OrCAD®

Bringing Electronic Design To The Desktop

README.WRI
12/12/95

THANK YOU FOR REVIEWING OrCAD CAPTURE FOR WINDOWS

Thanks for taking time to review the latest version of OrCAD Capture for Windows. Please take a moment to look through this file. It contains information that isn't addressed in the online help or in the Getting Started with OrCAD Capture for Windows guide. If there is other information you need, please contact OrCAD and we'll be glad to help you.

INSTALLING CAPTURE

Please refer to the Getting Started with OrCAD Capture for Windows guide for details on installing Capture and Win32s (where applicable).

STARTING THE CAPTURE DEMO

This demo version of Capture contains all the product's features. The only exceptions
are that it does not save or write design files (you can, however, produce BOMs, netlists, prints, etc.)

1. When the installation of the demo software completes, you'll have an opportunity to run the interactive tutorial. The lessons in the tutorial provide an overview of Capture's functionality and the excercises will lead you through some examples.

2. Double-click the Capture Help icon to review our online documentation.

3. Double-click the Capture demo icon to start OrCAD Capture for Windows.

4. From Capture's File menu, choose Open and then Design. The standard Windows File Open dialog appears. Sample schematic designs are located in the "SAMPLES" subdirectory. To load a sample design, just double-click on it.

5. Once a design is loaded you can open one of the schematics it contains by double-clicking on a schematic on the left side of the design manager. This displays all the schematic pages found in the selected schematic. To open a schematic page, just double-click on it.

6. To open a new, blank design or library, click on the leftmost button on the tool bar. Capture opens whatever was last active (a design or library).

NEW FEATURES IN CAPTURE FOR WINDOWS

INTUITIVE DESIGN ENTRY
* Online help and interactive tutorials make you productive quickly.

* "Point-and-click" selection uses standard Windows methods for multiple and block selection.

* Hot keys, toolbars, and tool palettes access common operations such as part selection, wiring, and drawing.

* Productive editing features include "press-and-drag" move with connection rubberbanding, fast double-click object editing and context-sensitive pop-up menus.

* Graphic part selector speeds part selection from 20,000+ part library.

* Point-and-click wiring automatically terminates a wire on a pin and creates a junction by clicking at a "T."

* Intelligent Repeat command lets you interactively set spacing and automatically increment labels.

* Easy-to-use drawing and text tools support TrueType fonts and user-definable colors, line, and fill styles.

INTEGRATED DESIGN MANAGEMENT TOOLS
* Design manager presents a structural view of all the schematic pages and parts in your design.

* Embedded spreadsheet editor allows you to add unlimited part property information to components, nets, and pins.

* Designs are fully contained in a single file, making them easy to transport.

* "Drag-and-drop" operation moves schematic pages and parts from one design or library to another.

ALL UTILITIES ARE AVAILABLE ONLINE
* Load and edit multiple designs or libraries simultaneously.

* Select a part from the schematic page for editing in the part editor.

* Output custom bills of materials and specify properties to be included.

* Active design rule checker that supports user-definable rules.

INTERTOOL COMMUNICATION (ITC) WITH OrCAD SIMULATE FOR WINDOWS AND OrCAD LAYOUT FOR WINDOWS
* Capture supports ITC with OrCAD's two newest Design Desktop for Windows products. The features supported include: cross-probing (part and net selection and highlighting between tools), the ability to exchange part and net properties between tools, annotation of signal values from Simulate to Capture, and forward and back annotation between Capture and Layout (including gate and pin swapping).

INTEGRATES EASILY WITH OTHER SOFTWARE
* Cross-probe with OrCAD Simulate for Windows and OrCAD Layout for Windows.

* Pass information to and from OrCAD Layout for Windows, with full forward and back annotation.

* Copy and paste into word processors for documenting.

* Import data from external databases and spreadsheets to create reports and bills of materials.

* Output netlists for all popular PCB, simulation, and FPGA/CPLD tools.

* Easily import and export OrCAD SDT 386+ and SDT Release IV designs.
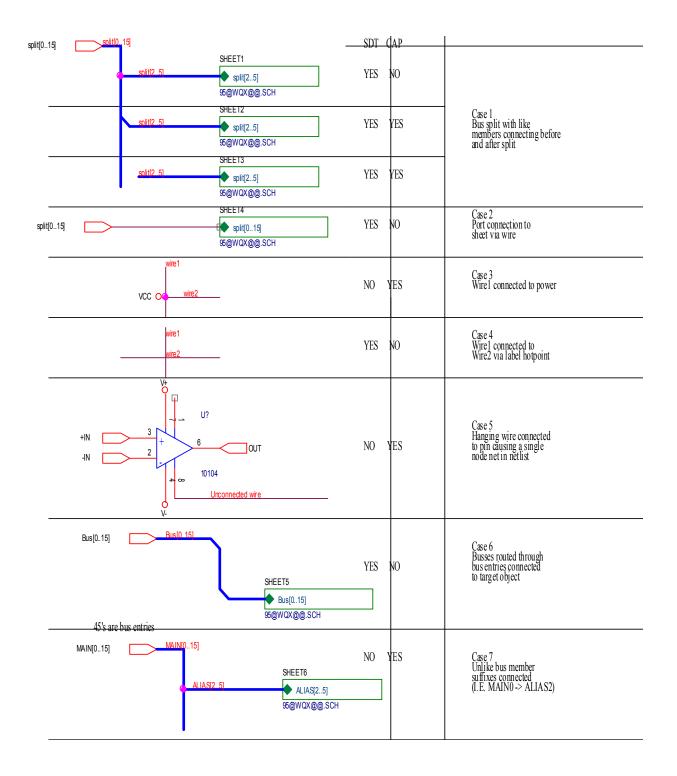
WINDOWS 95 COMPATIBLE
* Capture runs in Windows 95.

