



**Model Parameter
Editor**

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the Model Parameter Editor document can be reproduced in any form or by any means without the prior written permission of Visionics. Model Parameter Editor document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems.

Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

Visionics, EDWinXP, Docone, EDComX, SimWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

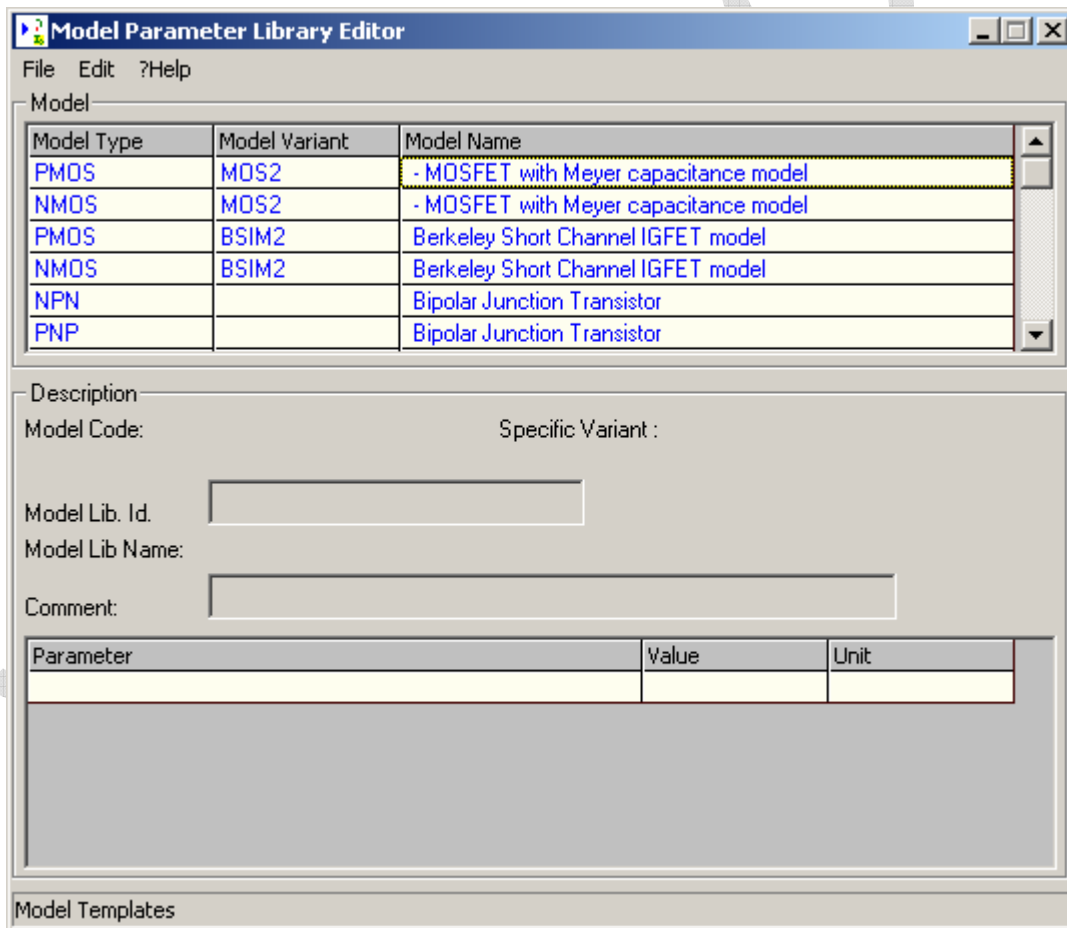
Contents

CONTENTS	2
MODEL PARAMETER EDITOR	3
HOW TO USE THE MODEL PARAMETER EDITOR.....	3
STARTING THE MODEL PARAMETER EDITOR	4
CREATING NEW MODELS	4
LOADING MODELS FROM A LIBRARY FILE	5
EXTRACTING MODELS FROM AN EXISTING SPICE NETLIST.....	5
MODEL LIBRARIES IN EDSPICE	6
RECREATE MODEL PARAMETERS LIBRARY INDEX.....	9

