EDWinXP

Electronic Design for Windows

ELECTRONIC DESIGN FOR WINDOWS

EDWinXP - An Integrated EDA Software Package

EDWinXP (Electronic Design for Windows) is a CAD/CAE software package of seamlessly integrated, task oriented modules covering all stages of the electronic circuit design process - from capturing the idea of a circuit in the form of schematic diagram to generate a full set of documentation for manufacturing and assembling of PCBs. Additionally the package includes various validation tools ensuring correctness and integrity of designed circuits. A design engineer can use the computerized tools provided in EDWinXP to create an electronic circuit, design the PCB and fabricate PCB. Complete design information is stored in the integrated project database, simultaneously accessible by Schematic Diagram Editor, PCB Layout Editor, Fabrication Output Manager and Simulators. Front and back annotation of all design changes is fully automatic. EDWinXP comes with extensive part library, which may be updated, customized and enhanced with the help of Library Editor.

Schematic Diagram Editor

The Schematic Diagram Editor contains a full set of manual and automated tools for placement of circuit elements on the diagram and for routing the connection. Project database supports circuits with multiple hierarchies (99) and each

hierarchy supports multiple page (99). Components are loaded either from library or from Schematic Browser. Connections are defined either by creating named nets and adding nodes or by physically drawing wires / busses. The netlist automatically updates itself when wires / busses are edited. Once the netlist has been created components may be automatically placed and wires automatically routed from scratch. Circuits defined in VHDL or as SPICE format netlists may be imported and converted into schematic diagrams. Automatic component packaging routines generate pin-out texts on the diagram and prepare the circuit for PCB layout design.

Subsequent design changes in real-time are front annotated on the PCB layout. Design rules may be used for auto placement and interactive auto

routing. Additional graphical and textual information may be created in the form of design notes (common to all diagram pages) and individual page notes.
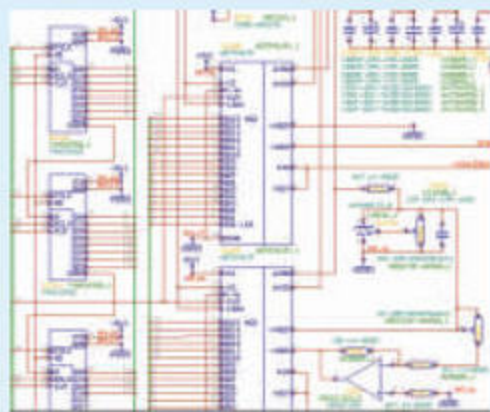
Circuit Simulators

The functioning of the circuit may be tested with the help of integrated simulators in EDWinXP. Mixed Mode Simulator, the system's native circuit level analyzer and EDSpice, the full implementation of XSPICE as defined by Georgia Tech are the two simulators in EDWinXP.

Mixed Mode Simulator supports TD, DC, AC, Parameter sweep, Fourier, Monte Carlo and Sensitivity analyses of analog, digital, and mixed-signal circuits. Extensive EDWinXP library contains simulation models linked to elements in the parts library. The package includes tools for creation of user-defined digital simulation models.

EDSpice also supports analyzes of mixed mode circuits. Extensive library of models (code models) for simulation of digital circuit elements comes with the package. Sub-circuits provided by manufacturers of electronic components are available readily and may be included in the EDSpice sub circuit's library. Results can be viewed either in wave form viewer or on a virtual CRO.
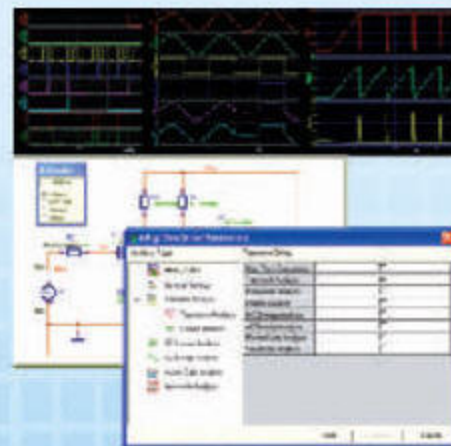

Microcontroller Kit

This kit contains project databases designed to illustrate functionality of 8051, AVR AT90S2313, PIC16C5X, and PIC16X84. It uses MMI technology that enables edition and compilation of programs in C and assembly language, and it also allows debugging the code in real time. The kit contains series of instrumental models such as memories, interrupt generators, serial/parallel pattern generators, 7-segment displays etc. for generating

parallel and serial binary data patterns and asynchronous hardware interrupts.

