



Expedition Flow 2005 SPac1 Release Highlights

Software Version 2005 SPac1

June 2006

© 2006 Mentor Graphics Corporation
All rights reserved.

This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND 03/97

U.S. Government Restricted Rights. The SOFTWARE and documentation have been developed entirely at private expense and are commercial computer software provided with restricted rights. Use, duplication or disclosure by the U.S. Government or a U.S. Government subcontractor is subject to the restrictions set forth in the license agreement provided with the software pursuant to DFARS 227.7202-3(a) or as set forth in subparagraph (c)(1) and (2) of the Commercial Computer Software - Restricted Rights clause at FAR 52.227-19, as applicable.

Contractor/manufacturer is:

Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

Telephone: 503.685.7000

Toll-Free Telephone: 800.592.2210

Website: www.mentor.com

SupportNet: www.mentor.com/supportnet

Contact Your Technical Writer: www.mentor.com/supportnet/documentation/reply_form.cfm

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other third parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the respective third-party owner. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/terms_conditions/trademarks.cfm.

End-User License Agreement: You can print a copy of the End-User License Agreement from: www.mentor.com/terms_conditions/enduser.cfm.

Introduction

This document provides a high-level summary of the Expedition Flow 2005 SPac1 (EXP2005 SPac1) release. It includes a summary of the new features in this release, any authorization code changes required, any major installation changes, and any transitioning issues you should be aware of before installing. Additionally, any last minute issues found in the final stages of testing are included.

This document is located on the top level of the CD and on SupportNet. Changes may be added to this document after the release. Refer to the Release Highlights documentation on SupportNet for the most up-to-date release information.

New Features for 2005 SPac1

Numerous customer-reported defect reports (DRs) were fixed in this release. Additional information on these DRs can be found in the EXP2005 SPac1 Release Notes on SupportNet.

The following new features were also added:

Constraint Editor System (CES)

The following is a list of new features in CES:

- **Constraint Groups.** Constraint groups you to view a specific subset of the constraint in the *nets, parts etc* sheets in CES. An example of a pre-defined constraint group would be *Net Properties, Simulated Delays* or *Delays and Lengths*. The aim of these constraint groups is to allow you to work with a group of data rather than manage the complete rules spreadsheet. In the 2005.1 release, the designer has the ability to create their own constraint groups. From the *Constraint Group* list box, the designer will find the new option; *...Edit User Groups*. The designer can then create their own groups with whichever constraints they wish. These new user groups will be added to the list box in alphabetical order.
- **Support For Embedded Passives.** CES 2005.1 introduces the ability to enter clearance rules for *Embedded Passives* (EP). The designer is able to enter the clearance rules between both the resistor and mask objects to the other copper features in the design. The EP rules can be found in the clearance sheet in CES.
- **Rules for SMD Pads.** The designer can now add clearance rules specifically the *Surface Mount Device* (SMD) pads in a design to both traces and vias.
- **Dialog Box Enhancements.** There have been many enhancements to the CES dialog boxes for the 2005.1 release. All the dialog boxes listed below have been enhanced to allow you to re-

size them. There have been other improvements to a number of the dialogs to make them more intuitive. The following is a list of dialog boxes affected

- **Variable list.**
 - Made expandable.
- **IBIS library search path.**
 - Made expandable.
- **Auto assign differential pairs.**
 - Made expandable.
- **Edit → Design Preferences Dialog... → Power and ground.**
 - Made expandable
 - Removal of index column.
 - Removal of description column.
- **Assign Physical Nets to Net Class.**
 - Made expandable.
 - Set focus to wildcard field
 - Set default action to Find (binoculars) instead of OK
- **Assign Physical Nets to Constraint Class.**
 - Made expandable.
 - Set focus to wildcard field
 - Set default action to Find (binoculars) instead of OK
- **Package Type Clearance Rules.**
 - Expandable dialog
 - Eliminate index columns
 - Expand Package Type columns
- **Create Constraint Template.**
 - Expandable dialog
 - Description is now multi-line
- **Select Constraint Template.**
 - Expandable dialog

-
- Remove Purpose column
 - Add a multi-line display for Description field
 - **Constraint Violations.**
 - Expandable dialog
 - Use a list view instead of tree view
 - Add Next and Previous buttons
 - Violation summary
 - **Setup → Options... → Column Fonts and Colors**
 - Expandable dialog
 - New layout of existing option - Swapped axis of columns and rows
 - **Dynamic Column Width.** The cells/column width have been enhanced so that as the designer types the column will grow to optimum size so that the data remain readable. This is especially helpful with the *Formula* column.