Model Parameter Editor

How to Use the Model Parameter Editor

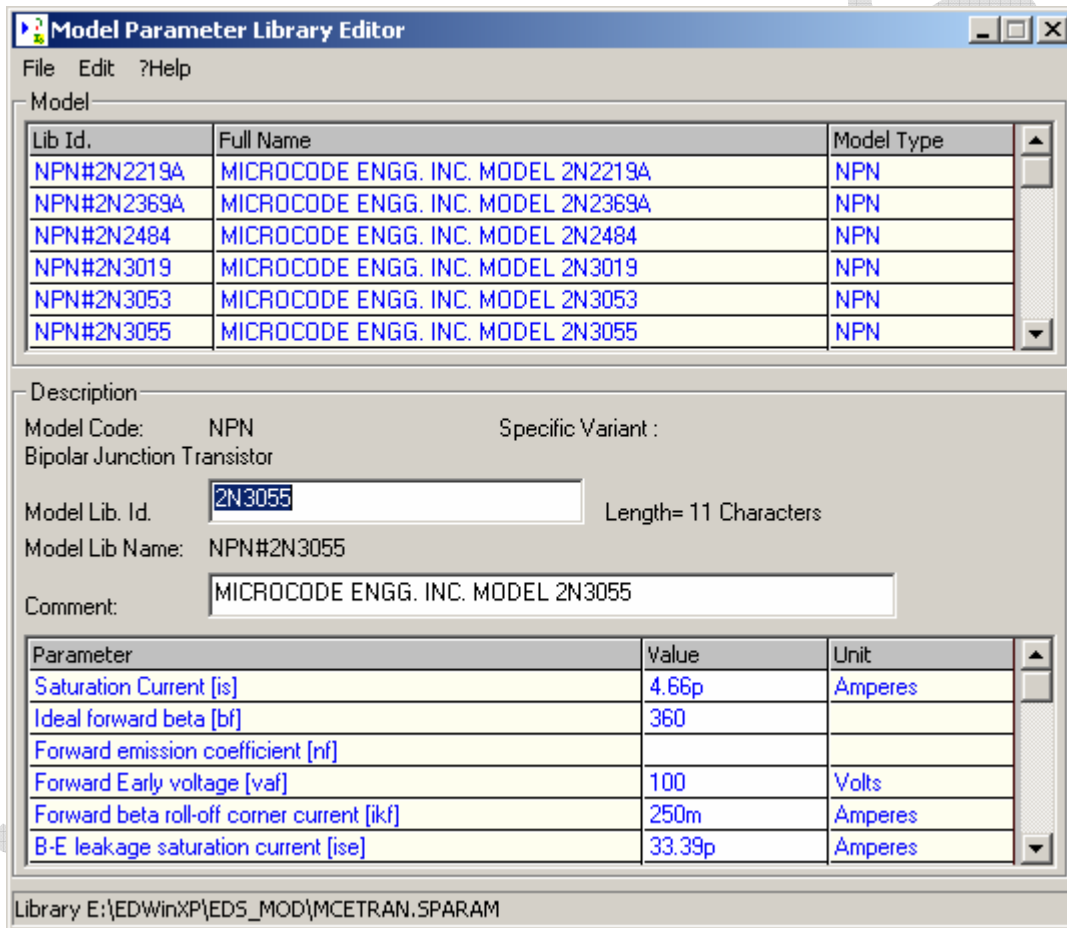
The Model Parameter Editor eases the management and creation of model libraries. It can be used in a variety of ways including extracting .MODEL lines from existing SPICE netlist, loading existing models from the model libraries allowing you to modify and re-save them, and the creation of new models.



Starting the Model Parameter Editor

This module may be invoked from Project Explorer in the following ways.

Right click **System** → **Model Parameter Editor** from the list. By default, the task toolbar will not be displayed. It may be enabled from View menu in the Project Explorer.



Creating New Models

- First select the type of template for which you want to create a model from the list of templates displayed in the section 'Model'.
- On selecting a template the type of model, a description of the model and a list of the available parameters are displayed. To edit any of the parameters just click on the cell

Value' to get the edit box. Change the value and press the ENTER key on your keyboard to accept the new value.

