Cadence Design Systems, Inc.



The Orcad-to-Mentor Graphics Interface

revision 2.0—March, 2001





13221 SW 68th Parkway, Suite 200 Portland, OR 97223 • USA

Orcad DIRECT	(800) 671-9505
Corporate offices	(503) 671-9500
Fax	(503) 671-9501
Technical support	(503) 671-9400
Bulletin board system	(503) 671-9401
Email	info@orcad.com
World wide web	http.//www.orcad.com

Copyright © 2001 Cadence Design Systems, Inc. All rights reserved.

Orcad® and Orcad Capture® are registered trademarks.

BoardStation, LAYOUT, LI BRARI AN, FABLI NK and other names of Mentor Graphics products referenced herein are trademarks or registered trademarks of Mentor Graphics Corporation.

All other brand and product names mentioned herein are used for identification purposes only, and are trademarks or registered trademarks of their respective holders.

OrCAD Capture Software Version: Release 9.2.2

Revision history

5/12/99	First issue
7/30/99	revision 1.0, Added backannotation
3/31/01	revision 2.0, Consolidation, editing and reformatting; added Capture dialog box examples

Contents

Introduction4
Installing and the Orcad-Mentor Graphics interface 5
Installation of the vendor DLL for Mentor Graphics5
The Mentor Graphics configuration file5
Notes 7
Design data flow8
Creating the Mentor Graphics netlist in Orcad Capture10
Converting Orcad output files to BoardStation- compatible files11
Completing a board design using Mentor Graphics Librarian13
Notes13
Back annotating design changes14
Setting up for back annotation in the Mentor Graphics environment14
Completing back annotation in Orcad Capture15
Notes

Introduction

This document describes the interface for using Capture, the Orcad schematic capture program, to create input for BoardStation, the PCB design tool sold by Mentor Graphics.

This document addresses the following topics:

- Installing the Orcad-Mentor Graphics interface.
- Design data flow within an Orcad-Mentor Graphics environment.
- Creating a Mentor Graphics netlist from within Orcad Capture.
- Creating a BoardStation PCB database in the Mentor Graphics environment.
- Laying out a board in BoardStation and generating back annotation information.
- Back annotating design changes from BoardStation to Capture.

Installing and the Orcad-Mentor Graphics interface

To implement the Orcad-Mentor Graphics interface within Capture, you must first do the following:

- 1 Install the MENTOR.DLL supplied by Orcad to the Vendor subdirectory of the directory where CAPTURE.EXE is located.
- 2 Ensure that the MENTINT.INI configuration file is copied to the directory where the Capture output files will be created. This is typically the directory in which your Capture .DSN file is stored, or your "design directory."
- **3** Edit the configuration file, changing the names of standard properties and adding optional property names as needed.

Installation of the vendor DLL for Mentor Graphics

During installation of the Capture software, the MENTOR.DLL is automatically placed in the VENDOR subdirectory of the Capture installation directory. This is the directory where the CAPTURE.EXE file is located. A typical path might be:

C:\Program Files\PSD_14.1|\Capture\Vendor\Mentor.dll

If you find that the MENTOR.DLL file is not located in the same place as CAPTURE.EXE, then you must copy it to the VENDOR subdirectory of the Capture installation directory. You should already have a VENDOR subdirectory if you are using other third-party or Orcad-supplied accessories. If not, Capture creates a VENDOR subdirectory during installation.

Note

This DLL requires Version 9.10.131 of Capture, or later, to work properly.

The Mentor Graphics configuration file

The purpose of the configuration file is to map the properties that get transferred between the Orcad and Mentor Graphics tools during netlisting and back annotation. Mentor Graphics uses properties to control component placement and net routing within BoardStation. Orcad uses the same approach. However, the names of corresponding properties differ between the two tools.

To properly configure Orcad to output Mentor Graphics files with appropriate Mentor Graphics property names, a configuration file named MENTINT.INI must be configured and placed into the same directory (path) as the Capture output files. This directory is typically the location where your Capture .DSN file resides. The MENTINT.INI file is an old Windows-style "INI" file as shown in the configuration file example on the next page. This file is also used for back annotation, described later.

The MENTINT.INI file has four sections:

• The [standardcompparams] section gives the names of the properties used for the seven standard fields in the COMPS file. The defaults for the property names are shown in the example below. If any of the parameters are omitted from this section of the configuration file, the Mentor Graphics default property name is be used.

The next three sections of the file give the names of properties to be output only if present on a part or net. There are no defaults for these property names. Each property name must be followed by an equals sign (=) and a value, such as yes.

