Precision Generic Diode Characterization for Simulation

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

ABSTRACT:

The characterization of a SPICE (Simulation Program with Integrated Circuit Emphasis) 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. The error is distributed over the published range of ambient temperatures to minimize the overall error. This paper will explain the nature of the SPICE diode model and how to solve for its DC characterization parameters given a reference ambient temperature profile. A simulation diode model so characterized will virtually superimpose itself over the published IV profile for that 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 restricted to mainframe use.

With the release of the Intel 80486 processor in 1980, with its 32-bit architecture, personal computers became powerful enough to accommodate SPICE. 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 effectively making them non-available for hobbyists and companies on limited budgets.

In 1999, Linear Technology released its first version of LTSpice[1] 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.

[1] LTSpice 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."

Precision Generic Diode Characterization for Simulation

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

THE SPICE DIODE MODEL

The diode model found in LTSpice, though it has many enhancements for convenience, still includes the original functionality of the SPICE 2g.6 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.

Figure 1 depicts the fundamental DC functionality for the physical silicon diode's voltage-current (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 (Y-axis) there is a dependent current (log X-axis). There are two regions of interest for simulation and design purposes described below.

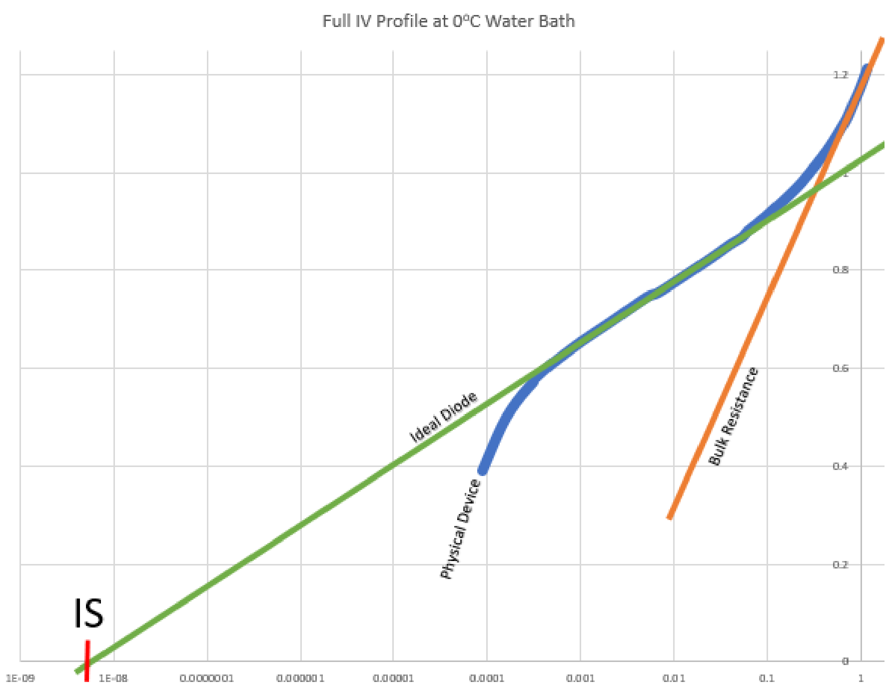

Figure 1 Illustration of the physical nature of the silicon diode with reference to its regions of operation: recombination; ideal; and high injection. The data of this figure was taken with the device under test immersed in a water bath at a controlled 25°C temperature.

In Figure 1, the heavy blue curve illustrates the physical IV performance for the physical 1N914 silicon diode through all of its regions. 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 Volts is not accounted for by the SPICE diode 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 following equation to solve for a forward voltage given a current and ambient temperature:

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

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

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

function of the environment and is not accessible by the user. There is a minimal conductance, GMIN, that governs the diode at very low forward voltages, V_f. Here V_f.GMIN will begin to dominate the above equation.

