11.1.3 Parameter Measurement

Very sophisticated fitting algorithms and software exist that can be used to fit the nonlinear SPICE equations to experimental data. These algorithms, however, require a set of initial values for the fitted parameters to be specified. In some cases, the nonlinear fitting depends very critically on the initial parameter values. Simple graphic methods can be used to determine the values of SPICE parameters. The parameter values obtained in this way can be used as initial values for nonlinear fitting. Importantly, the graphic methods provide a visual demonstration of how good fit can be achieved between selected SPICE equations and specific experimental data.

Measurement of I_S, n, and r_S

The three device parameters involved in the current–voltage equation for $V_{D0} > -BV$ are I_S, n, and r_S. Proper values of these SPICE parameters need to be set to ensure correct simulation. The default values of these parameters (typically, $I_S = 10^{-14}$ A, $n = 1, r_S = 0$) cannot guarantee an acceptable agreement between the model and the real characteristic of any possible type of diode. Figures 6.11 and 6.12 demonstrate the importance of properly setting the values of n and r_S, respectively.

Figure 11.1 illustrates a graphic method for determination of I_S, n, and r_S parameters from experimental I_D–V_D data. Given that the current depends exponentially on the voltage, a $\ln I_D$–V_{D0} graph is used to linearize the problem. The open symbols show the raw experimental data—that is, when the measured V_D voltage is used as the P–N junction voltage V_{D0}. Because the voltage across r_S (which is $r_S I_D$) is neglected in this case, the voltage V_{D0} is effectively overestimated by $r_S I_D$ (refer to Fig. 6.12). This effect is not pronounced at small currents as $r_S I_D \ll V_{D0}$ (the linear part of the graph); however, it becomes observable at high currents. A good initial guess for r_S can be obtained by judging the maximum

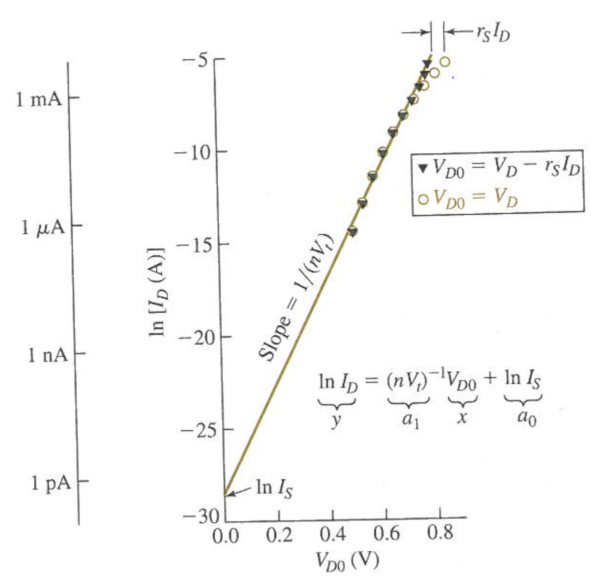

Figure 11.1 Measurement of diode static SPICE parameters, I_S, n, and r_S. The experimental data (symbols) and the fitting (line) are also shown in Fig 6.12.

deviation $r_S I_D$ of the raw experimental data from the straight line extrapolated from the low-current linear portion of the I_D–V_D dependence. The maximum deviation has to be, obviously, divided by the maximum current I_D to obtain r_S. Using the estimated value of r_S, the experimental diode voltage points V_D are transformed into P–N junction voltage points as $V_{D0} = V_D - r_S I_D$. If a straight line is obtained, the value of r_S is taken as the final value. Alternatively, r_S is altered and the process repeated until a straight line is obtained.

The closed symbols in Fig. 11.1 show the straight line obtained after the voltage effect of the parasitic resistance, $r_S I_D$, is extracted from the raw experimental data. In other words, the closed symbols represent the experimental characteristic of the current source in the SPICE diode model (Table 11.1). Because the experimental data are collected in the forward-bias region, where $\exp(V_{D0}/nV_t) \gg 1$, the SPICE $I_D(V_{D0})$ equation is reduced to

$$I_D = I_S e^{V_{D0}/nV_t} \tag{11.2}$$

Therefore, the logarithm of the current $\ln I_D$ linearly depends on V_{D0}:

$$\ln I_D = \frac{1}{nV_t} V_{D0} + \ln I_S \tag{11.3}$$

Figure 11.1 illustrates that the parameters I_S and n are obtained from the coefficients a_0 and a_1 of the linear $\ln I_D$–V_{D0} dependence as

$$\begin{aligned} I_S &= e^{a_0} \\ n &= 1/a_1 V_t \end{aligned} \tag{11.4}$$

EXAMPLE 11.1 Measurement of Static SPICE Parameters

A set of measured I_D–V_D values for a P–N junction diode are given in Table 11.3. Obtain SPICE parameters I_S, n, and r_S for this diode.

