

# Practical Implementation of Diode SPICE Model with Reverse Recovery

Denys Igorovych Zaikin <sup>a</sup>

Advent Technologies A/S, Lyngvej 8, Aalborg DK-9000, Denmark

Keywords: Diode Reverse Recovery, SPICE, Circuit Simulation.

Abstract: Peter O. Lauritzen and Cliff L. Ma proposed an approach for creating a physical model of reverse recovery for soft recovery diodes in 1991. The current paper demonstrates how to create the proper SPICE sub-circuit using only the specifications from the diode datasheet from the manufacturer. Software for characterization tools has been developed, tested, and is now openly accessible for use.

## 1 INTRODUCTION


Of the many SPICE-based simulators on the market, most still use the old standard diode SPICE model that does not cover reverse recovery correctly. Both LTspice (Analog Devices, Inc., 2008) and Pspice (Cadence Design Systems, Inc., 2015) are powerful pieces of software that are widely used for power electronics simulation. These SPICE simulators use a basic diode model. Adding a feature to simulate diode reverse recovery will improve loss estimation and circuit behaviour simulation. This is especially attractive for LTspice, which is a powerful, free simulator that can be used in complex design simulation. This paper applies original theoretical work (Lauritzen and Ma, 1991) to the practical implementation of a SPICE macro model of diodes with reverse recovery. The model described in (Lauritzen and Ma, 1991) is based on real physical processes in a diode and, because of this, is robust.

A Windows OS application was created to generate a diode macro model using only parameters from the diode manufacturer's datasheet or measurement data.

## 2 DIODE MODEL DESCRIPTION

Original work (Lauritzen and Ma, 1991) provides the following three equations for diodes with reverse recovery:

$$i(t) = \frac{(q_E - q_M)}{T_M}, \quad (1)$$

<sup>a</sup>  <https://orcid.org/0000-0003-4080-5631>

$$0 = \frac{dq_M}{dt} + \frac{q_M}{\tau} - \frac{(q_E - q_M)}{T_M}, \quad (2)$$

$$q_E = I_s \tau (e^{(\frac{v}{nV_T})} - 1). \quad (3)$$

From equations (1)-(3), the forward DC-bias characteristic can be obtained:

$$i = \frac{I_s}{(1 + T_M/\tau)} (e^{(\frac{v}{nV_T})} - 1). \quad (4)$$

Here,  $i$  is the diode current,  $v$  is diode junction voltage,  $V_T = kT/q$  is the thermal voltage,  $I_s$  is saturation current (similar to the SPICE basic diode model parameter) and  $n$  is the emission coefficient (similar to the SPICE basic diode model parameter). The variables from (Lauritzen and Ma, 1991) are as follows:  $T_M$  represents diffusion time,  $\tau$  recombination lifetime,  $q_M$  total stored charge and  $q_E$  charge variable. This model is completed with ohmic resistance  $R_s$  and junction capacitance  $C_j$ , as shown in Figure 1.

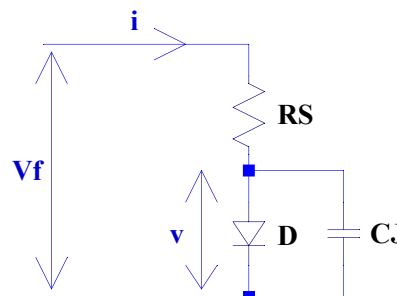


Figure 1: Diode model components.

The practical implementation of equations (1)-(4) in the SPICE model, along with ohmic resistance and junction capacitance, are shown in Figure 2.

```
.param Temp_diode=1.250000e+002 k=1.381000e-023 q=1.602000e-019 Vt=3.430949e-002
.param Rs=2.196899e-002 ls=7.942178e-004 N=2.357317e+000 Tau=9.273759e-008 Tm=1.527752e-007
.param VJ=2.000000e+000 Cj0=9.311509e-010 M=6.974055e-001 FC=0.500000
.model Cap d (ls=1e-14 N=200 rs=10u Xti=0 Eg=0 Cj0=(Cj0) M=(M) VJ=(VJ) FC=(FC))
```

