Wesley Cardone, N8QM

10626 Coopersfield Rd., Chelsea, MI 48118; **n8qm@arrl.net**

Precision Generic Diode Characterization for Simulation

This CW audio filter demonstrates the digital signal processing capability of a microcontroller.

The characterization of a Simulation Program with Integrated Circuit Emphasis (SPICE) diode can have high fidelity reproducing, in computer simulation, its published current-voltage (IV) profile for its reference ambient temperature. However, a loss of fidelity will be observed with ambient temperatures differing from the reference temperature (TNOM). Therefore, industry-released device-specific model characterizations are designed to satisfy a wide range of ambient temperatures by uniformly distributing the error. This paper will explain the nature of the SPICE diode model and how to solve for its dc characterization parameters given a published reference ambient temperature profile. A simulation diode model so characterized will virtually superimpose itself over the published IV profile for the reference ambient temperature.

Background

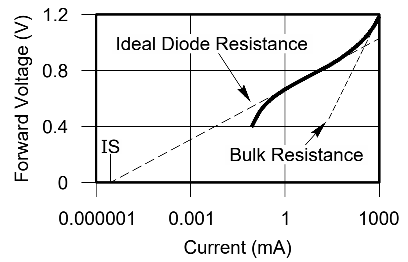
SPICE was originally developed by the University of California, Berkley as a class project in 1969-70 and was released into the public domain in 1972 as Version 1. In 1975 version 2g.6 was released, which became the backbone of analog circuit computer simulation on a global scale though only available by mainframe.

With the release of the Intel 80486 processor in 1980, with its 32-bit architecture, personal computers became powerful enough to accommodate SPICE without special AT bus boards. Many companies embedded public domain SPICE 2g.6 within their own graphical user interfaces selling the software as a sophisticated analog circuit simulation tool. But while SPICE 2g.6 was quite sophisticated in its numerical processing for analog design, characterizations for discreet analog components such as bipolar junction transistors and diodes were virtually non-existent. There were a number of companies that developed SPICE libraries for sale at very high prices. High prices, of course, effectively made them unavailable for hobbyists and companies on limited budgets.

In 1999, Linear Technology released its first version of LTSpice, which is free software originally developed by Linear Technology. An LTSpice download may be obtained at the **www.analog.com** website. Upon entry to that website, enter the search terms "ltspice download" for Microsoft Windows. The current release of LTSpice (17.0.32.0) has had many enhancements and new capabilities together with a very extensive professional library of discrete semi-conductor components. Virtually all of today's desktop and laptop computers (Mac and Windows) are capable of running LTSpice. The sophistication of SPICE for the hobbyist has arrived.

The Spice Diode Model

The diode model found in LTSpice, though it has many enhancements relative to SPICE 2g.6, still includes the original functionality of the SPICE diode model. In the following text we will first look at what physical performance to expect from a silicon diode under controlled environmental conditions and then compare those with the SPICE diode simulation model.



QX2303-Cardone01

Figure 1 — Illustration of the physical nature of the silicon diode with reference to its regions of operation: recombination; ideal; and high injection. The data of this figure were taken with the device under test immersed in a water bath at a controlled 25 °C temperature.

Figure 1 depicts the fundamental dc functionality for the physical silicon diode's current-voltage (IV) profile. The curve represents measurements taken for this paper using a 1N914 diode immersed in a 0 °C water bath. For a given diode forward voltage (linear Y-axis) there is a dependent current (log X-axis). There are two regions of interest for simulation and design purposes described below. In **Figure**

1, the wavy curve illustrates the physical IV performance for the physical 1N914 silicon diode through all its regions at 25 °C. There are two regions of interest for both practical design and therefore simulation: ideal and resistive. Note that the physical curve departs from either of the two straight lines at two places. The departure that occurs for forward voltages less than 0.6 V is not directly accounted for by the SPICE diode model and neither is it of interest for practical design purposes. However, this region of operation (with its very low forward voltages) is roughly accounted for by means of the *GMIN* (minimum conductance) parameter described later in this paper.

SPICE uses the Shockley diode equation to solve for a forward voltage given a current and ambient temperature:

$$V_f = N\,V_t \log_e\left\{\frac{i}{IS}+1\right\} + i\,RS + V_f GMIN$$

William Shockley of Bell Telephone Laboratories developed the IV (current-voltage) characteristic equation representing an idealized diode for either forward or reverse bias applications.

