

**PACKAGE ID** - 001135MLTPL00 SPICE-SANDIA.LIB

**KWIC TITLE** - Library Analog Semiconductor Devices SPICE  
Simulators

**AUTHORS** - Deveney, M.F.  
Sandia National Labs., Albuquerque, NM (United States)

Archer, W.  
Sandia National Labs., Albuquerque, NM (United States)

Bogdan, C.  
Sandia National Labs., Albuquerque, NM (United States)

**LIMITATION CODE** -COPY                   **AUDIENCE CODE** - LIM

**COMPLETION DATE** - 06/06/1996   **PUBLICATION DATE** - 06/06/1996

**DESCRIPTION** - SPICE-SANDIA.LIB is a library of parameter sets and macromodels of semiconductor devices. They are used with Spice-based (SPICE is a program for electronic circuit analysis) simulators to simulate electronic circuits.

**PACKAGE CONTENTS** - Media Directory; Software Abstract; Media Includes Text Library;

**SOURCE CODE INCLUDED?** - No

**MEDIA QUANTITY** - 2 3.5 Diskettes

**METHOD OF SOLUTION** - This library is used with SPICE (Simulation Program with Integrated Circuit Emphasis) simulation methods (trapezoidal method).

**COMPUTER** - MLT-PLTFM

**OPERATING SYSTEMS** - NA

**PROGRAMMING LANGUAGES** - Proprietary Language

**SOURCE CODE AVAILABLE (Y/N)** - N

**UNIQUE FEATURES** - Temperature dependent models, temperature tracking models, radiation models.

**OTHER PROG/OPER SYS INFO** - This is version 2 of SPICE-SANDIA.LIB. The library is to be used with PSPICE or SPICE2G.6 compatible simulators.

**HARDWARE REQS** - NA

**TIME REQUIREMENTS** - NA

E S T S C  
ENERGY SCIENCE & TECHNOLOGY SOFTWARE CENTER  
SOFTWARE ABSTRACT

PAGE 2  
DATE 03/12/2002

**PACKAGE ID** - 001135MLTPL00 SPICE-SANDIA.LIB

**ABSTRACT STATUS** - Submitted 12/4/96. Released AS-IS 1/10/97

**SUBJECT CLASS CODE** - T

**KEYWORDS** -

COMPUTER PROGRAM DOCUMENTATION  
S CODES  
SEMICONDUCTOR DEVICES  
COMPUTERIZED SIMULATION  
INTEGRATED CIRCUITS  
MATHEMATICAL MODELS  
TEMPERATURE DEPENDENCE

**EDB SUBJECT CATEGORIES** -

990200 426000

**SPONSOR** - DOE/DP

**PACKAGE TYPE** - AS - IS