Continuing with reference to Figure 1, the most significant region is the ideal. Here, the diode's PN junction follows an ideal semi-log path as a function of the forward voltage. The heavy green straight line covers the device ideal mode operation but for illustration purposes extrapolates into the adjacent regions. Observe the point where the ideal curve extrapolation crosses zero on the Y-axis (forward voltage). The corresponding ideal current extrapolation to zero volts represents the diode's saturation current and is marked in Figure 1 by the red 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.

The green line in the illustration is a visualization of the ideal region as a straight line on a semi-log plot of the form $y = mx + b$. Written in this form, b represents the saturation current, IS. Given the Figure 1 XY semi-log plot, it is possible to solve for any value of current given a voltage.

Crucial to understanding 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 constant 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 use. 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. The SPICE diode model accounts for this region with a linear bulk resistance as illustrated graphically in Figure 2. This bulk resistance serves as supplemental voltage drop to the ideal drop. At relatively low currents, the bulk resistance drop is insignificant 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 voltage drops continue to increase with increasing current though the resistive drop will eclipse the ideal drop with further increases in current.

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

The key to understanding how the SPICE diode model's RS parameter is developed is graphically shown in Figure 2. A pseudo data-pair[2] may be picked at any point along the ideal mode straight-line extrapolation (V_{ext}, i_d). A corresponding data-pair having the same current is then identified on the device IV profile (V_d, i_d). The voltage difference between the actual device, V_f, and the extrapolated ideal mode drop, V_{ext}, divided by the common current, i_d, represents the resistance, RS. The pseudo data-pair is best selected where the current is highest and where the RS drop is most dominant.

Solving for RS in this manner is unnecessarily complicated. 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.

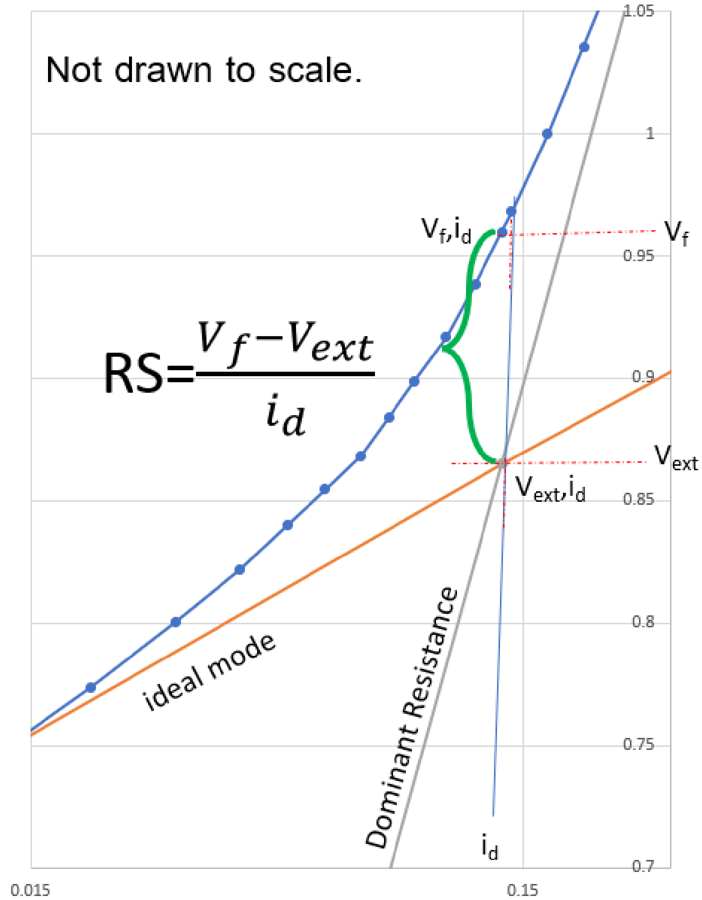

Figure 2 Graphic illustration on obtaining the bulk resistance from the published IV profile.

WHAT'S THE DIFF?

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

```
1N914 D(Is=2.52n Rs=0.568 N=1.752)
```

Referring to Figure 3, observe the heavy black lines showing the published outer limit ambient temperature values -40°C and +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 results from Table 1 are illustrated in Figure 3 as blue dots.