Within this equation there are three foundational parameters governing a diode's dc performance — *N* (emission coefficient), *IS* (saturation current), and *RS* (bulk resistance). A thermal voltage, V_t, is a function of the environment and cannot be manipulated by the user. There is a minimal conductance, *GMIN*, that governs the diode at very low forward voltages, V_f. Here the product (V_f *GMIN*) will begin to dominate the above equation.

The most significant region in **Figure 1** is the ideal. Here, the diode's PN junction follows a semi-log path as a function of the forward voltage. The "ideal diode region" line covers the device ideal mode operation but for illustration purposes extrapolates into the adjacent regions. Observe the intercept point where the ideal curve extrapolation crosses zero volts. The corresponding current represents the diode's saturation current and is marked in **Figure 1** by the solid wavy line. The simulation would return an *IS* current for a zero voltage drop except for the minimal conductance parameter, *GMIN*. All device models in SPICE have a *GMIN* parameter to circumvent numerical convergence issues associated with zero.

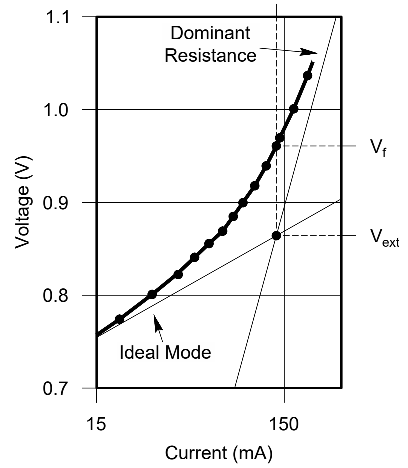
Crucial to understanding the premise of this paper, the saturation current *IS* has a dependence on the ambient temperature. As the ambient temperature increases, the saturation current also increases, given a forward voltage. In addition, as the ambient temperature increases, the emission coefficient (*N*) changes slightly. The sole purpose of the *XTI* parameter (in conjunction with *N*) is to move *IS* to make it track the ambient temperature. In practice, little change results from *XTI*. For this reason, *XTI* is not identified in this paper as a foundational dc characterization parameter.

The important observation is that the SPICE diode model moves the curve in the correct direction to track a changing ambient temperature. However, the accurate quantification of that change is only rudimentary with respect to published profiles leaving much to be desired. This accounts for a need to characterize a given device for a best approximation where a wide range of ambient temperatures is to be allowed for.

Viewing **Figure 1**, consider the region where resistive effects are dominant (bulk resistance line). The SPICE diode model accounts for this region with a linear bulk resistance as illustrated graphically in **Figure 2**, dominant resistance line given by:

$$RS = \frac{V_f - V_{ext}}{i_d}$$

The bulk resistance (illustrated in **Figure 2**) serves as supplemental voltage drop to the ideal drop. At relatively low currents, the bulk resistance drop is not significant relative to the ideal drop. However, as current increases further, the voltage drop from the resistance begins to become measurable as a supplement to the diode ideal drop. At this point there are two voltage drops being summed together. Both drops continue to increase with increasing current though the resistive drop will eclipse the ideal drop with further increases in current.



QX2303-Cardone02

Figure 2 — Graphic illustration on obtaining the bulk resistance from the published IV profile.

The key to understanding how the SPICE diode model's bulk resistance parameter is developed, is graphically shown in **Figure 2**.

A pseudo data-pair may be picked at any point along the ideal mode straight-line extrapolation (V_{ext}, i_d). For this illustration, a point was picked at the crossing of the two straight lines for convenience. The current at this location also approximates a reasonable value for the *IKF* parameter, which is beyond the scope of this paper. Note that *IKF* is generally of little use and necessitates a change in *RS*.

A corresponding data pair having the same current is then identified on the device IV profile (V_d, i_d) curve with dots. The voltage difference between the actual device, V_f, and the extrapolated resistive drop, V_{ext}, divided by the common current, i_d, represents the resistance, *RS* given earlier.

While the pseudo data-pair may be selected from any point on the gray line, the data pair is best selected where the current is highest.

Solving for *RS* in this manner is unnecessarily complicated but serves as an excellent graphic illustration. A step-by-step equation set is given in the **Appendix** (see **Table 4**) to solve for the parameters *IS*, *N*, and *RS*. The **Table 4** simplified equations were derived from the above-described process.

What's The Difference?