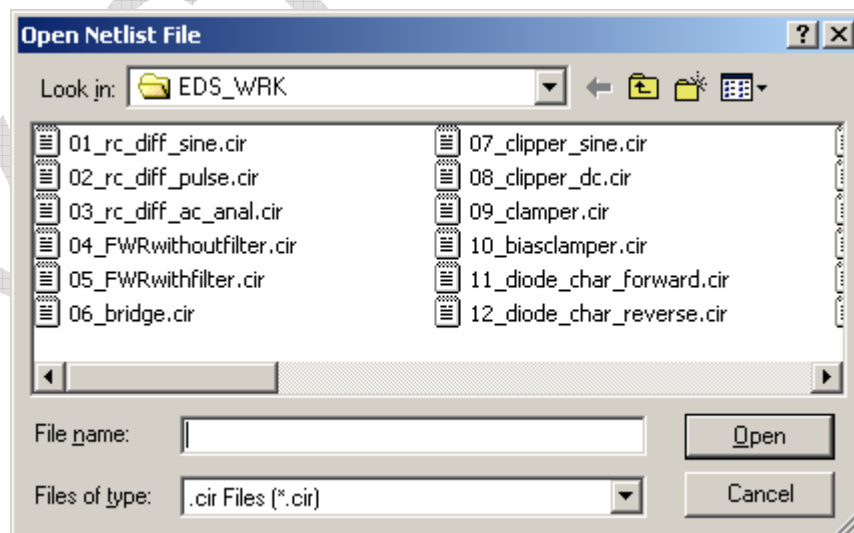
- Once you have finished editing the parameters select File/ Save Model to save the model. You have to supply a model library ID, from which the model library name for this model is generated, and a comment/ description about the model. You can either select an existing library file to save this model into or specify a new file name.

Loading Models from a Library File

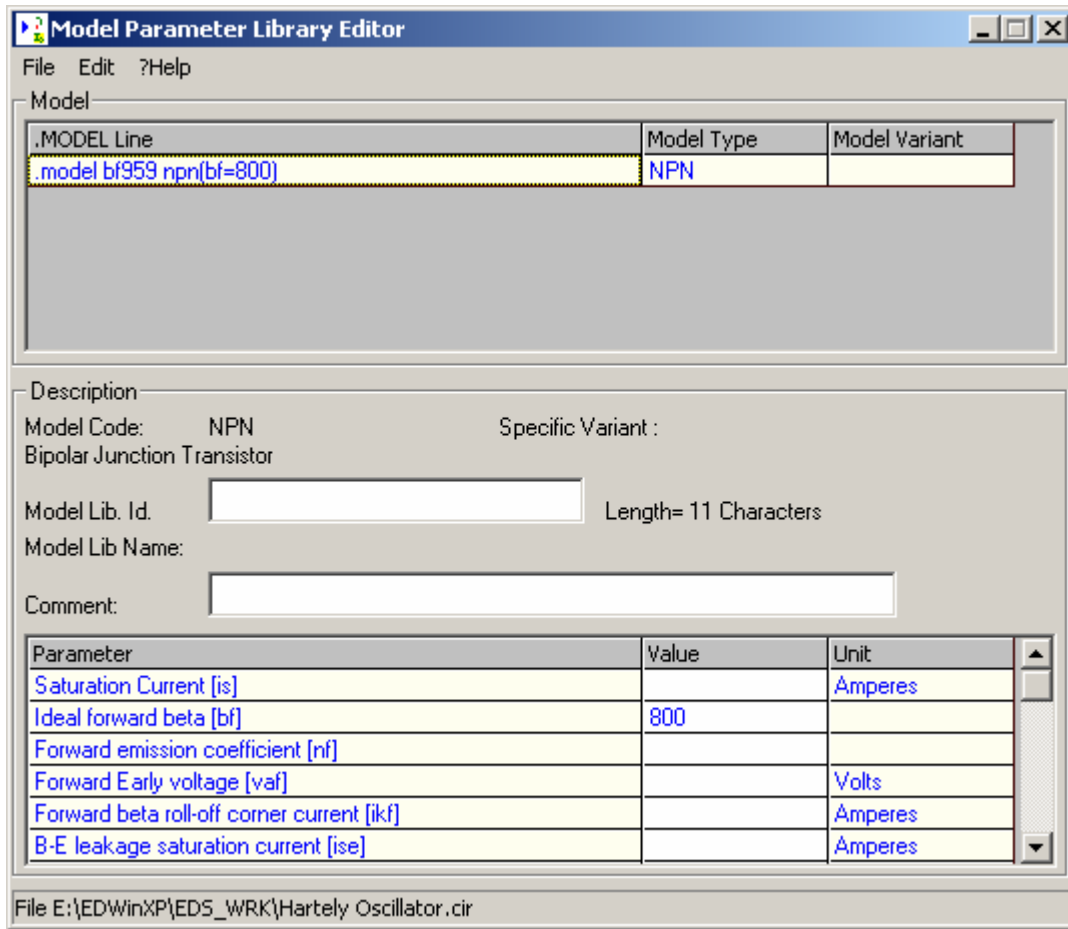
- Select File/ Open Model Parameters Library. From the Open Library window, select a model library file by clicking on the name of the file you want from the list supplied. A list of model names will then be displayed. Once you have selected a model its parameters will be displayed. Any modifications can then be made by clicking on the column 'Value' to edit the values corresponding to the various parameters. Select File/ Save Model to update this file.
- You can delete the selected model from the library by selecting Edit/ Delete.