Filter Designer

Automated filter Designer lets you choose the type of filter and enter the output frequency and quality range. The system will automatically calculate the parameters and then generate the circuit in Schematic Editor, which can be imported to the diagram and then use as part of other circuits.
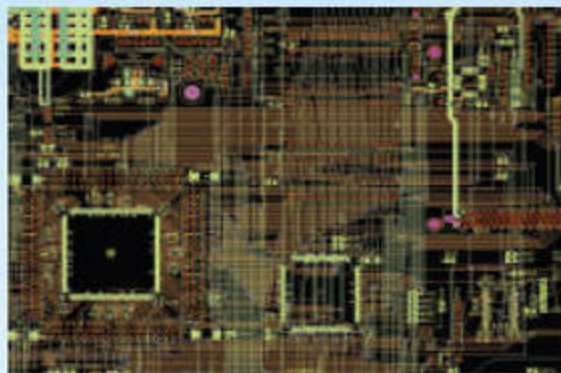
PCB Layout Editor

EDWinXP project supports design of 32-layer boards (28 trace layers, 2 silk-screen and 2 solder masks). Components are created automatically as a result of packaging executed while editing the schematic diagram of the circuit. New parts also may be inserted on the board in similar fashion as in the Schematic Editor. Location and orientation of components is defined either by manual relocation to desired position or with the help of the auto placer.

Traces may be routed manually with automatic via insertion whenever a routing layer is changed. Sixteen types of user defined via pads are supported. The Editor includes a number of online automatic functions to route/reroute traces for single nets and to reroute existing traces for relocated components. A dedicated "full board" auto router module is integrated with the PCB Layout Editor. EDWinXP also has length and width trimming feature which can be done interactively, it allows inserting pattern of segments into selected trace in order to obtain exact specified length.

Copper pour areas are defined as polygons and may be placed on any trace layer. Insertion of air gaps and thermal pads on artworks is automatic. Design rules for manual, semiautomatic and automatic routing of traces and component placement are user defined and may be set individually for each project. Design rules violation, clearance errors and missing or incomplete connections are also detected automatically.

All changes introduced in the circuit design are automatically back annotated to schematic diagram.

Board Analyzers

The Electromagnetic Analyzer and Thermal Analyzer checks the integrity and correctness of layout design.

Electromagnetic Analyzer presents graphically the predicted intensity of electromagnetic fields inside and outside board boundaries. The Signal Integrity Analyzers detects distortion, noise and crosstalk for critical signals.

The temperature distribution on a finished PCB may be analyzed graphically with the help of Thermal Analyzers. The result of the analysis is displayed using isotherms or color mapping schemes.
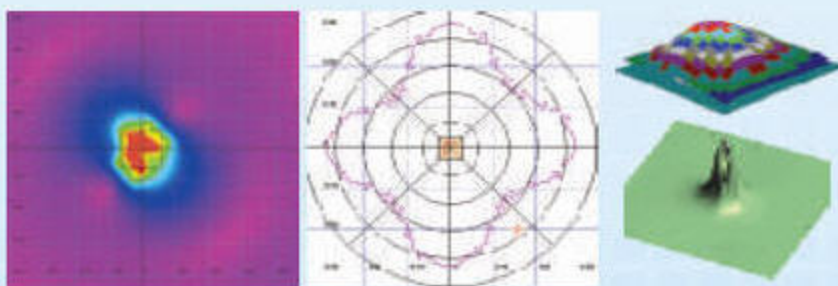
Fabrication Manager

All CAM functions of EDWinXP are grouped in this module. The user has the option to add targets, coupons, thieving and venting areas. Copper pour areas are checked for possible shorts or areas isolated by air-gaps. Automatic or manual

copper removal is also implemented. Final artworks are prepared for photo plotting. EDWinXP supports both standard (RS-274D) and extended (RS-274X) Gerber ASCII formats. The powerful Gerber ASCII file viewer is integral part of this module, enabling verification of artwork prior to plotting and fabrication.
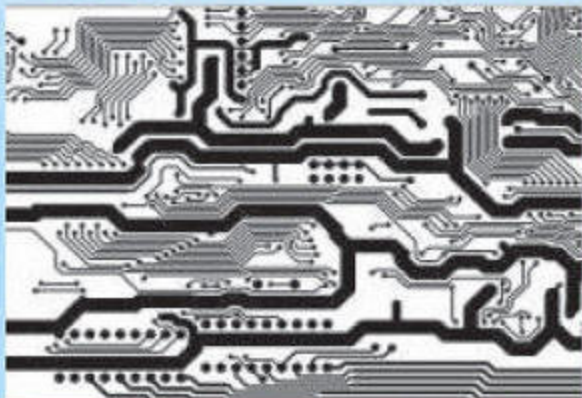
NC Drill data are generated in Excellon format with or without tool movement optimization. Drill templates are created optionally and may be printed or saved as Gerber files for photo plotting. This module includes functions to create mechanical drawings of the PCBs.

