *Base-Emitter Junction*

This section describes the ANN base-emitter junction model, which is similar to ANN diode modeling. Particularly, an accurate model of the voltage $V_{BE,sat}$ is critical for applications where the transistor is used as a switch (either cut-off or in saturation), as shown in the simplified block diagram in Figure 8. If the maximum low-output value V_{OL} of the digital buffer driving the bipolar transistor exceeds $(V_{BE,sat})_{Min}$, then the transistor can turn-on unexpectedly, possibly causing equipments to turn-on accidentally. This is just an example where an inaccurate SPICE model for the BE junction can produce an error in the circuit WCA.

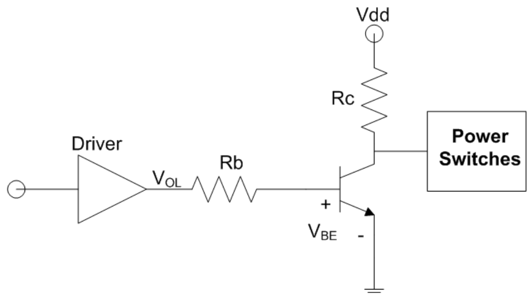

Figure 8: Simplified diagram of a typical power circuit where the bipolar transistor acts as a switch.

The value of $V_{BE,sat}$ strongly varies due to the device initial tolerance, a factor that is not usually captured by built-in/regular SPICE or manufacturers models. In order to illustrate this variation, calculations for a specific transistor establishes $V_{BE,sat}$ between 0.47V and 1.4V [5] when accounting for initial tolerances. On the other hand, built-in simulation models produce a variation between 0.8V and 1.0V, corresponding to an error of at least 30 % to the targets. This section presents an ANN behavioral model that can be tuned by the user to produce Minimum, Nominal or Maximum values of $V_{BE,\ SAT}$, therefore enveloping all possible behaviors for this parameter. A compact ANN (2 x 4 x 1) was used: it receives two inputs, the input voltage applied at the junction, V_{BE}; and the initial tolerance *Tol*. The latter is used to produce a configurable model, defined as: Minimum V_{BE} when *Tol* equal 0; Nominal V_{BE} when *Tol* equals 0.5; Maximum V_{BE} when *Tol* equals 1.0. The ANN output is the BE diode current I_E. Figure 9 depicts the training profile.

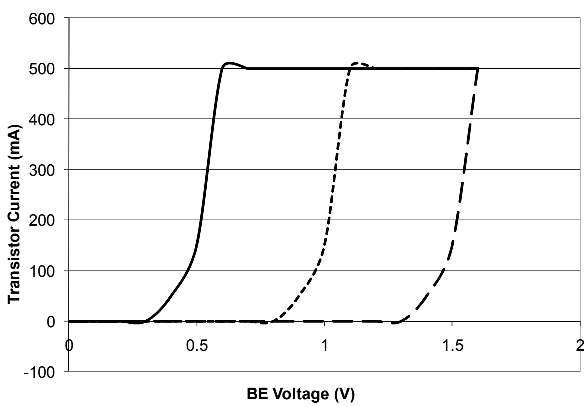

Figure 9: BE junction ANN Model: Training Profile. From left to right, Minimum, Nominal and Maximum curves.

The training profile encompassed the three cases described above. The $V_{BE,SAT}$ parameter corresponds to the V_{BE} value at the threshold of the BE diode conduction. The ANN model was incorporated into a SPICE netlist as described in Figure 8; and the results are shown in Figure 10.

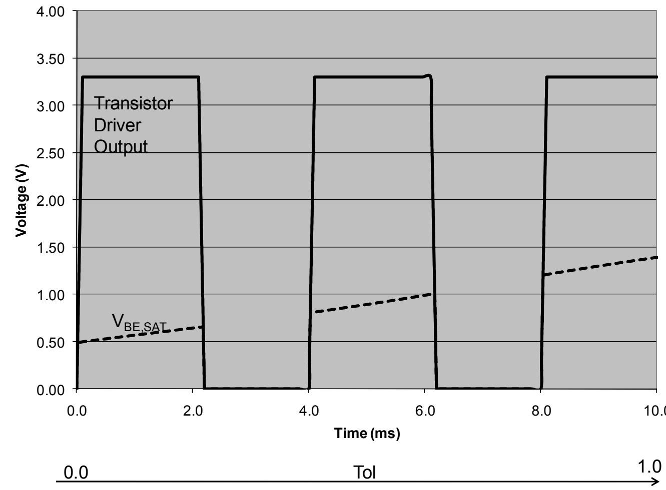

Figure 10: Simulation results for ANN base-emitter junction transistor model.

Figure 10 illustrates the circuit response as the driver output switches between 0 and 3.5V, driving the transistor in and out of saturation; and the tolerance input slews from 0.0 to 1.0. It can be seen that the ANN model produces $V_{BE,SAT}$ ranging from 0.5V (*Tol* = 0) and 1.4V (*Tol* = 1.0), which approaches the real behavior of the transistor.

3.3.2 Modeling β

An ensemble of three ANNs are used to model the value of the transistor gain β, which is a function of the temperature, Total Ionization Dose and transistor current. The three ANNs respectively generate Nominal, Minimum and Maximum values for the gain. The ANN consists of 3 x 6 x 1 topology, the inputs being Total Ionization Dose (TID), transistor current (I_C) and temperature (Temp); and the output being the transistor gain β.

