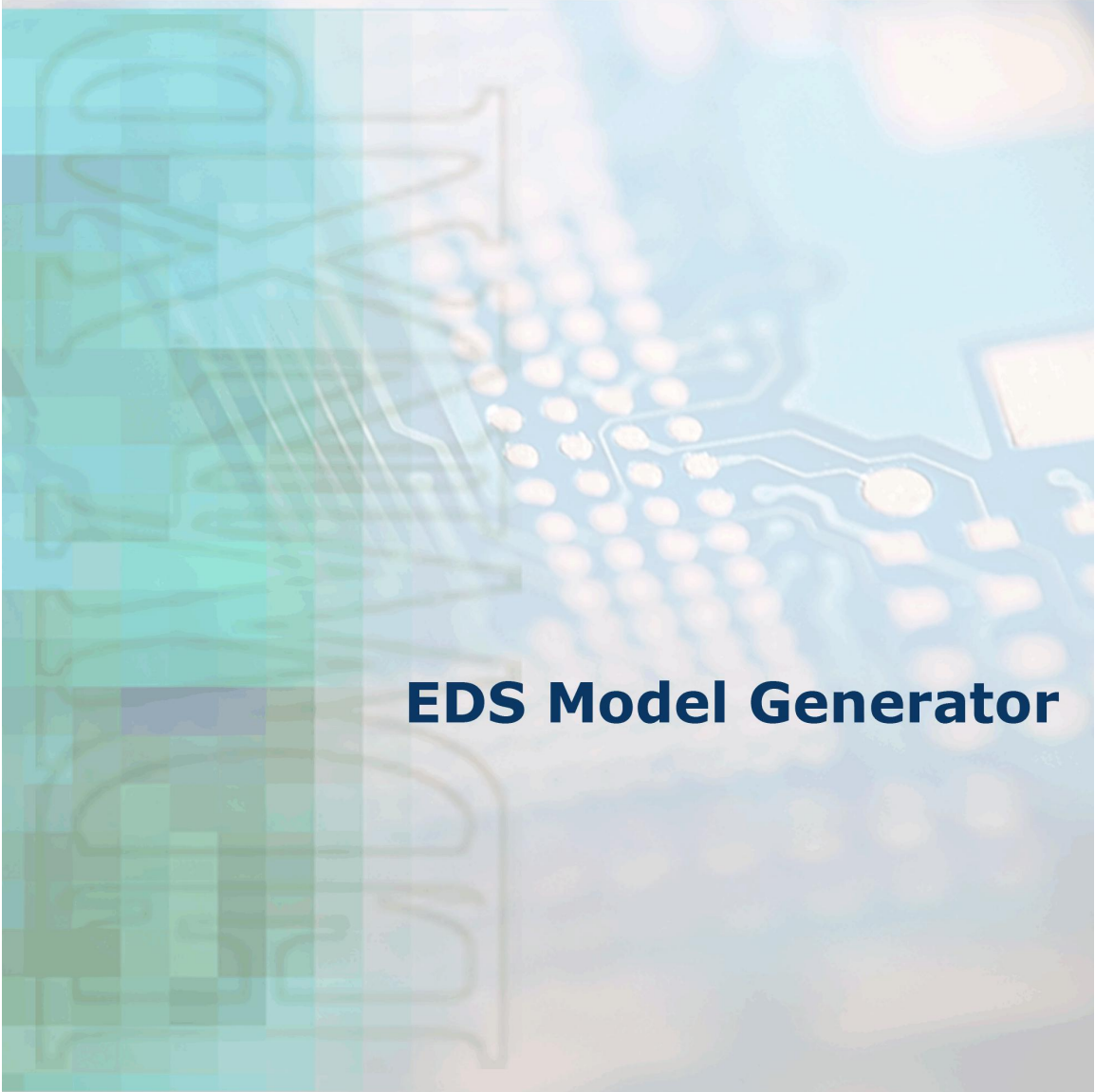


EDWIN XP
ELECTRONIC DESIGNER FOR WINDOWS



EDS Model Generator

VISIONICS

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved. No part of the EDS Model Generator document can be reproduced in any form or by any means without the prior written permission of Visionics. EDS Model Generator 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

EDS Model generator	4
Invoking EDSpice Simulation Model Generator	4
Operation	5
Generating models for a wrs file	5

EDS Model generator

EDS Model Generator converts VHDL source file to a simulatable component in EDSpice, the SPICE based simulator (which is a plug-in). This also serves the purpose of making simulation much faster.