DIFFERENCES BETWEEN CAPTURE AND SDT 386+/ SDT Release IV

During translation, Capture transforms SDT IV and SDT 386+ objects into equivalent Capture objects. In most cases, there is a direct mapping of objects, but there are some cases where SDT's behavior could not be replicated in Capture. The most common of these cases are listed below.

CONNECTIVITY DIFFERENCES
The connectivity differences between Capture and SDT are described in the diagram below. The SDT column indicates whether a connection exists in SDT, while the CAP column indicates whether a connection exists in Capture.

	SDT	CAP	
split[0..15] ... split[0..15]			
SHEET1 split[2..5] / split[2..5] / 95@WQX@@.SCH	YES	NO	
SHEET2 split[2..5] / split[2..5] / 95@WQX@@.SCH	YES	YES	Case 1 Bus split with like members connecting before and after split
SHEET3 split[2..5] / split[2..5] / 95@WQX@@.SCH	YES	YES	
SHEET4 split[0..15] / split[0..15] / 95@WQX@@.SCH	YES	NO	Case 2 Port connection to sheet via wire
wire1 / VCC O wire2	NO	YES	Case 3 Wire1 connected to power
wire1 / wire2	YES	NO	Case 4 Wire1 connected to Wire2 via label hotpoint
U? (+IN 3, -IN 2, OUT 6, V+, V-, 10104) Unconnected wire	NO	YES	Case 5 Hanging wire connected to pin causing a single node net in netlist
Bus[0..15] ... SHEET5 Bus[0..15] / 95@WQX@@.SCH 45's are bus entries	YES	NO	Case 6 Busses routed through bus entries connected to target object
MAIN[0..15] ... ALIAS[2..5] SHEET6 ALIAS[2..5] / 95@WQX@@.SCH	NO	YES	Case 7 Unlike bus member suffixes connected (I.E. MAIN0 -> ALIAS2)

1. In Capture, when two names are placed on a contiguous bus, the name with the largest range becomes the name of the bus. In order to rip a subrange from the bus, a bus entry must be used.

Note: Buses must be labeled in order to connect members even if the bus is being passed between two hierarchical ports (sheet nets).

2. Wires cannot be used to connect to a bused port on a hierarchical block (sheet symbol); a bus must be used. Case 1 is bus splitting. There are three sub-cases: direct, via bus entry, and implied.

3. If a contiguous wire runs over power, ground, pin, or port endpoints, Capture will connect.

4. Net aliases do not have "hot points" in Capture. Therefore, an alias will not create a connection in case 4 as SDT would.

5. If a wire is connected to a pin, but does not connect to any other pin, a single node net is generated in the netlist.

6. In Capture, a bus entry requires bus segments on either side to be labeled in order to produce a correct netlist. Bus entries used to create 45 degree corners on buses are replaced with bus segments during translation to avoid this problem. Tip: it's easy to draw bus segments at 45 degree angles: simply hold down the shift key while drawing the bus.

7. In Capture, signals are mapped between connected buses of unlike base names by the index of the signal: MSB to MSB. If the basenames are the same, signals are mapped to like named signals.

LIBRARY PART DIFFERENCES
1. On library parts where the CONVERT image uses a different pinout than the NORMAL image, the CONVERT image is not translated. References to the CONVERT images of these parts are treated as missing parts.

2. Irregular fills are not translated. Only fills contained within a regular closed shape are translated. This includes fills enclosed in a rectangle, circle or polygon.

TEXT DIFFERENCES
1. Text is translated for each type of object using the fonts supplied in the design template. See the help file for more details. To change globally, change the font on the Design Properties dialog box (accessed via the Design Properties command on the Options menu in the Design Manager).

2. Extended ASCII text is rendered differently based on the font selected.

QUESTIONS?

If you have any questions about Capture or would like more product information, please contact **OrCAD DIRECT** at **503-671-9505** or send email to **info@orcad.com**. If you reside outside of North America, please contact the OrCAD Reseller in your region or send email to **intl@orcad.com**.

Thank you for considering OrCAD Design Desktop for Windows products.

Bringing Electronic Design to the Desktop