PCB Assembly output is generated in IPC-355 and Bare Board Testing output in IPC-356A formats. The whole project database may be also exported in GenCAM format.

ODB++ Export / Import

ODB++ Export allows exporting the selected project contents to ODB++ format. And ODB++ Import allows importing projects of CAD packages which support ODB++ formats to EDWinXP.

Library Editor

EDWinXP component libraries may be updated, customized or enhanced with the help of Library Editor. Functionality of this module allows definition of graphical representation of components in schematic diagram (Symbol Editor) and on Printed Circuit Board (Package Editor). These elements are included then in the component description, which also contains the packaging information, thermal parameters and link to simulation modules.
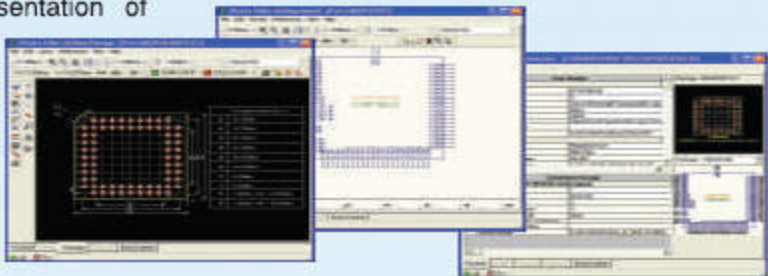
3D Editor and Viewer

EDWinXP may present designed PCB in "real life" three dimensional views. In order to make it possible, 3D views of component packages (created according to IPC, JEDEC and EIA standards.) are included in the system part library. Tools are provided in Library Editor to create and edit 3D views for newly created packages.

VHDL Editor and Compiler

VHDL source files (Level 0 syntax) may be created with VHDL Editor. This may then be compiled and imported to EDWinXP as a Project. This project may then be exported to netlists of the format CUPL, XILINX, JEDEC etc.
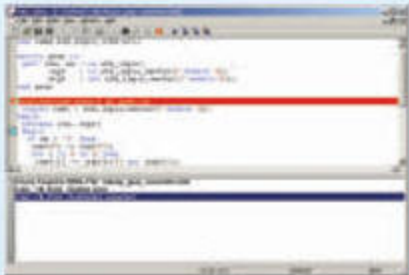
List of Material Editor

List of Material Editor generates customized Bill of Materials that can be imported to Excel for further processing.

Project Version Control

Keep track the changes in EDWinXP projects.

Features

Schematic Editor

- Top down Hierarchical Circuits (99)
- Up to 99 pages
- Industry Standard Page Sizes
- Customizable Component browser
- Definable connection and bus width
- Intelligent and interactive routing
- Auto packaging
- Instant Packaging
- Smart Autoplacer
- Auto covering
- Circuit DRC
- Direct assignment of logic function to schematic components
- Component Specific MM Simulator Models
- Filter Designer
- Page / Design notes
- Property boxes

- Block Diagram Elements
- Truth table to diagram converter
- VHDL code to diagram converter

PCB Layout Editor

- 32 layers (28 copper ,2 mask & 2 silk screen)
- Component browser
- Smart Autoplacer
- Auto router
- Trace pattern repeat
- Online trace clearance check.
- Automatic "trace project" feature
- "XOR-ed" display of traces
- Trace Length and Width Trimming
- Curved Traces
- DRC

- Test Point
- Connectivity Check
- Router and Placer Keep off Zones
- Automatic correction of selected clearance errors
- Automatic clearance correction after routing and rerouting
- User definable cutouts, regions and templates
- View Holes
- Through Hole and Buried Vias
- Tear drop pads
- Hatch copper plane
- Copper pour tools
- Copper pour connectivity test
- 3D View of Layout

Fabrication Manager

- Artwork Venting and Thieving Areas
- Excellon NC Drill format
- IPC Standard for PCB assembling and Bare board testing
- Intelligent Copper Pour
- Gerber output with user defined apertures
- Supports EIA standards, RS-274D format and RS-274X
- Automatic ground plane with heat relief item
- Automatic Dimensioning (Linear and Angular)
- Gerber Viewer (WYSIWYG)
- Circular Interpolation
- Filled polygons
- Square flash fill of rectangles
- Artwork stepping
- Graphic import (Reverse Engineering)
- DXF Board Outline Import
- Simple means for aligning images
- Export CNC data in G-CODE
- Improved multi layer copper pour connectivity check with 3D presentation
- Polygonal copper relieve
- 3D Export in IDF Format
- 3D IDF File Viewer
- Copper Removal Tools
- Add Solder Paste/Glue Mask Items to pad stacks
- Export Formats in common dialog in the fabrication manager
- Improved Gerber Viewer Setup
- Square Holes in the Gerber Artwork
- Improved Reconstruction of projects
- Enhanced Copper Removal