Extracting Models from an Existing SPICE Netlist

- Select File/ Extract from Netlist. From the Open Netlist File window select a file (.cir) and click the OPEN button to open the existing SPICE netlist file.



- Once a file has been selected a list of .MODEL lines, from the SPICE netlist file, are displayed. Click on a line to select it.



· On selection, the parameters of the selected model will be displayed in a table. Enter the Model Id and the Comment. Edit the values, if required, for the various parameters by clicking on the column 'Value' to get the edit box. After setting the parameters, save the model to an existing model parameter library or to a new library.

Note: EDSpice Simulation References are assigned to parts and not to symbols.

Model Libraries in EDSpice

There are three kinds of model libraries used by EDSpice. The first two types facilitate the simulation of a wide range of circuit elements whose functionality is defined by software extensions to the simulation engine. The behavior of the element is encoded in a dedicated software module, which is dynamically linked to the simulation engine during runtime. This feature of the EDSpice Simulator is called code models. The program modules which contain this

functionality, and currently available in EDSpice, are stored in a separate subdirectory on the hard disk. This library of code models is automatically created while the EDSpice package is installed on the user's machine. The EDSpice Code Model Library contains program modules allowing for simulation of more than a hundred digital and analog circuit elements.

Similarly, the second type - library of *User-Defined Nodes* - is also available with the EDSpice Simulator. User-Defined Node types allow you to specify nodes that propagate arbitrary data structures and data other than voltages, currents, and digital states. User-Defined Nodes communicate with the simulation engine in the same manner as the code models. The User-Defined Nodes Library is automatically installed with the rest of the EDSpice package.

According to SPICE convention, the characteristics of a circuit element are defined by part parameters, on the element line of a netlist, representing an instance of the element. However, some complex elements, such as transistors, are characterized by both part parameters, defined on the element line, and model parameters, grouped in a .MODEL statement. The model statement allows you to specify only one set of parameters common to a number of elements, for example, the parameters of all transistors with the same geometry integrated on one silicon chip. Therefore, you can have multiple instances of a model.

All part and model parameters of circuit elements should be properly set before simulation is executed. It applies without difference to elements defined by external code models and models implemented internally by SPICE. For the same type of element, each set of different parameter values describe characteristics of a specific, real life electronic component. These values are normally listed in data books and catalogs supplied by manufacturers. In many cases, associated with BJT's, MOSFETs and other more complicated circuit elements, the process of setting up these parameters may be quite tedious, very often requiring a good knowledge of the internal working of the component.