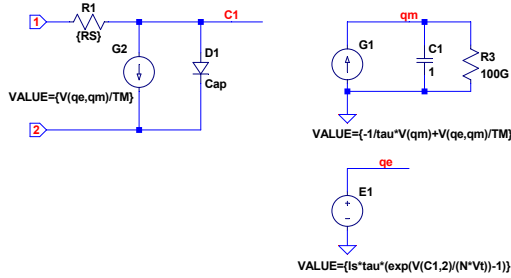
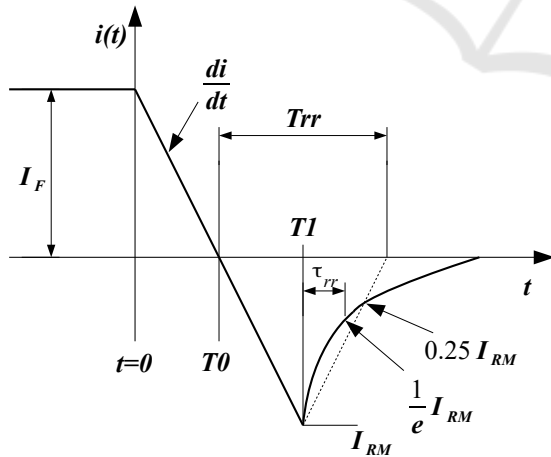


Figure 2: Implementation of new diode model in SPICE.

### 3 EXTRACTION OF MODEL PARAMETERS

Equations (1)-(3) fully describe diode reverse recovery and DC bias characteristics of the diode. To use these equations, it is necessary to define  $\tau$ ,  $T_M$ ,  $n$  and  $I_s$  parameters.

Parameters  $\tau$  and  $T_M$  are defined with the approach used in (Lauritzen and Ma, 1991). An additional intermediate parameter – reverse recovery time constant  $\tau_{rr}$  – is used and can be measured directly from the reverse recovery waveform or defined from the diode datasheet's parameter  $T_{rr}$ , the reverse recovery time. Figure 3 shows the JEDEC Standard (JEDEC Standard No. 282B.01, 2000) definition of  $T_{rr}$ .


 Figure 3: JEDEC reverse recovery time  $T_{rr}$  definition and waveform.

From Figure 3,  $\tau_{rr}$  can be found as follows:

$$\tau_{rr} = 0.75 \left( \frac{I_{RM}}{\left(\frac{di}{dt}\right)} - T_{rr} \right) \frac{1}{\ln(0.25)}. \quad (5)$$

Now, when  $\tau_{rr}$  is known, parameters  $\tau$  and  $T_M$  can

be found using numerical equation solving of the following equations from reference (Lauritzen and Ma, 1991):

$$1/\tau_{rr} = 1/\tau + 1/T_M, \quad (6)$$

$$I_{RM} = \frac{di}{dt} (\tau - \tau_{rr}) (1 - e^{-\frac{T1}{\tau}}). \quad (7)$$

To find parameters  $I_s$ ,  $n$  and ohmic resistance  $R_s$  in equation (4), the standard diode forward DC-bias SPICE model equations are used (Figure 1):

$$V_f = R_s \cdot i + v, \quad (8)$$

$$i = I_s' \cdot \left( e^{\frac{v}{nV_T}} - 1 \right). \quad (9)$$

Based on Equation (9) and Figure 4, it can be seen that  $I_s'$  is the leakage reverse current at the maximum reverse voltage according to the datasheet's reverse DC-bias characteristic of the diode.

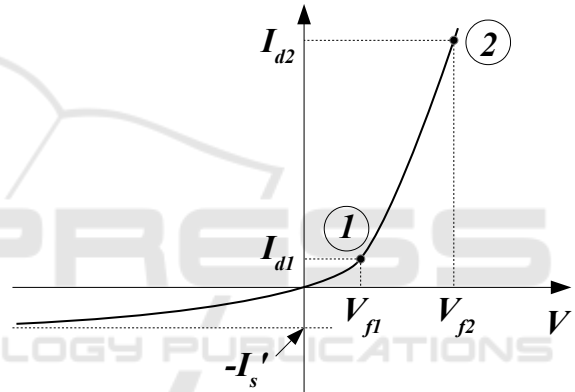


Figure 4: DC-bias characteristic points.