We have noted that while the SPICE diode at least pushes the simulation IV curves in the correct direction with respect to a defined ambient temperature, it is not able to provide a high level of fidelity for a broad sweep of ambient temperatures. To demonstrate this, we will compare the Linear Technology Corporation's 1N914A released SPICE characterization with its published curves:

Qn YQT @d Hi •] RNUR– @r •] PNUVX@ n] QNWURI

Referring to **Figure 3**, observe the heavy black lines showing the published outer limit ambient temperature values –40 °C +65 °C. These are relative to the nominal 25 °C, which has been shadowed out for simplicity. **Table 1** lists a comparison of published with simulation results for the three ambient temperatures, each at two or three forward voltages. The simulation

Table 1 – Simulation results using the LTSpice released characterization for the 1N914 diode. These six data-pair values (V_{fwd}, i_{fwd}) are shown graphically in Figure 3.

V_{fwd} (V)	T_{amb} (°C)	I_{fwd} (mA) Simulation	I_{fwd} (mA) Published
0.35	65	0.046	0.018
0.55	–40	0.0087	0.017
0.55	25	0.426	0.65
0.55	65	2.3	0.65
0.65	–40	150	150
0.65	25	3.76	1.9
0.65	65	14.0	3.9
0.75	–40	2.5	2.3

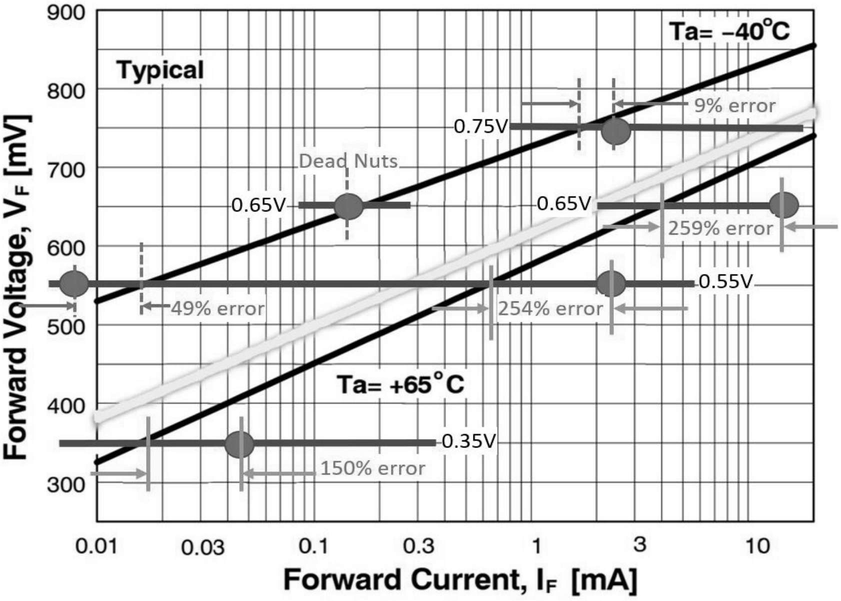


Figure 3 — Comparison of the published 1N914 IV performance curves (bold sloping lines) for named ambient temperatures together with selected simulation data-pair results from Table 1. The shadowed-out line de-emphasizes 25 °C for simplicity. The dots mark the data-pairs for returns of the SPICE simulation for the two ambient temperatures at forward voltages of 0.35, 0.55, 0.65, and 0.75 V.

results from **Table 1** are illustrated in **Figure 3** as heavy dots.

Observe in **Figure 3** that the simulation results (heavy dots) for –40 °C show a crossover point. With a forward voltage of 0.65 V the simulation returns 150 mA, which is the precision we can be satisfied with. But for the same temperature at any other forward voltage, the simulation returns results either too low or too high with errors less than 50%.

Moving to the simulation results at 65 °C we see that for the range plotted, all the returned currents are too high with errors greater than 150%. Though not illustrated, errors at 25 °C for the range plotted are from 35 to 98%.

This "even distribution" of error makes it clear that Linear Technology's characterization was intended to accommodate a logical best approximation across a wide-ranging set of ambient temperatures. It could be said that "nobody is right, but everybody is at least pretty close."

