Simulation

SPICE Device Models: Diode Example–Part 1

2019.05.09

Points of this article

- There are two types of SPICE models: “device models” and “subcircuit models”.
- In SPICE device model, each parameter value is set.

In the previous article it was explained that models used in SPICE simulations can be device models or subcircuit models. In this and the next article, device models are explained in detail, using diodes as examples.

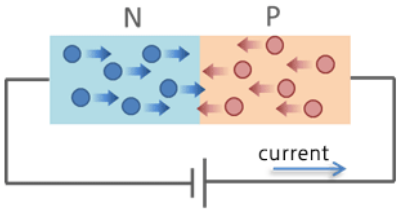
SPICE Device Models: Principle of Operation of Diodes

Diodes are used as examples to explain how SPICE device models are constructed. To this end, we first review the principle of operation of a diode. Here semiconductor theory is involved, but what is important is the parameters used in the device model, and so there is no real need for an understanding of the theory itself.

The diagram below illustrates the basic principle of operation of a silicon diode. When a bias voltage is applied to the diode, the electric field in the depletion layer weakens, carrier diffusion occurs, and a current flows. The forward current Id and the depletion layer capacitance Cj of the diode can be

Simulation

- Optimization of Inverter Circuits －Introduction－
- Foundations of Electronic Circuit Simulation Introduction
- About the ROHM Solution Simulator
- Optimization of PFC Circuits



Forward current

$$Id = Is \cdot \left\{ \exp\left(\frac{q}{KT} V \right) - 1 \right\}$$

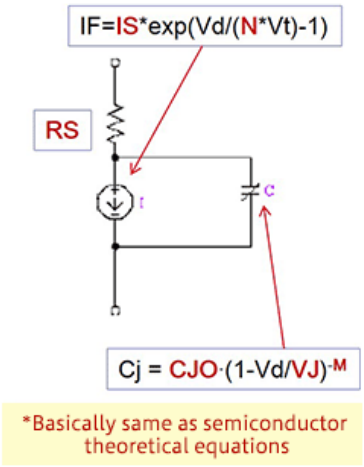
Depletion layer capacitance

$$Cj = Cjs \Big/ \left(1 - \frac{V}{Vbi} \right)^{M}$$

Diode Parameters

Parameter	Description	Unit	Default Value
AF	Flicker noise exponent	-	1
BV	Reverse breakdown knee voltage	volt	Infinite
CJO	Zero-bias p-n junction capacitance	farad	0
EG	Bandgap voltage (barrier height)	eV	1.11
FC	Forward-bias depletion layer capacitance coefficient	-	0.5
IBVL	Low-level reverse breakdown knee current	amp	0
IBV	Reverse breakdown knee current	amp	1.00E-10
IKF	High-injection "knee" current	amp	Infinite
IS	Saturation current	amp	1.00E-14
ISR	Recombination current parameter	amp	0
KF	Flicker noise coefficient	-	0
M	p-n junction grading coefficient	-	0.5
N	Emission coefficient	-	1
NBV	Reverse breakdown ideality factor	-	1
NBVL	Low-level reverse breakdown ideality factor	-	1
NR	isr emission coefficient	-	2
RS	Parasitic resistance	ohm	0
TBV1	bv temp. coefficient (1st)	$°C^{-1}$	0
TBV2	bv temp. coefficient (2nd)	$°C^{-2}$	0
TIKF	ikf temp. coefficient (1st)	$°C^{-1}$	0
TRS1	rs temp. coefficient (1st)	$°C^{-1}$	0
TRS2	rs temp. coefficient (2nd)	$°C^{-2}$	0
TT	Transition time	sec	0
T_ABS	Absolute temperature	°C	
T_MEASURED	Measured temperature	°C	
T_REL_GLOBAL	Relative to current temperature	°C	
T_REL_LOCAL	Relative to AKO model temperature	°C	
VJ	p-n junction potential	volt	1
XTI	IS temp. coefficient	-	3

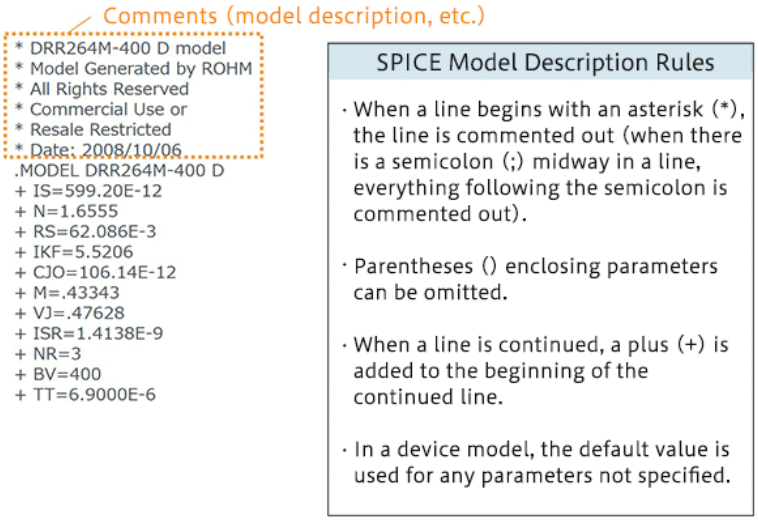
The diagram below is the equivalent circuit of a diode. The forward current IF (represented by Id in the equation above), depletion layer capacitance Cj, and parasitic resistance RS are shown. Upon substituting the diode parameters (see the table) into the above equations, we obtain the following model equations.



Model equations are essentially the same as semiconductor theoretical equations. In a device model, these parameter values are set.

SPICE Device Models: Case Study of Diode Device Model

A device model of an actual diode is presented. This model is provided by



Although mentioned in the list of description rules, lines beginning with an asterisk (*) are commented out. Simple explanations of the model and the like can be provided as desired. In this example, the actual description begins with “MODEL”. Lines beginning with pluses (+) set parameter values. Please compare the device model description and the parameter table. When a default value can be used as the parameter setting, a value need not be stated.

(Continued to Part 2)

Information on Downloading Technical Documents

Downloadable materials, including lecture materials from ROHM-sponsored seminars and a selection guide for DC-DC converters, are now available.

See list of Documents

List of articles related to the「SPICE Device Models: Diode Example–Part 1」

- Foundations of Electronic Circuit Simulation Introduction
- About SPICE
- SPICE Simulators and SPICE Models

spice-diode-device-model | spice-device-model | spice-subcircuit-model | spice-model | electronic-circuit-simulation | electronic-circuit-simulator | ads | analog-devices | pspice-community | hspice | cadence | keysight-technologies | eldo | linear-technology | ltspice | mentor-graphics | orcad | simetrix | simetrix-technologies | spectre | synopsys | pspice | spice