Using Equations (4) and (9),  $I_s$  can be found:

$$I_s = I_s' \cdot (1 + T_M/\tau). \quad (10)$$

To find  $R_s$  and  $n$ , two points should be defined on the DC-bias diode characteristic (Figure 4). From equations (8) and (9), there is system of equations with two unknown variables,  $R_s$  and  $n$ :

$$\begin{cases} V_{f1} = n \cdot V_T \cdot \ln(I_{d1}/I_s' + 1) + R_s \cdot I_{d1} \\ V_{f2} = n \cdot V_T \cdot \ln(I_{d2}/I_s' + 1) + R_s \cdot I_{d2}. \end{cases} \quad (11)$$

After system (11) is solved,  $R_s$  and  $n$  are found:

$$n = \frac{V_{f1} \cdot I_{d2} - V_{f2} \cdot I_{d1}}{\ln \left( \frac{\left(\frac{I_{d1}}{I_s'} + 1\right) \cdot I_{d2}}{\left(\frac{I_{d2}}{I_s'} + 1\right) \cdot I_{d1}} \right)} \cdot \left( \frac{1}{V_T} \right), \quad (12)$$

$$R_s = \frac{V_{f2} - V_T \cdot n \cdot \ln(I_{d2}/I_s' + 1)}{I_{d2}}. \quad (13)$$

To simulate non-linear junction capacitance, equations from the standard diode SPICE model are used (Kielkowski, Ron M, 1995):

$$C_J = C_{J0} \cdot \left(1 - \frac{v}{V_J}\right)^{-M}, \quad v < F_C \cdot V_J, \quad (14)$$

$$C_J = \frac{C_{J0}}{(1 - F_C)^{M+1}} \times \left(1 - F_C \cdot (M+1) + \frac{M \cdot v}{V_J}\right), \quad v \geq F_C \cdot V_J. \quad (15)$$

In this model for junction capacitance, fixed parameters are assumed:  $V_J = 2.0$  and  $F_C = 0.5$ . It is also necessary to find parameters  $M$  and  $C_{J0}$  using two points on the datasheet's reverse bias capacitance curve in Figure 5.

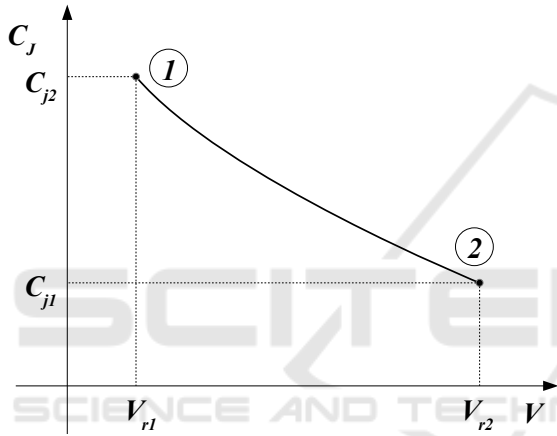


Figure 5: Junction capacitance points.

Using these two points, a system of equations can be obtained based on Equation (14):

$$\begin{cases} C_{j1} = C_{J0} \cdot \left(1 + \frac{V_{r1}}{V_J}\right)^{-M} \\ C_{j2} = C_{J0} \cdot \left(1 + \frac{V_{r2}}{V_J}\right)^{-M} \end{cases} \quad (16)$$

After (16) is solved,  $M$  and  $C_{J0}$  can be found:

$$M = \frac{\ln\left(\frac{C_{j1}}{C_{j2}}\right)}{\ln\left(\frac{1+V_{r2}/V_J}{1+V_{r1}/V_J}\right)}, \quad (17)$$

$$C_{J0} = C_{j1} \cdot \left(1 + \frac{V_{r1}}{V_J}\right)^M. \quad (18)$$

To implement junction capacitance in the new diode model, the standard SPICE diode model is placed in parallel with the diode body (Figure 2).

The new diode SPICE model implements the behavior of the diode at a fixed temperature. Lead inductances should be added externally for parasitic simulation.

## 4 SIMULATION RESULTS

The newly generated model was tested in two simulators – LTspice and Pspice. The LTspice IV simulation results for the MUR460 diode are shown in Figure 6.

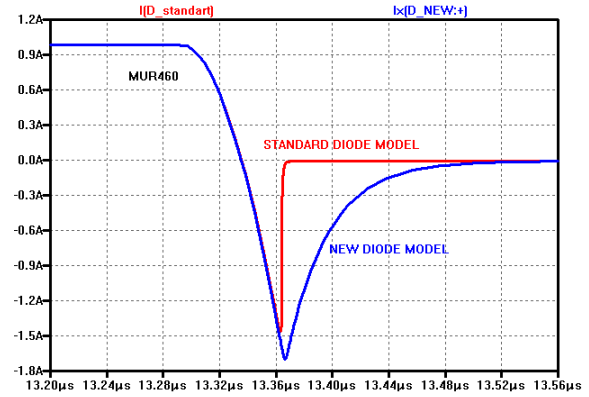


Figure 6: Comparison of current and standard waveform LTspice IV simulation results for the MUR460 diode. The MUR460 standard diode model was taken from the LTspice IV library.