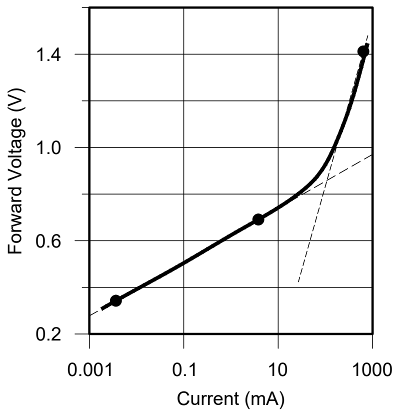
What is critical to note, however, is that the SPICE diode model has the capability to reproduce a published IV profile very accurately for a reference ambient temperature, whatever that may be. The premise of this paper centers around that critical observation.

Table 2 - SPICE diode IS and N parameters for characterization at 25 °C (298.15 K) to replace those found within the LTSpice released model.

Temp (°C, K)	25, 298.15
N, emission coefficient	1.9807084
IS (nA) saturation current	5.5276707
RS (Ω) bulk resistance	0.6163538

Table 3 - The critical data from the published charts (see Figure 4) for solving for the characterization parameters shown in Table 2.

Data pair	V_f (V)	i_f (mA)
1	0.381682	0.01000
2	0.616516	1.00971
3	1.420452	757.7306



QX2303-Cardone04

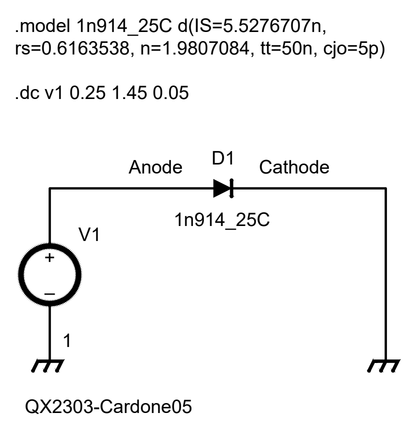
Figure 4 — The illustration depicts the foundation of characterization though data selection. The two straight lines depict the two relevant regions of operation: ideal (a function of the ideal diode equation) and resistive. The lowest current data pair must be selected low enough in voltage to not have any significant resistive influence.

Relevant Data Collection

Please refer to **Figure 4** illustrating that there are two processes that are operational simultaneously. One process is dominant under low current conditions and the other process is dominant under high current conditions. In addition, there is sharing in a transition region where the two meet. The raw data from **Figure 4** is assembled in **Table 3** where the three data-pairs (V_f, i_f) upper, mid and lower dots, have

Table 4 - Equations used to solve for the varied components of the overall characterization process. Temperature is in Kelvins.

Parameter	*value*
Thermal voltage	$V_t = kT/q$
k, J/K	$1.3806\mathrm{E}-23$
q, C	$1.6022\mathrm{E}-19$
Emission coefficient	$N = \dfrac{V_2 - V_1}{V_t ln\left(\frac{i_2}{i_1}\right)}$
Saturation current	$IS = \dfrac{i_{1\ or\ 2}}{e^{\frac{V_{1\ or\ 2}}{N\ V_t}} - 1}$
Bulk resistance	$RS = \dfrac{V_3 - V_t N\ ln\left(\frac{i_3}{IS}\right)}{i_3}$



```
.model 1n914_25C d(IS=5.5276707n,
rs=0.6163538, n=1.9807084, tt=50n, cjo=5p)

.dc v1 0.25 1.45 0.05
```

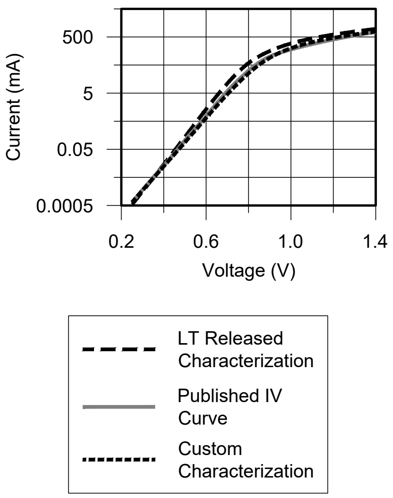
QX2303-Cardone05

Figure 5 — LTSpice simulation circuit used to sweep the 1N914 diode from 0.25 V to 1.45 V in increments of 0.05 V. TT and CJO were given values even though they do not contribute to a dc solution. This is simply a good practice contributing to preventing convergence issues.

been captured. The data pairs were then plugged into the solution protocol given in **Table 4** of the **Appendix**) to arrive at the solution set given in **Table 2**. This characterization will return very accurate results for transient analysis simulations where ambient temperatures are near 25 °C.