[2] 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.

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

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

Table 1 Shown here are simulation results using the OnSemi released characterization for the 1N914 diode. These simulations returned forward currents, i_{fwd}, given a specified forward voltage, V_{fwd}, and ambient temperature, T_{amb}. These six data-pair values (V_{fwd}, i_{fwd}) are shown graphically in Figure 3.

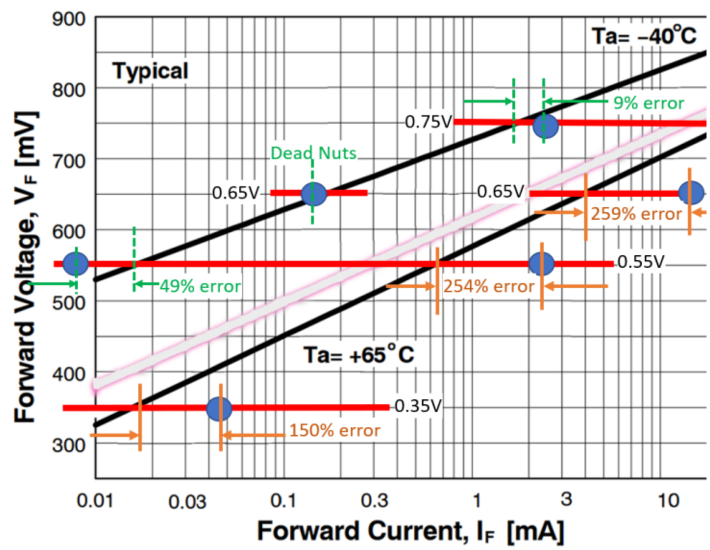

Figure 3 Shown is a comparison of the published 1N914 IV performance curves together with selected simulation data-pair results given in Table 1. The heavy black lines mark the published IV 1N914 performance for three ambient temperatures. The shadowed-out line removes 25°C for simplicity. The blue dots mark the data-pairs for returns of the SPICE characterization for the two ambient temperatures at 0.35, 0.55, 0.65, and 0.75 Volts.

Observe in Figure 3 that the simulation results (blue dots) for -40°C show a crossover point. With a forward voltage of 0.65 Volts the simulation returns 150mA 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 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.

Precision Generic Diode Characterization for Simulation

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

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

Table 2 Shown here are the SPICE diode IS and N parameters for an accurate characterization for 25°C. These were solved for using the equations and protocol outlined in the appendix together with the selected raw data shown in Table 3.

25	**Temp (°C)**	**Environment**
298.15	**Temp (°K)**	
Data-Pair number	V_f (V)	i_f (mA)
1	0.381682	0.01000
2	0.616516	1.00971
3	1.420452	757.7306

Table 3 The critical data from the published charts is assembled here (refence Figure 4) for solving for the characterization parameters shown in Table 2.