Library

- ANSI & IEC Standard symbol Library
- JEDEC/EIA and IPC Standard packages
- Symbol and Package Creation Wizards
- Automatic pin assignment
- 3D creation/editing of packages
- Rectangular, Triangular, Oval & Circular Padstacks

Simulation

- Mixed mode Simulator
- EDSpice Simulator
- Logic Analyzer
- Multi channel Oscillograph
- EDSpice interactive Interpreter
- Model interface for Mixed Mode Simulator
- Circuit File Editor
- Instant Probes
- Model Generators
- VHDL to SPICE Model
- VHDL to Mixed Mode

Board Simulators

- Thermal Analyzer
- Electromagnetic Analyzer
- Signal Integrity Analyzer
- Field Analyzer

Other Features

- Look and feel of Windows
- Complete back and front end annotation
- Seamless integration between all modules
- Dynamic changing of working languages
- Shortcut and hotkeys
- Automatic file backup
- Import of old EDWinXP databases and Libraries
- Semi auto functions with user preference
- Real time display of ratsnest, active nodes, single line or true trace width
- Merging and intelligent splitting of nets
- Screen hardcopy
- Switching on/off facility for tools and scrollbars
- Monochrome mode for better print resolution
- Bitmap support
- Angular rotation with 0.01 degree Precision
- Editing of objects with bullets and property windows
- Unlimited nodes, nets, bend points, Connections
- Component gate and pin swap
- Block / save / load / move / delete
- Rotate / scale / mirror
- Pin to pin routing
- Reroute after move
- Integrated UNDO/REDO functions
- User definable text sizes
- Zooming facility, grids, ruler
- Precision of 1 micron
- Maximum board and page size of 4m x 4m
- Metric and imperial units
- Unlimited repeat
- True Type of Font supported

Export

- EDIF version 2.0
- EDWinXP Schematic netlist
- Scicards netlist
- Orcad PCB II wirelist
- EED3 Layout wirelist
- DXF output
- Specctra and Maxroute autorouters
- CUPL netlist
- XILINX netlist
- JEDEC netlist
- ALTERA EDIF
- Export CNC data in G-CODE
- ODB++ Export

Import

- EDWinXP PCB wirelist
- EDWinXP Schematic netlist
- Orcad PCB II wirelist
- VHDL (Level '0' syntax)
- Gerber ASCII (Reverse Engg)
- DXF
- EED3 Layout wirelist
- SPICE netlist (SPICE3F5 & XSPICE)
- ALTERA EDIF
- ODB++ Import

Online Support & Services

- Technical Support
- Library on Request
- Service Pack
- Library Updates
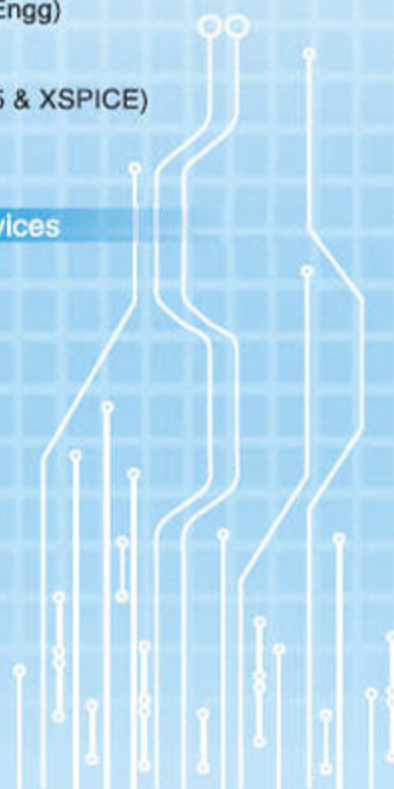
PRODUCT OF NORLINVEST SWEDEN

EDWinXP

ELECTRONIC DESIGN FOR WINDOWS

Integrated EDA/Circuit Simulation Software Package

Research & Development Centre

VISIONICS

VISIONICS (INDIA) PVT.LTD
A2, Periyar Technopark, Trivandrum-695581, India
Tel: 91-471-2700673 Fax: 91-471-2700270
Email : support@visionics.a.se | URL: www.visionics.a.se

DEALER / DISTRIBUTOR