This paper used, as ANN training data, actual measured values for β at different temperatures (10 °C to 30°C), TID effects (up to 1Mrad) and transistor current conditions (0.5mA to 150 mA) for the a specific commercial device. The data is further split into three sets based on the initial tolerance: Nominal, Minimum and Maximum. The initial tolerance is provided by a Mission specific Worst Case Database[5] and has a large impact in the transistor gain: for example, the gain changes from 100 to 400 at 10 mA as a result of tolerance effects only. The three ANNs provide the designer/analyst with three transistor models enabling a thorough simulation of the circuit design corners.

The 3-layers ANN was able to map the training data relatively easily: the training time was less than 5 minutes and the average error was 5.7 %. Figure 11 plots the ANN response against the measured data (target) for the Nominal gain. Similar results were achieved for the Minimum and Maximum cases. The graph vertical axis shows the transistor gain and horizontal axis shows the sample index. A total of 64 samples were used to train the ANN, each sampling representing a different combination of temperature, TID and transistor current values. The ANN was also able to respond to inputs not shown in the training phase, with a similar error margin as the one for the training phase.

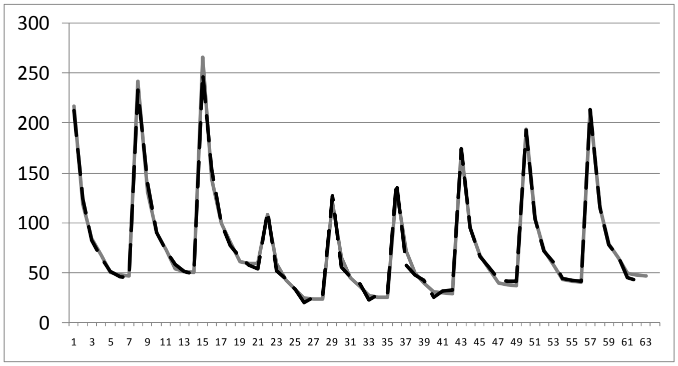

Figure 11: Target (full line) and ANN output (traces) for transistor gain β**.** Vertical axis shows the Gain values and horizontal axis shows the sample index.

4. Convergence Issues

As of the cases studied above, the Neural Networks can easily be trained to capture the transfer function of the devices. Instead, the main challenge resides in the incorporation of ANN behavioral models into a SPICE. Normally, SPICE may produce convergence problems as the ANN behavioral models get more complex, thereby constraining the ANNs to a compact size. The number of neurons in the hidden layer was therefore a critical parameter in the cases above: too many neurons led to SPICE convergence issues; too few neurons led to a poor ANN performance in terms of error to the target behavior. It was verified that, by keeping the ANN hidden layer at twice the size of the input layer, we avoided convergence issues without degrading the ANN performance. Besides convergence issues, a large number of neurons in the hidden layer (e.g., more than twice as in the input layer) can cause generalization problems, i.e., the ANN may respond poorly to inputs not shown in the training phase. Whenever replacing a physical by a behavioral model, care must be taken to ensure that the behavioral model does not contain singularities that could cause a convergence problem.

Another potential pitfall observed for diode models refers to its leakage current, which is typically in the range of nano-amps. The ANN may produce a model that shows a leakage current at the micro-amp level. Depending on the way the training data is normalized, this kind of discrepant model may still show a very small error to the target, but will produce an erroneous operating point when integrated into a circuit in SPICE.

Another strategy to overcome potential convergence problems is to change the grouping of the ANN equations, particularly to avoid overflow errors. This is accomplished by consolidating one or more neuron output equations into only one circuit node, as opposed to having one circuit node associated to each output neuron equation as shown in Figure 1. Overflow errors usually happened when the ANN exponential functions returned very large numbers, normally due to the fact that the SPICE algorithm would try an initial solution too distant from the circuit operating point.

The above strategies were followed in addition to standard SPICE recommendations to overcome convergence issues, such as ramping up the power supplies and reducing the time step of transient analysis.

5. Conclusions

This paper demonstrated the use of Neural Networks as a modeling tool to deliver accurate SPICE simulation of circuits. This modeling tool may have a major impact in the reliability analysis of aerospace related circuits, where accurate characterization of the devices under extreme environmental conditions (temperature, radiation, etc) is critical to the success of the project [6]. This paper also identified specific cases where the use of ANN behavioral models flagged worst case violation not normally flagged when using regular or built-in device models.

6. References

[1] – Rumelhart, D., McClelland, J., "Parallel Distributed Processing: Explorations in the Microstructure of Cognition", Vol. 1, Cambridge MA, MIT Press.

[2] – Hammouda et Al, "Neural Based Models of Semiconductor Devices for SPICE Simulator", American Journal of Applied Sciences, 5 (4):385-391, 2008.

[3] – Cheknane et al, "SPICE Implementation of Neural-Based Models for Solar Cells Under Solar Concentration", International Review of Physics, Vol. 1, N. 3, August, 2007.

[4] – Schilling and Belove, "Electronic Circuits: Discrete and Integrated", McGraw-Hill Inc., 1979.

[5] – Fettig, R., "JPL Worst Case Part Variation Database", December, 2007.

[6] – Worst Case Circuit Analysis Workshop, AEi Systems, August, 2009.

Acknowledgements

This research was carried out at the Jet Propulsion Laboratory, California Institute of Technology, under a contract with the National Aeronautics and Space Administration and funded through the internal Research and Technology Development program.