Expedition

- **CMC (Copy Move Circuit)**
- **MVOs (Multiple Via Objects)**
- **EP (Embedded Passives)**
- **Tuning on one side of a diff pair**
- **Route Fences**
- **Conditional splitting**
- **Local Gloss**
- **Tuning Iterations**

FabLinkXE

Following are the new features in FabLinkXE 2005.1:

- **Silkscreen Generator** – In previous releases the Silkscreen Generator was only available within a PCB database through Expedition PCB. With the 2005 release, FabLinkXE is now able to support both a Panel and PCB database in a standalone mode. The Silkscreen

Generator has now been added to the standalone mode of FabLinkXE and is available for both a Panel and PCB databases.

- **PDF Enhancements** – There are several PDF Output related enhancements:
 - **Mirrored output** – If the design database is mirrored, through Display Control, the PDF Output file will also be mirrored.
 - **Additional Paper sizes** – European sizes have now been added.
 - **Black and White only option** – By default, the colors used in the design’s Display Control schemes are used in the PDF Output files. For plotting and printing to paper, the “Black and White Only” option alleviates the need to apply a different Display Control scheme within the design.
 - **Individual Detail Views** – In previous releases, PDF Output only supported “all or nothing” printing and plotting of Detail Views. There is now an option to select individual Detail Views for each PDF sheet.
- **Interactive update of board locations** - In previous releases of FabLinkXE, if the network location of a PCB database that was placed in the Panel design was changed, the next time the Panel was opened, FabLinkXE was unable to resolve the board instance(s) within the Panel. The only way to resolve this was to delete the board instance(s), Browse to the new network location and replace those boards within the Panel design. Now, on invocation, can simply Browse to the new location and the pathnames will be resolved interactively without having to delete and replace the board instances.
- **Update all design data** - In previous releases of FabLinkXE, the only time that the board instance would be updated in a Panel was on invocation. This has now been enhanced as follows:
 - **On invocation**
 - **On demand**
 - **On generating manufacturing outputs**

The above options now ensure that the board instance(s) data is always in sync or up to date. Rather than having to exit and re-invoke FabLinkXE and open the Panel design, now you are able to verify and accept any board level instance changes at any time in the Panel design process. This also facilitates a concurrent Board and Panel design process.

- **Dimensioning Enhancements** – Dimensioning has now been enhanced to support board level objects in a Panel design. Previously you were unable to dimension say between a board and panel level object, or board to board level objects within a Panel. This enhancement continues to support the associations between the board and panel level objects.
- **ODB++ Output** – The two stage, ODBG plus ODBG>ODB++ post processor has now been replaced with a direct ODB++ output. This removes the dependency on a 3rd party translator and provides additional output capabilities that are supported by the Valor tools, such as “Full”, “Fab” and “Assy” modes.

-
- **Automation Pro** – Automation Pro support has been added for the following “batch engines”:
 - **Gerber Output**
 - **NC Drill Output**
 - **DRC**
 - **PDF Output**
 - **Manufacturing Output Validation**
 - **Design for Fabrication (DFF)**

This will enable the batch engines listed above to be run external from the application, at the command level and through the use of automation scripts.