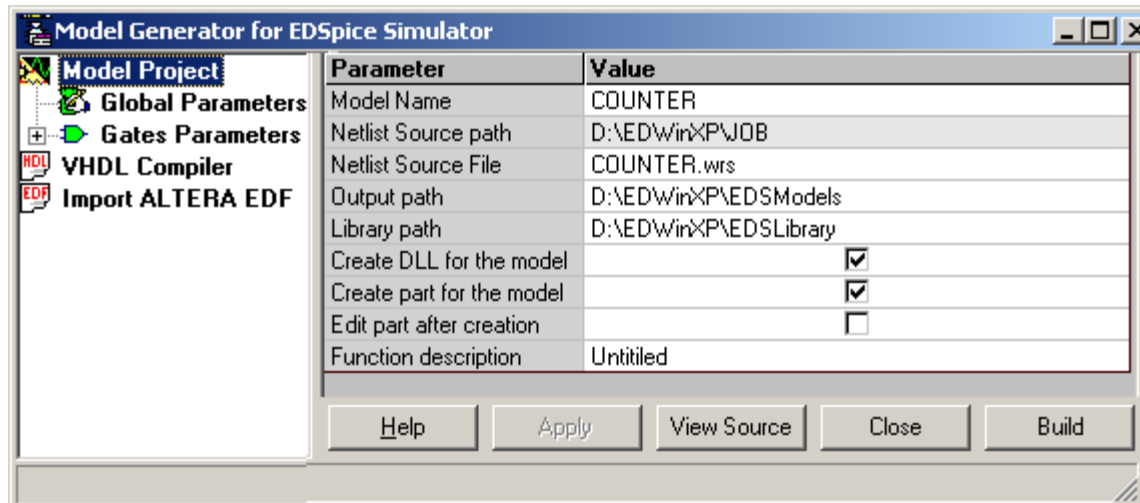
Since VHDL can only be used to create digital models, only digital simulation models can be created using EDSpice Simulation Model Generator. Earlier, knowledge of programming was essential for writing digital models. It would be much easier for an electronic engineer to create the models using VHDL rather than learn other non-standard programming languages for creating models. Therefore a need for developing a tool arose which could convert VHDL models to SPICE models.

Note: *EDComX is a tool which can be used for creating both analog and digital models, however it assumes programming knowledge and the user has to build his own simulation SPICE Model.*

Invoking EDSpice Simulation Model Generator

- Select EDSpice Simulation Model Generator from the tasklist or task toolbar of the Project Explorer.
- Select EDSpice Simulation Model Generator from the floating menu that appears on right clicking System menu in Project Explorer.

A window titled Model Generator for EDSpice Simulator appears



Operation

Double click once in the cell next to VHDL Source File name and an ellipsis will appear. Click on it to select the file. As soon as the file is selected, EDSpice Generator will compile the VHDL file and exhibit the results in File Viewer windows. The errors in the VHDL file (if any) have to be corrected before proceeding. Once the VHDL file is error free, the focus automatically moves to the Model Project window.

Generating models for a wrs file

1. **EDWinXP → Project Explorer → System → EDS Model generator**
2. Model name → Give a proper model name
3. Source Netlist file → Select appropriate Netlist file
4. MM function description → Give a description
5. Click on Build
6. This will create a special part in the MMMGEN.PART in the library explorer, which can be used as simulatable components.