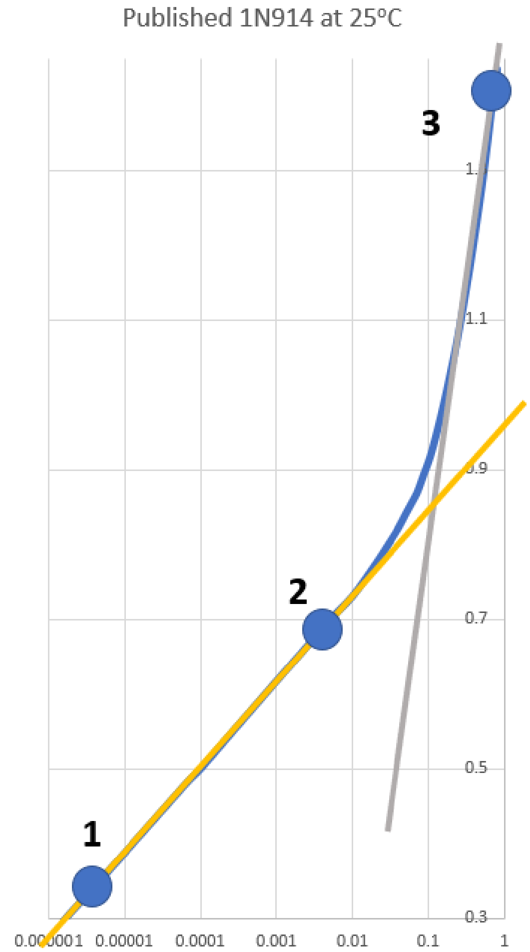

Figure 4 The illustration depicts the foundation of characterization though data selection. The two straight lines depict the two relevant regions of operation: ideal (a function of the ideal diode equation) and resistive. Data-pair 1must not have resistive influence.

RELEVANT DATA COLLECTION:

Please refer to Figure 4 illustrating that there are two processes are operational simultaneously. One process is dominant under low current conditions and the other process is dominant under high current conditions. However, there is sharing in a transition region. The raw data is assembled in Table 3 where three data-pairs (V_f, i_f) have been captured. The data-pairs were then plugged into the solution protocol given in the appendix (Table 4) to arrive at a solution set given in Table 2. This characterization will return very accurate results for simulations using ambient temperatures near 25°C.

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

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°C.

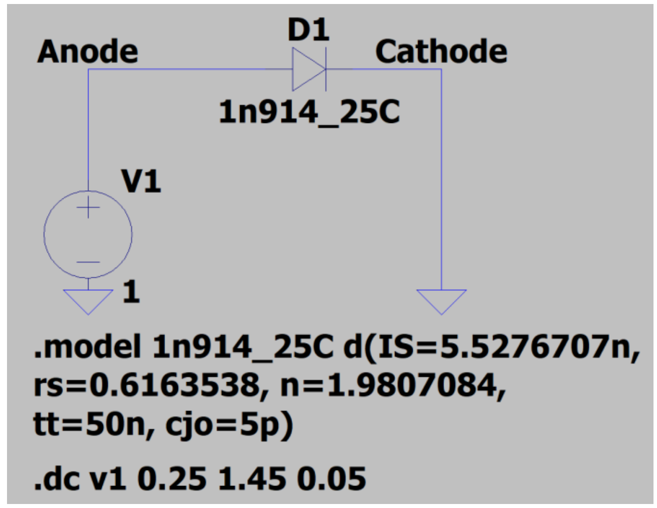

Figure 5 Shown is a screen capture of the LTSpice simulation circuit used to sweep the 1N914 diode from 0.25 V to 1.45 V in increments of 50mV. Though not discussed earlier, TT and CJO were given values even though they do not contribute to a DC solution. This is simply a good practice contributing to preventing convergence issues.

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 using a device without a temporal understanding, SPICE may fail to converge due to what it detects as a discontinuity. That is, a signal is supposed to be both one value and another 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 O-scope 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°C.

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 simply 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 with no iteration of parameter values.

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

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.