- **Embedded Component Support (FabLinkXE Pro only)**– The 2005.1 release of Expedition PCB will deliver functionality for the design of Embedded Components. In order to fully support the manufacturing requirements, the following outputs and new functionality has been added to FabLinkXE Pro
 - **GDSII Output**
 - **IPC-D-356-B Netlist Output** – This is currently the only industry “standard” netlist that offers support for embedded component technology.
 - **Mask Generator** – The Mask Generator provides the ability to combine graphical database elements, using logical or Boolean expressions to create specific manufacturing data sets, such as AND, OR, XOR, add/subtract shapes (touching, crossing, enclosed by etc.) The resultant data may be written to a user-defined layer then manufacturing outputs generated from that data. Some examples of use of the Mask Generator include extracting different embedded component material on specific layers, creating di-electric openings of a particular size and extracting all soldermask pads and undersizing/oversizing for a manufacturing process and/or tolerance change, rather than having to modifying the library padstacks.
- **Design for Fabrication (DFF) Analysis Improvements** - There are several DFF Analysis related enhancements:
 - **Rule Areas and Window support** – Much like Net Class and Clearance rules, DFF schemes may now be applied to Rule Areas. DRC windows may also be applied to DFF Analysis, rather than having to perform the analysis on the whole design.
 - **Use defined limit for maximum number of Hazards** – In previous releases, there was a hard coded 10K limit on the maximum number of Hazards. This meant that if, for example an incorrect rule set had been applied that resulted in bogus errors, the 10K limit would have to be reached before the analysis would stop. Now you may apply a different limit for each analysis operation, ensuring that a reasonable

limit may be set and not have to wait for an excessive number of hazards to be reached before completion.

- **Browse option for rule schemes** – In previous releases, the DFF rule schemes could only be saved and/or retrieved either locally, with the design, or with the “system files, in the install directories. Now you may browse to any network location to both, save new schemes and retrieve existing schemes.
- **Blank fields for Rules and Clearances** – When entering values for each of the rules, previously you had to enter a value for all three fields: Severe, Moderate and Warning. Now you may just add the value for one or two fields rather than all three, significantly simplifying and the time taken for rule entry.
- **Review Hazards summary count** – In Hazards Review when using the Hide/Visible options, a summary count will now be provided so that you can easily see the number of hazards of a particular analysis/rule that are hidden versus the total number for that analysis/rule.

HyperLynx 7.5

HyperLynx did not change for this release.

I/O Designer

The following are the new features for I/O Designer:

- **Synchronization Wizard.** I/O Designer 2005.1 introduces a new mechanism to synchronize all data related to the FPGA that the I/O Designer database is opened for. The new Synchronization wizard that replaces the old Update wizard works both ways (import/export). It warns the designer when the design is out of date and suggests the next logical step necessary in the process to synchronize the entire design.
- **New Hierarchical reserved sheets mechanism.** I/O Designer 2005.1 has a changed methodology to update the hierarchical schematics. Instead of trying to update the hierarchical schematics with a lot of limitations, it now creates reserved sheets that are only updated by IOD and in addition the designer can add their own hierarchical sheets with components that are not touched by IOD anymore.
- **Automatic instantiation of page borders.** New functionality in I/O Designer 2005.1 enables the designer to specify a page border to be automatically added to the generated schematics in the DxDesigner flow.
- **HDL and PCB signal names and mapping.** I/O Designer 2005.1 introduces the ability to use different signal name conventions for the schematic/pcb design and the HDL design. Each signal or bus can have a different name for the FPGA and schematic/PCB design. Existing FPGA's and PCB's can be mapped to each other. This new feature also introduces the ability

to have PCB only signals, and therefore connect configuration pins or power pins to them without the necessity to have the same signal in the HDL

- **Assignment enhancements.** New features are added to assign busses pin by pin in an enhanced way or swap large groups of pins such as busses, banks or entire swapgroups.
- **Local Clocks and Clock regions.** Beside the already known I/O Bank view in the device window, new views have been introduced to show local clocks or clock regions.
- **Lattice support.** I/O Designer 2005.1 supports the latest available versions of all FPGA vendor tools such as Actel Designer 7.0, Altera Quartus II 5.1 and Xilinx ISE 8.1. New in 2005.1 is support for Lattice (ispLever5.1). In addition to new library support the library sizes have been reduced also due to a new more compact format.
- **Timing constraints and CES support.** A first step has been made in I/O Designer 2005.1 to bring timing closure between the FPGA and PCB world. This version introduces a new timing tab to enable the designer to enter any timing constraints to be synchronized with the FPGA constraint files and timing report files. The new integration between I/O Designer and CES makes sure that any pin properties (e.g. differential pairs, I/O Standards) are being kept consistent between the FPGA and PCB design flows.
- **New windows management system.** Windows can be docked/undocked and free positioned inside or outside the main I/O Designer window as of version 2005.1. Also the ability to tab windows in order to save screen space has been introduced.