Pspice 16.6 simulation results for the HFA25TB60 diode are shown in Figure 7.

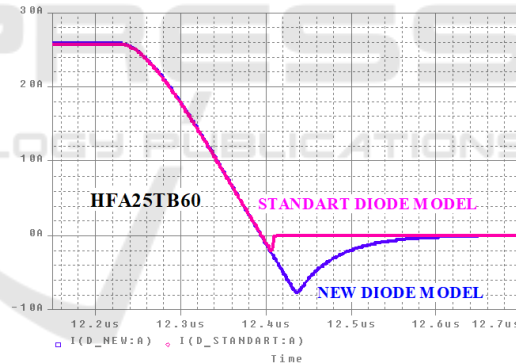


Figure 7: Comparison of current and standard waveform Pspice 16.6 simulation results for the HFA25TB60 diode. The HFA25TB60 standard diode model was taken from the Pspice 16.6 library.

The Pspice 16.6 simulation results for ISL9R3060 diode are shown in Figure 8.

## 5 SOFTWARE DESCRIPTION

To generate the diode SPICE model using the manufacturer's datasheet characterization, the software (SW) tool "DiodeRRSubmodel" was made (Figure 9). It is a Windows OS application and can be freely downloaded from the link (ZAIKIN D.I., 2021).

Next, input data from the datasheet are used as fol-

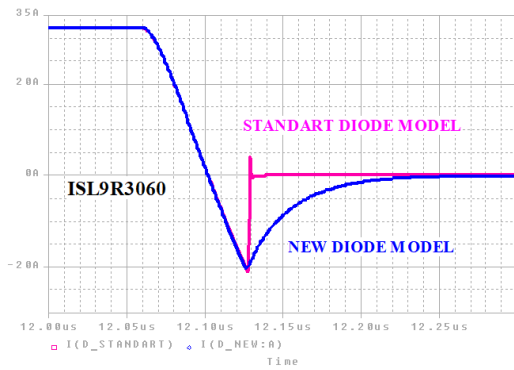


Figure 8: Comparison of current and standard waveform Pspice 16.6 simulation results for the ISL9R3060 diode. The ISL9R3060 standard diode model was taken from the manufacturer webpage.

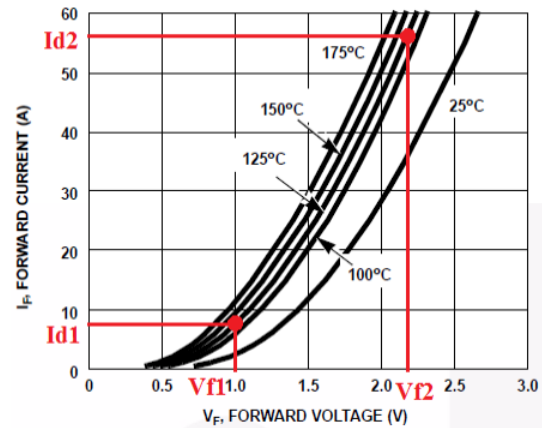


Figure 10: Two points selected on the DC-bias characteristic.

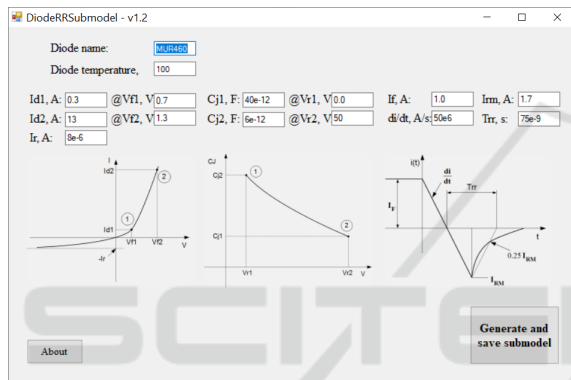


Figure 9: Windows OS application SW for diode model extraction.

lows:

- user enters diode name;
- user defines work temperature of diode;
- user enters two points on the DC-bias forward characteristic for the specified temperature (Figure 4, Figure 10). Points far enough away from each other should be chosen;
- user enters reverse leakage current for diode at specified temperature;
- user enters two points on the junction capacitance (Figure 5, Figure 11). Points far enough away from each other should be chosen;
- user enters the reverse recovery specification from the diode datasheet at the specified temperature:  $I_f$ ,  $di/dt$ ,  $I_{RM}$ , and  $T_{rr}$  (Figure 3, Figure 12).