Deployment Characterization Example for the 1N914 Diode

Shown in **Figure 5** is a screen-capture from an LTSpice simulation to document the custom characterization (see **Table 2**) dc performance at the ambient temperature for which it was characterized: +25 ℃.



QX2303-Cardone06

Figure 6 — Comparison of published performance with the simulation results (IS=5.5277 nA, N=1.9807, and RS=0.6163 Ω) for an ambient temperature of 25 °C.

Note that two temporal parameters were included which do not affect dc performance — TT (transit time) and CJ0 (zero bias junction capacitance). If not specified, their values default to zero creating a remote possibility for a loss of numerical convergence.

Under some conditions where a design has switching functionality using a device without temporal understanding, SPICE may fail to converge due to what it detects as a discontinuity. That is, a signal that is supposed to be both one value and another value at the same time.

As humans, we look at a square wave and think "step-function." But we all know that buried deep within any oscilloscope square wave trace that there is a rise time. It is a good practice to include temporal parasitic model effects even when their presence is not going to be significant. The reason is that SPICE cannot understand a signal transition that takes place in zero time. When it encounters a discontinuity, it may possibly fail to converge aborting the simulation abnormally leaving you in the dark as to why.

Comparison of Custom With Released Characterization

In **Figure 6** there is a comparison of the LTSpice 1N914 diode with the custom characterization accomplished in this paper for an ambient temperature of 25 ℃.

It is important to note that this characterization was algorithmic, leaving very little to engineering judgment. The steps were to first identify three data pairs on the published curve for the ambient temperature of interest. We then plugged those data pairs into the equation solution mechanism found in the **Appendix**. Three parameters were solved for: *N*, *IS*, and *RS*. With just these three parameters the characterization nearly superimposed itself over the published curve for the named ambient temperature with no iteration required for parameter values. Some might call this plug-and-chug.

But My Customization Didn't Superimpose

For the case where simulation results for a custom diode characterization produce excess error, consider choosing different data pairs from the published IV profile. The most common error is selecting data pair number 2 (reference **Table 3**) at too high of a current level where resistive effects are beginning to become significant.

Conclusion

In most cases, the LTSpice distributed-error released model will be more than sufficient in supplying reasonably accurate results for any ambient temperature. However, it may also be that the forward characteristics with respect to the ambient temperature are critical such as in applications where the natural log characteristic of the silicon diode is used to accurately detect and quantify temperature changes within an electronic or other device.

It may also be that the diode called for in a design is not found in the LTSpice released library. A custom diode SPICE model characterization would be required. An example of a diode not found in the LTSpice library at the time of this writing is the 1N4729 Zener diode. Pick a temporally close proximity Zener characterization from the LTSpice library and substitute the *IS*, *N*, and *RS* parameters solved for here.

What we have shown in this paper is that the SPICE model diode is capable of accurate characterization for a published IV profile (for a named ambient temperature) for virtually all silicon diodes.

Appendix

The equations for the SPICE diode parameter characterizations are shown in **Table 4**. This is a step-by-step process that must be accomplished in order. Attempting to skip a step will immediately reveal the necessity of following an order since each step develops a variable that is used in the next step.

Refer to **Figure 4** as an aid in following the data-pair descriptions below. Data pair number 1 is taken at the lowest published current available. Data pair number 2 can be tricky to evaluate and correctly select. This data pair must be taken from the ideal region of operation with care taken to be certain that resistive effects are not significant. Data pair number 3 is readily identified as simply the data-pair with the highest current.

Wesley Cardone , N8QM, graduated from Cal Poly, San Luis Obispo with a BSEE. He has worked for Boeing at both Vandenburg AFB and Seattle, and the Johns Hopkins University Applied Physics Laboratory in Maryland. He has also worked in the automotive industry for Ford Motor, Lear Corporation, Fiat-Chrysler, and Caresoft Global of Livonia. Wes has pursued amateur radio since 2018 and currently serves as trustee for the Chelsea Amateur Radio Club (CARC) repeater. Wes also teaches a weekly class by video conference in amateur radio electromagnetics, sponsored by CARC. The class is designed for the licensed amateur radio public not highly schooled in electronics but wanting to upgrade to the amateur radio General and Amateur Extra classes. The class is named, "I Hate Cookbooks Guide to Amateur Radio Electromagnetics."