Quiet Expert

- **Additional API Set.** A new API, `qe_NetOverlap`, has been added to Quiet Expert for reporting vertical overlap between two nets. The API returns results as a list of polygons where the overlap condition occurs.
- **Net Classification.** Net grouping has been improved to allow better classification of nets. In addition to classifying nets by assigning them to net groups (Clock, Power, Ground, or Unclassified), you can now designate nets as having one or more of the following electrical characteristics:
 - o sensitivity to noise
 - o generating noise
 - o analog signal
 - o low-power signal

These net designations do not affect the net grouping, and a given net can be assigned more than one characteristic at a time.

-
- **Various DR Fixes.** There are numerous DR fixes to the Quiet Expert engine and translation process. Additional information on these DRs can be found in the Release Notes on SupportNet.

XtremePCB

The following are the new features for XtremePCB:

- Ability to turn off online DRC.
- Ability to manipulate Embedded Passives (license dependant).

New Features for 2005.05

The following new features are available in this release:

Constraint Editor System (CES)

CES has been extended to support electrical constraints, enhanced physical constraints, and improved design flow integration.

- **Electrical CES**
 - Define electrical constraints, such as simulated delays, simulated crosstalk, and overshoot and ringback.
 - Assign IBIS models, and construct electrical nets and identify differential pairs from IBIS model keywords.
 - Validate model assignments, looking for missing or mismatched models.
 - Use the IBIS Model Wizard to quickly create or edit complex IBIS models.
 - Generate trace model schematic in ICX Pro Explorer for solution space analysis and define constraint templates.
 - Define simulation settings and driver stimulus.
- **Enhanced Physical Constraints**
 - Minimum, maximum, and matched length or Time-of-flight rules directly on electrical nets with series devices.
 - Pin-set topologies for automatic virtual pin insertion, placement, and trace balancing.
 - Trace width and layer overrides for pin-to-pin segments.

-
- Maximum length on restricted layers specified by net or constraint class.
 - Package-type to Package-type clearances.
 - Noise rules (parallelism and crosstalk) can be specified for constraint classes.
 - Package Pin Length defined in spreadsheets and factored into length tuning/hazards calculations.
 - **Improved Design Flow integration**
 - Supports new DxDesigner-CDB-Expedition design flow.
 - Now supports XtremePCB.
 - Define constraints on Expedition layout templates.
 - Synchronize net class constraints without full annotation cycle.

DMS Librarian

The existing Library Manager application, that is used to manage Central Libraries in the Expedition flow and for the Design Capture/Expedition PCB tool combination, has been extended (via the DMS Libr. Mgmt. Add-on Ap SW PN 226377) to support a transparent integration to DMS. This extended version of Library Manager is accessible by selecting “DMS Librarian” from the Windows Start Menu or running “DMSLibrarian” from the command line. The major benefits of DMS Librarian are:

- DMS Librarian is an extension of Library Manager, which enables you to have the full benefit of DMS library management while using your accustomed library management environment. Managing libraries becomes easy, and you have the full value of data management without having to be concerned with the details.
- Easy, “next-step” migration to full part request process (see Librarian Flow Manager)
- While all Library object editing functionality has moved from dmsDesktop to DMS Librarian, DMS Librarian provides access to dmsDesktop for queries, component editing, etc.
- DMS Librarian includes a new tool called Update Production Library Wizard, which substantially simplifies the process of updating a production library in the Expedition 2005 SPac1 flow. While existing scripts and processes remain fully compatible with DMS2005 SPac1, you also gain the option of using a production library wizard:
 - Convenient interface assists you in selecting incremental or complete cache updates.
 - Status management helps automate behavior in terms of updating existing parts and synchronizing newly released ones.
 - By providing an automated wizard environment, the repeatability of cache updates is increased and the reliability of the process are greatly increased.

Note: Functionality similar to the Update Production Library Wizard was available in ISD2004 SPac2 in the Library Manager to DMS Wizard. The Library Manager to DMS Wizard has been replaced by functionality in the DMS Librarian extension to Library Manager and the Update Production Library Wizard. This is supported by the DMS Libr. Mgmt. Add-on Ap SW (PN 226377) license.