- The [optionalcompparams] section lists the names of the properties to be output to the COMPS file for each part, if present. In the example, if a part has a ROOM property, if will be output in the form (ROOM, ROOMvalue) following the standard fields for that part.
- The [optionalgateparams] section lists the names of the properties to be output to the GATES file for each gate, if present. In Mentor Graphics terms, each gate is an instance, but in Capture terms, each line more nearly corresponds to an occurrence. Occurrence properties, if present, override instance properties. For example, if a part has a TECH property, it will be output in the form (TECH,TECHvalue), following the standard fields for that gate, and an occurrence TECH property value will be used instead of an instance value.
- The [optionalnetparams] section lists the names of the properties to be output to the NETS file for each net, if present. In the following example, if a net has a SOURCE property, the property will be output in the form (SOURCE,SOURCEVALUE) after the pins of the net.

Sample Mentor Graphics configuration file

```
[standardcompparams]
type=Value
partnum=partnumber
footprint=PCB FootPrint
locationX=X
locationY=Y
rotation=pcbrotation
surface=surface
[optionalcompparams]
room=yes
. .
[optionalgateparams]
TECH=yes
[optionalnetparams]
source=yes
restrict=yes
```

Notes

- 1 Property names in Capture 9.0 are case-sensitive, but in 9.1 they are not. This means that in Capture 9.0, the property name on the part or net *must* match exactly the property name in the configuration file. For example, in 9.0, if the property name in the configuration file is room, the property name on the part must also be room.
 - In 9.1, the property name on the part could be Room, ROOM, or roOm, and it would still be output if the configuration file specified as room. For 9.1 and later versions of Capture, the property name is written to the output file as it is shown in the configuration file. Check the Mentor Graphics documentation to determine whether your version of BoardStation has case-sensitivity requirements for property naming.
- 2 Do not use rotation for the name of the rotation property in the [standardcompparams] section of the configuration file. This property name is already defined by Capture to specify the rotation of the schematic symbol, which bears no relation to the rotational placement of the part on a PCB.

Design data flow

The interface between Capture and BoardStation is implemented in the MENTOR.DLL, a vendor DLL located in the VENDOR subdirectory of the directory where the CAPTURE.EXE file is installed. MENTOR.DLL implements the file translation functions of netlist creation and back annotation between the Orcad Capture and Mentor Graphics BoardStation environments.

The output files from Capture must be converted into BoardStation design objects and copied to the proper location. Back annotation is accomplished by using Mentor Graphics' NEUTRAL and WASIS files. These files are generated by BoardStation and then transferred to Capture and be read into the schematic.

The basic design flow is described as follows:

- **1** Create a schematic using the Orcad Capture software.
- **2** From within Capture, add Mentor Graphics BoardStation LAYOUT properties to the schematic.
- **3** From within Capture's project manager, generate output files for Mentor Graphics using the Create Mentor Netlist dialog box. These files list information about the components, gates, and nets in the schematic.
- **4** Transfer the output files to the Mentor Graphics BoardStation workstation.
- **5** Convert the output files into Mentor Graphics design objects for use by BoardStation.
- **6** Create a Mentor Graphics LAYOUT database using the Mentor Graphics LI BRARI AN tool.
- 7 Design a PCB board using the Mentor Graphics LAYOUT PCB tool.
- 8 Back annotate design changes made in BoardStation—including pin and gate swaps and property assignments—to the schematic by creating a NEUTRAL and WASIS file from within the Mentor Graphics LAYOUT program.
- **9** Merge changes back into the schematic using Orcad-Mentor Graphics interface from within Capture's project manager.

This design flow is illustrated by the diagram on the next page.

Orcad to Mentor Graphics design flow

① Create a **KEY** Merge changes schematic using from Layout within **Orcad Capture** the Orcad Capture Capture's project design environment program. manager. **Mentor Graphics** design environment 2 Add Mentor **Graphics BoardStation** LAYOUT properties to the Orcad 8 Back annotate schematic. design changes to the schematic by creating **NEUTRAL** and WASIS files. 3 From within Capture's project manager, generate output files for Mentor Graphics. Design a printed circuit board using the MGC LAYOUT **4** Transfer the PCB tool. output files to a Mentor Graphics BoardStation. **⑤** Convert the **6** Create the output files into Mentor Graphics Mentor Graphics LAYOUT database design objects used using the MGC

by BoardStation.

LI BRARI AN tool.

Creating the Mentor Graphics netlist in Orcad Capture

To netlist the Capture design, you invoke the Orcad-Mentor Graphics interface from within Orcad Capture's project manager. Once in the project manager you select the Orcad design (.DSN) object and choose one of these **Accessories** pull-down menu options:

Accessories > Mentor > Mentor Netlist

Accessories > Mentor > Mentor Netlist (To UNIX)

The **Mentor Netlist** choice outputs DOS-compatible files. The **Mentor Netlist (To UNIX)** choice outputs files with line feeds only, and no carriage returns. This makes these files directly compatible with UNIX file systems and means that these files don't need to be converted to UNIX format using DOS-UNIX copy utilities.