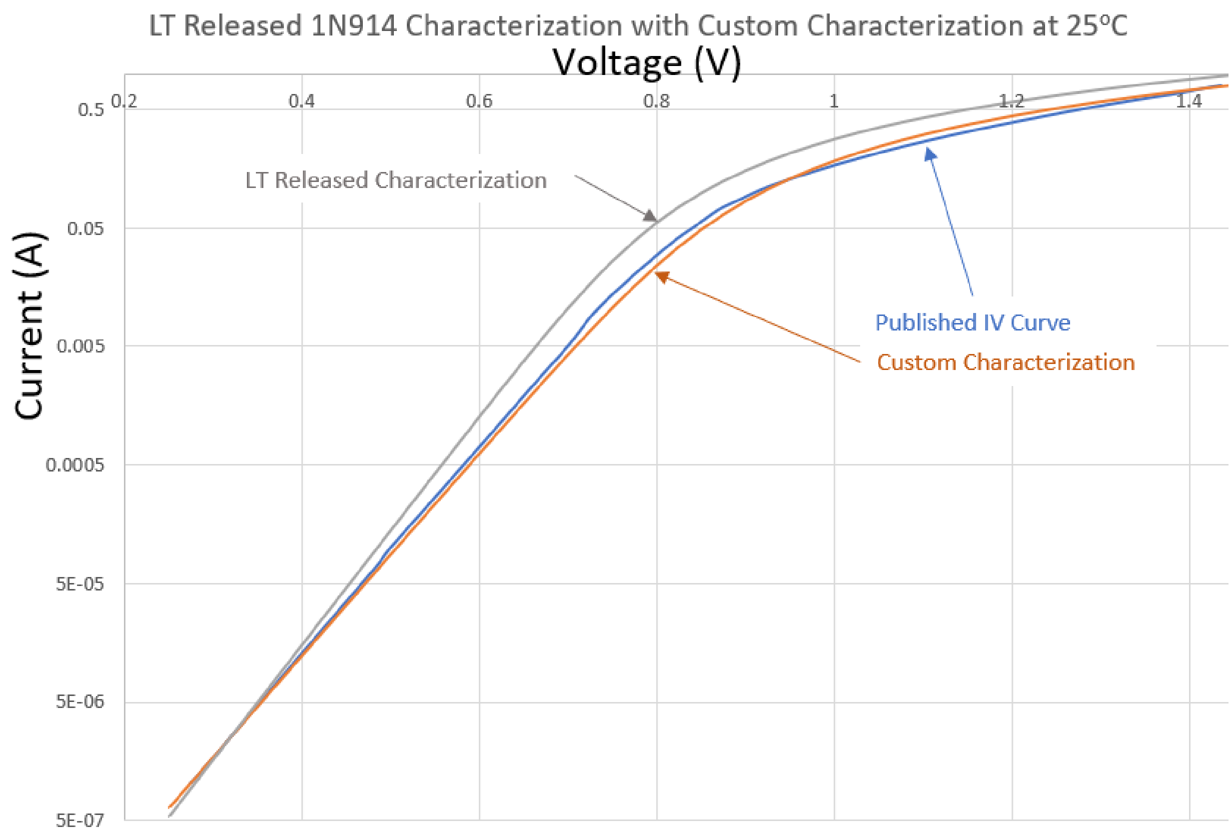

Figure 6 Shown above is a comparison of published performance with the simulation results for an ambient temperature of 25°C. The blue line illustrates the published performance of the physical 1N914 diode at 25°C. The tan line marks the simulation results of this paper's custom characterization of the 1N914 diode for that same ambient temperature. The grey line marks the performance of the LT released 1N914 diode at 25°C.

CONCLUSION:

In most cases, the LTSpice released model will be more than sufficient in supplying an accurately performing diode model within your design for any ambient temperature given the requirements of one's design. 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.

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

It may also be that the diode called for in a design is not found in the LTSpice. A custom diode SPICE model characterization would be required[3].

What we have shown in this paper is that the SPICE model diode is fully capable of accurate characterization for a published IV profile (for a named ambient temperature) for virtually all silicon diodes given three parameters solved for: 1.emission coefficient (N); 2. Saturation current (IS); and 3. Bulk resistance (RS).

[3] 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.

Wesley Cardone, N8QM
n8qm@arrl.net – January 12, 2022
10626 Coopersfield Rd.
Chelsea, Michigan 48118

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.

Thermal voltage	$V_t = {kT}/{q}$
k	$= 1.3806e - 23 \, {J}/{^\circ K}$
q	$q = 1.6022e - 19 \, C$
Emission Coefficient	$N = \frac{V_2 - V_1}{V_t ln\left({i_2}/{i_1}\right)}$
Saturation Current	$IS = \frac{i_{1\,or\,2}}{e^{{V_{1\,or\,2}}/{N\,V_t}} - 1}$
Bulk Resistance	$RS = \frac{V_3 - V_t N\, ln\left({i_3}/{IS}\right)}{i_3}$

Table 4 Shown above are the equations, and values where applicable, used to solve for the varied components of the overall characterization process. Carefully note that the temperature T used in the thermal voltage must be in degrees Kelvin.