The extracted model file is placed in the same folder as the .exe file of the SW. An example of a generated netlist is shown in Figure 13.

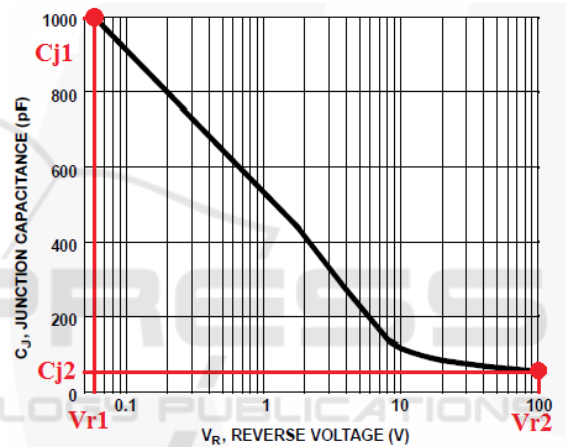


Figure 11: Two points selected on the junction capacitance characteristic.

$C_{rr}$	Reverse Recovery Charge	-	450	nC
$t_{rr}$	Reverse Recovery Time	$I_f = 30$ A,	60	ns
S	Softness Factor ( $I_{rr}/I_f$ )	$di/dt = 1000$ A/μs,	1.25	-
$I_{rr}$	Reverse Recovery Current	$V_R = 300$ V,	21	A
$C_{rr}$	Reverse Recovery Charge	$T_C = 125^\circ\text{C}$	730	nC
$di_{rr}/dt$	Maximum di/dt during $t_{rr}$		800	A/μs

Figure 12: Reverse recovery datasheet's specifications.

```
* Diode isl9r3060g2 A K
.subckt isl9r3060g2 1 2
.param Temp_diode=125 k=1.381E-23 q=1.602E-19 Vt=0.03430948
.param Rs=0.02156586 ls=0.0004614982 N=2.460456 Tau=5.909138E-08
+Tm=3.181049E-08
.param VJ=2 Cj0=9.311509E-10 M=0.6974055 FC=0.5
R1 C1 1 (RS)
G1 0 qm VALUE={-1/tau*V(qm)+V(qe,qm)/TM}
C1 qm 0 1
E1 qe 0 VALUE={ls*tau*(exp(V(C1,2)/(N*Vt))-1)}
G2 C1 2 VALUE={V(qe,qm)/TM}
R3 qm 0 100G
D1 C1 2 Cap
.model Cap d (ls=1e-14 N=200 rs=10u Xti=0 Eg=0 Cj0={Cj0} M={M} VJ={VJ}
+FC={FC})
.ends isl9r3060g2
```

Figure 13: Generated netlist of new diode sub-model.

## 6 CONCLUSION

In the work that is being presented, the diode SPICE model is implemented together with an accurate simulation of reverse recovery. The study that is given is based on the physical representation that was first created in the source (Lauritzen and Ma, 1991). The current work offers automated diode model development based on datasheet specifications from the manufacturer. Open source software is publicly accessible for diode characterisation and model generation (ZAIKIN D.I., 2021).

## REFERENCES

- Analog Devices, Inc. (2008). LTspice - SPICE simulator software. <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>. Accessed 19-05-2023.
- Cadence Design Systems, Inc. (2015). PSPICE - Circuit Simulation. [www.pspice.com](http://www.pspice.com). Accessed 19-05-2023.
- JEDEC Standard No. 282B.01 (2000). Silicon Rectifier Diodes.
- Kielkowski, Ron M (1995). *SPICE: Practical Device Modeling*. McGraw Hill.
- Lauritzen, P. and Ma, C. (1991). A simple diode model with reverse recovery. *IEEE Transactions on Power Electronics*, 6(2):188–191.
- ZAIKIN D.I. (2021). Software tool for the article: Practical implementation of diode SPICE model with reverse recovery. <https://doi.org/10.6084/m9.figshare.14912769>. Accessed 19-05-2023.