When you invoke the interface, the Create Mentor Netlist dialog box appears prompting for the name of a COMPS file. In the dialog box above, the COMPS file is named TEST_DESI GN.TXT. After typing a user-name for the COMPS file and clicking the Save button, the netlister generates four files: NETS.TXT, GATES.TXT, PKGI NST.TXT, and your user-named COMPS file. The PKGI NST.TXT is a copy of the GATES.TXT file. All of these are ASCII text files.

These four files contain the netlisting information from Capture that Mentor's BoardStation needs. Unless you designate an alternate directory, these files are all saved in the directory where your Capture .DSN project is located. In the dialog box example, above, the four files would be saved in the MY_DESIGN directory.

After you have successfully generated the four netlist files, you must transfer them to the workstation where the Mentor Graphics BoardStation software is installed.

Converting Orcad output files to BoardStation-compatible files

Mentor Graphics requires that any data imported from an ASCII file be converted into versioned Mentor Graphics design objects for processing by LIBRARIAN.

This process consists of several key tasks:

- 1 Transferring the Capture netlist files to a Mentor Graphics workstation.
- **2** Creating a special directory required by Mentor Graphics, named PCB, for the Mentor Graphics design objects.
- **3** Converting the Capture output files into Mentor Graphics design objects.

Once the data is transferred to the Mentor Graphics workstation (step 4 in the design flow diagram), the files must be placed in a PCB design object directory for LI BRARI AN to properly read them. You can easily create such a directory by executing the **mkdir** command within the UNI X environment. An example would be:

/bin/mkdir /users/orcad/design/pcb

The files must then be converted into Mentor Graphics design objects. Mentor Graphics provides a utility for this conversion task named **PCB_DESIGN_DATA_PATH**. Either though a script or at the command line, you can execute the **PCB_DESIGN_DATA_PATH** utility on each of the files generated by the Orcad-Mentor Graphics interface.

A sample script that both creates the PCB directory and converts the ASCII netlist files for use in BoardStation is listed on the next page.

Sample Make PCB Script

You can set up Capture data on the Mentor Graphics workstation manually, or you can write a script to automate the process. The script below illustrates the commands required to convert Orcad data into Mentor Graphics BoardStation format. Of course, this script would need to be adapted to reflect the differences between UNIX shells and Windows environment. Also, in the script you must substitute your design's COMPS file name for the file <USER-NAMED_COMPS_FILE>.TXT.

```
#!/bin/sh
# create pcb directory for design objects under design directory
/bin/mkdir /users/orcad/design/pcb
# convert output files from orcad into mgc design objects
$Mentor Graphics HOME/bin/pcb design data path -objtype comps
     -writefrom users/orcad/design/<user-named COMPS -file>.txt
     -design /users/orcad/design
$Mentor Graphics_HOME/bin/pcb_design_data_path -objtype gates
     -writefrom /users/orcad/design/gates.txt
     -design /users/orcad/design
$Mentor Graphics_HOME/bin/pcb_design_data_path -objtype pkginst
     -writefrom /users/orcad/design/pkginst.txt
     -design /users/orcad/design
$Mentor Graphics_HOME/bin/pcb_design_data_path -objtype nets
     -writefrom /users/orcad/design/nets.txt
     -design /users/orcad/design
 the end
```

Completing a board design using Mentor Graphics Librarian

After setting up the four input files, converting them into Mentor Graphics design objects, and placing them into the PCB design object, the database then needs to have appropriate PCB footprints or geometries loaded so the design will open in LAYOUT. To load these values, you invoke LI BRARI AN on a design database from the command line as follows:

\$Mentor Graphics_HOME/bin/librarian /users/orcad/design

This invokes LI BRARI AN for processing of the design, reading in the required geometries. To read in the required geometries, choose the **Geometries > Resolve Geometries** menu option. This displays the Resolve Geometries dialog box.

Once the all required geometries have been read into the design, you can proceed with configuring the design as normal. This includes:

- Reading in the board outline
- Configuring the technology file and layer stackup
- Reading into your design any additional geometries, vias, logos, and so on.

The final step is to save the design using the **File > Save** menu. This creates a complete, versioned database that can be opened in Mentor Graphics LAYOUT.

Notes

The process to bring the schematic changes forward to Mentor Graphics BoardStation is the same as it is for starting a new design. When you run the **PCB_DESIGN_DATA_PATH** utility on existing design data, the data is automatically revised and the appropriate files are versioned. Unless changes have been made requiring additional footprints, the resolve geometries process in LI BRARI AN can be skipped and the design can be opened directly in LAYOUT.

Back annotating design changes

Back annotating consists of two steps:

- 1 Creating the back annotation files, the NEUTRALS and WASIS files, using the Mentor Graphics LAYOUT tool.
- **2** Reading the NEUTRALS and WASIS files into the schematic through the Orcad-Mentor Graphics interface.