SOLUTION

Let us assume that the parasitic resistance r_S is on the order of 10 Ω. In that case, the voltage across the parasitic resistance is $\le 1 \times 10^{-3} \times 10 = 0.01$ V for currents ≤ 1 mA. This means that the parasitic resistance effect can be neglected (0.01 V is much smaller than ≈ 0.7 V appearing across the P–N junction) for currents ≤ 1 mA. The measured diode current I_D can then be directly related to the measured voltage V_D as $I_D = I_S \exp(V_D/nV_t)$. This exponential equation can be

TABLE 11.3 Current–Voltage Measurements

V_D (V)	0.67	0.70	0.73	0.76	0.80	0.84	0.91	1.00	1.26	1.65
I_D (mA)	0.1	0.2	0.5	1.0	2.0	5.0	10.0	20.0	50.0	100.0

TABLE 11.4 Linearization of Current–Voltage Data

I_D (mA)	$y = \ln I_D$ ln(mA)	$x = V_D$ (V)
0.1	−2.303	0.67
0.2	−1.609	0.70
0.5	−0.693	0.73
1.0	0.000	0.76

linearized in the following way:

$$\ln I_D = \ln I_S + \frac{1}{nV_t} V_D$$

that is

$$y = a_0 + a_1 x$$

where $y = \ln I_D$, $x = V_D$, $a_0 = \ln I_S$, and $a_1 = 1/nV_t$. The results of this linearization, applied to the first four experimental points ($I_D \leq 1$ mA) from Table 11.3, are given in Table 11.4.

The graphic method, explained in the previous section, can be used to find the coefficients a_0 and a_1 of this linear relationship. Alternatively, these coefficients can be calculated using the numerical linear regression method. For the case of a one-variable linear equation (and two parameters, a_0 and a_1), the following system of two linear equations has to be solved:

$$\begin{aligned} na_0 \quad &+ \left(\textstyle\sum_{i=1}^{n} x_i\right) a_1 = \textstyle\sum_{i=1}^{n} y_i \\ \left(\textstyle\sum_{i=1}^{n} x_i\right) a_0 &+ \left(\textstyle\sum_{i=1}^{n} x_i^2\right) a_1 = \textstyle\sum_{i=1}^{n} x_i y_i \end{aligned} \qquad (11.5)$$

where n is the number of experimental points used for the linear fitting. Applying the system of equations (11.5) to the data of Table 11.4, one obtains

$$\begin{aligned} 4a_0 + 2.86a_1 &= -4.605 \\ 2.86a_0 + 2.0494a_1 &= -3.1752 \end{aligned}$$

The solution of the above system of equations is $a_0 = -19.80$, and $a_1 = 26.08$. The parameters I_S and n can now be calculated as $I_S = \exp(a_0) = 2.52 \times 10^{-9}$ mA $= 2.52 \times 10^{-12}$ A, $n = 1/a_1 V_t = 1.48$.

If the parasitic resistance r_S was zero, the voltage V_D at the highest current $I_D = 100$ mA would be $V_D = 1.48 \times 0.02585 \times \ln(100/2.52 \times 10^{-9}) = 0.93$ V. It can be seen from Table 11.3 that the measured voltage is 1.65 V. The difference $1.65 - 0.93 = 0.72$ V is due to the voltage across r_S: $r_S I_D = 0.72$ V. Using this difference, the parasitic resistance is estimated as $r_S = 0.72\ \text{V}/I_D = 0.72/100\ \text{mA} = 7.2\ \Omega$. If $r_S = 7.2\ \Omega$ is a proper value, the voltage across the P–N

TABLE 11.5 Transformed Current–Voltage Data

I_D (mA)	$V_{D0} = V_D - r_s I_D$ (V)	$V_{D0} = nV_t \ln(I_D/I_S)$ (V)
	$I_S = 2.5 \times 10^{-12}$ A, $n = 1.48$, $r_S = 7.2\ \Omega$	
0.1	0.67	0.67
0.2	0.70	0.70
0.5	0.73	0.73
1.0	0.76	0.76
2.0	0.79	0.78
5.0	0.80	0.82
10.0	0.84	0.84
20.0	0.86	0.87
50.0	0.90	0.91
100.0	0.93	0.93

junction V_{D0}, calculated as $V_D - r_S I_D$, should closely match the values calculated from the diode equation $nV_t \ln(I_D/I_S)$. The results of these calculations are presented in Table 11.5. It can be seen that the theoretical values (the third column) closely match the transformed experimental values (the second column). Therefore, we conclude that $I_S = 2.5 \times 10^{-12}$ A, $n = 1.48$, and $r_S = 7.2\ \Omega$ represent a good set of SPICE parameters for the considered diode. If the matching was not good, the value of r_S would be altered to try to improve the matching.

Measurement of $C_d(0)$, V_{bi}, and m

It is not possible to completely linearize the model for the reverse-biased P–N junction capacitance, given by Eq. (6.57). Therefore, the graphic or the linear regression methods cannot be directly applied. The situation is further complicated by the fact that the measured data contain an additional, parasitic capacitance component. The P–N junction capacitance can be measured in different ways, perhaps most suitable being by means of a bridge. The measurement frequency can be set low enough so that the parasitic series resistance becomes negligible compared to the impedance of the capacitor. However, the parasitic capacitance, caused mainly by pin capacitance, stray capacitance, and pad capacitance, cannot be avoided. Assuming that the parasitic capacitance C_p does not depend on the voltage applied, the measured capacitance can be expressed as

$$C_{meas} = C_d(0)\left(1 + \frac{V_R}{V_{bi}}\right)^{-m} + C_p \qquad (11.6)$$

Although the parameter C_p in Eq. (11.6) is not needed as a SPICE parameter, it has to be extracted from the experimental data.

Curve fitting can be the most effective way of parameter measurement in this case, provided the initial parameter values are properly determined. There are four parameters