The EDSpice package allows for the maintenance of a preset model parameters library. Parameters of every model once set for simulation in one circuit may be stored on the hard disk in a selected library file and reused in other circuits. This library is called Model Parameters Library and is the third type of library of the EDSpice package.

Anticipating problems with model parameter setup, the manufacturers of electronic components supply data files in SPICE compatible format with the appropriate model parameters describing characteristics of their products. The interactive module of EDSpice includes features which can extract .MODEL lines from files compatible with the SPICE netlist format and append the set of parameters contained in one line as a single item in the Model Parameters Library. This feature is

also useful for designers who already use other SPICE simulators and wish to add models from existing netlist files. The feature is called the Model Library Editor.

It is assumed that users will enhance the Model Parameters Library. During installation of the package, the subdirectory intended for this library is automatically created on the hard disk.

The model parameters are stored in the library, using an internal format, and are accessible only for usage by a circuit elements setup features within the interactive module.

For identification purposes, the models are tagged with library names, which may be up to 15 characters long. Within one library file, the names must be unique and must conform to certain naming conventions. According to this convention, aimed for easier recognition of the models, the general format of the model library name is as follows:

prefix#userID

Prefix (with a maximum length of 7 characters) is automatically added to the model library name. It is always the type of the model according to SPICE syntax. For example, all models containing parameters for NPN transistors will be assigned the prefix NPN#. Users may add their own ID to the model name, so that the total number of characters including the prefix# does not exceed 15. Apart from the name, each model in the library may contain a description provided as additional identification of the models.

The model library structure also supports the family approach of grouping the models. There are two purposes to group model parameters family libraries.

1. The user may find the models easier.
2. It may happen that model parameters may have the same identifier. For example BC807 from two manufacturers, Motorola and NS. In such cases they may be stored with the same identifier in different family library file and still be unique.

The grouping of models from a particular manufacturer are made to a single .Sparam file. For example transistors from MOTOROLA have been saved to MOTOBJTS.Sparam under the directory ../EDS_MOD. Likewise the grouping is done for models from different manufacturers.

Apart from the .Sparam files representing models from different manufacturers, there are two more files - MISC.Sparam and MIXMODE.Sparam. SAMPLES.Sparam contains model parameters used by samples. MIXMODE.Sparam is used by the system.

Family Prefix may be used (optionally) to connect a part to specific model parameter library file. EDSpice Reference for models looks in following way:

MMM#IIII

Where MMM is Model Type (ex PNP) and ILLL is identifier (ex BC807). When PNP#BC807 is coded as EDSpice Ref then the program would search through all library files in the Model Parameters Library until it finds or not finds the PNP#BC807.

The Family Prefix is optional (settable in EDSpice Symbol Editor) and is nothing else but model parameters library file name (no extension). Hence if it is decided to use Family prefix and if the parameters are stored in MOTOBJTS.Sparam then the EDSpice Reference would be as follows:

MOTOBJTS.PNP#BC807

The program will try to load this parameters from MOTOBJTS.Sparam. If unsuccessful then it would work like without family prefix - meaning that it will search through other files in this subdirectory.

The model library name may be used as the Simulation References to assign these models to parts. Such components created as parts with appropriate Simulation Reference set automatically attaches model parameters corresponding to these Simulation Parameters while preprocessing is done. This is a one time activity, i.e., model parameters pointed to by the Simulation Reference is loaded only if the component does not have any model parameters set. Once model parameters are changed by the user, the new parameter set always overrides the default ones set using Simulation Reference.

Note: EDSpice Simulation References are assigned to parts and not to symbols.

The component has to be packaged before pre-processing by EDSpice. The different operations of the menu items are as listed below

Recreate Model Parameters Library Index

Recreate model parameter library index allows to recreate model parameter library and stores in Library Index file (.IDX). Thereby, enhancing the Speed of Model Parameter Library references at the loading time of EDSpice.

Select **Edit → Recreate Model Parameter Library Index** after making necessary changes to the Model Parameters. A building process takes place and stores the recreation in .IDX file. The default path is /EDWinXP folder.