Setting up for back annotation in the Mentor Graphics environment

The WASIS and NEUTRAL files are created by menu picks within BoardStation. The files are then placed in a directory on a PC system that runs Capture.

The Mentor Graphics back annotation routine requires a PKGI NST file for proper generation of the WASIS file. The PKGI NST file is simply a copy of the GATES file written out by Capture. The back annotation program within LAYOUT generates a PKGI NST file which is used for creation of the WASIS file. Basically, the Mentor Graphics program compares the contents of the current GATES file to that of the PKGI NST file to determine what changes have occurred to the design. Those changes are then written out to the WASIS file.

In the Mentor Was-Is dialog box, you specify the location of the WASIS.TXT file generated on the Mentor Graphics LAYOUT workstation. In the Mentor Neutral File dialog box, you specify the location of the NEUTRAL_FILE.TXT file generated on the Mentor Graphics LAYOUT workstation. The system processes the resulting properties data and updates the Capture schematic to reflect the latest properties values.

The NEUTRALS file can be created in either LAYOUT or FABLINK by choosing the **File > Create > Neutral File** menu item.

After issuing this command, you are prompted to specify a pathname for the file to be written to using the name NEUTRAL_FILE.TXT. You specify no board offset and include these data types of components: *data* and *nets data*. This action generates an ASCII file containing the neutrals information which needs to be transferred back to the Orcad workstation.

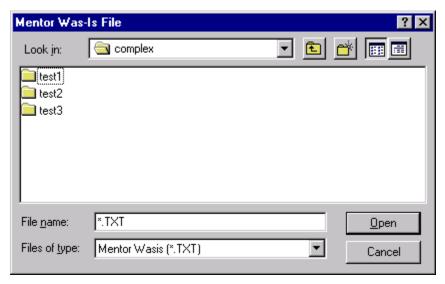
The WASIS file can be generated from LAYOUT by choosing the **Reports > Placement Swap Results** menu item. You select the To Pathname switch and specify that the data be written to the file WASIS.TXT.

After transferring the NEUTRALS and WASIS files back to the Orcad design directory, the files must be read into the Capture schematic through the Orcad-Mentor Graphics interface.

Completing back annotation in Orcad Capture

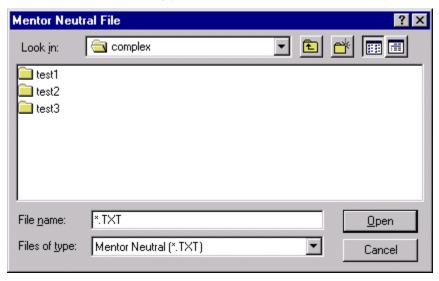
The back annotation function in Capture reads the WASIS and NEUTRAL files created by BoardStation. During back annotation, the Capture design is updated with gate, pin, and package swap information, as well as updates to net and part properties added or changed in BoardStation.

To back annotate, you load the subject design into Capture, select the design (.DSN object) in the project manager, then choose the **Accessories > Mentor > Mentor Back Annotation** menu item. A dialog box appears, prompting you for the location of the WASIS file.



There is no UNI X-vs-non-UNI X choice in back annotation, since either UNI X- or DOS-compatible files can be read by Capture's back annotation function.

After specifying the file name and clicking Open, the Mentor Neutral File dialog box will appears, prompting you for the location of the NEUTRAL file.



Both these files should reside in the same directory. A MENTI NT.I NI configuration file, if used, must be in the same directory as the NEUTRAL file.

Once you have run a back annotation, be sure to save your design.

Notes

- 1 When you finish the design or need to make changes to the schematic, make sure to back annotate the latest property and swapping information from the PCB to the schematic *before* performing your changes. Neglecting to completely back annotate changes to the schematic before changing the schematic will likely result in a loss of design data at the PCB level.
- 2 A backup copy of the design should be made before running the back annotation function. This way, if there are errors in the WASIS or NEUTRAL files, or if the Capture design has gotten out of sync with the BoardStation version, the original Capture design won't be damaged.
- 3 Back annotation updates the memory copy of the design, not the design on disk. For this reason, if you open a page of the design, and then close the page, you will be prompted to save the page. If you do not save the page, the back annotation changes on that page are discarded.
- 4 The MENTINT.INI configuration file will normally be the same MENTINT.INI file used in the Capture-to-BoardStation (netlisting) phase, unless new user properties were applied to nets or parts while running BoardStation. If this is the case, the property names must be added to the [optionalcompparams] or [optionalnetparams] sections so they will be back annotated to the design. The updated MENTINT.INI file should be used the next time the Capture-to-BoardStation (netlisting) function is run so the new properties can be re-sent to BoardStation.