Documentation System Improvements

InfoHubs are HTML information centers that organize your sets of documentation and provide information links into SupportNet. InfoHubs provide a combination of HTML- and PDF-based user, reference, process, and release information. Most of the HTML content is also available as PDF manuals. InfoHubs also include a local HTML search based on keywords, and an enhanced natural-language search of SupportNet.

InfoHubs are replacing the existing PDF- and Help-based documentation systems. This release is transitional. Some of the DxDesigner documentation is available as both HTML and PDF, while most of the Expedition and other product documentation is in PDF, WinHelp, and HTMLhelp. All of the documentation is migrated to the new InfoHub documentation system in future releases.

You can access InfoHubs by loading the following file in your HTML browser:

```
<install_dir>/2005EXP/docs/infohubs/index.html
```

<install_dir> is the target directory where you install your Mentor Graphics application software. For more information, refer to the *Managing Mentor Graphics PCB Systems Software* manual available on the software release CD or via SupportNet.

Note: Changes to the default setting for Internet Explorer on Windows XP SP2 prevents the InfoHub from functioning. A fix for this issue is provided in the Additional Release Notes and Workarounds section later in this document.

DxDesigner

The following describes the new features for DxDesigner:

- **DMS Connector.** DMS provides seamless integration for the DxDataBook product. The following features enhance the DxDataBook capabilities in combination with a DMS Database:
 - Using the 'native' DxDesigner instantiation, update, and replace mechanisms leverage DMS content.
 - DMS component datasheets are available in DxDesigner.
 - Using <http://<server>/DMSCconnector> URL link in DxDataBook, you can do a sophisticated part research in DxDataBook. An upgrade option is available to utilize this functionality.

-
- **Expedition Integration.** Integration with Expedition layout has been significantly upgraded through the use of the Common Database (CDB) and CDB Compiler and Packager utilities (as used with Design Capture-Expedition) to generate, store and exchange design, connectivity and packaging data between DxDesigner and Expedition. The use of the CDB Compiler and Packaging utilities provides the DxDesigner-Expedition user with a much tighter level of integration between DxDesigner, Expedition and also with the Central Library since the packaging data is pulled directly from PDB (Parts Database).

Expedition Design Configurations have been enhanced with new properties to support new CDB flow integration. New Tool Bar Menus have been added to support the CDB flow integration. New Device selection from the PDB menu and an “Add Device” tool bar have been added. You can search and add the Central Library PDB parts using this new feature.

Note: To use the CDB integration, upgrade to the DxDesigner for Expedition or DxDesigner 100 package. Please contact your account manager or sales representative to learn how to upgrade to one of these packages.

For more detailed information about Expedition integration, refer to the *What’s New* document on SupportNet.

- **Library Manager.** Library Manager has been updated to support new features in the DxDesigner/Expedition flow. Refer to Library Manager later in this section.
- **Performance Improvements.** This release provides improved performance and stability of DxDesigner on all platforms. Some of the specific areas of improvement include:
 - Overall stability and memory usage by several applications.
 - DxDesigner startup time. A new option is available to suppress the loading of addins until the addin is needed. This allows DxDesigner to start quickly in situations where the Project Navigator is not needed. Refer to the DxDesigner documentation for more information.
 - Project Navigator help tab performance. The help tab no longer loads the contents of the individual help files, reducing the time it takes to display the top-level contents hierarchy.
 - Scout run time on large designs. There is now a “No Verbose” check box in Scout that sends all error messages to the scout.err file, rather than to the Dashboard log window.
- **Location Map Support.** Environment variables and/or location map entries can now be used as input to DxDesigner to resolve soft pathnames. A soft pathname contains a variable reference that points to a specific file system location. Soft pathnames can be used in .ini files (viewdraw.ini/draw.ini and files referenced by draw.ini) or some DxDesigner dialog boxes that require a pathname to a file or directory.
- **Find/Replace Enhancements.** Find/replace is an extension of the existing select/replace. A new feature of find/replace is SCOPE. There are four scopes available for use while finding the objects. Find/replace supports regular expressions and works across a design, through hierarchy, and on any schematic object.

-
- Page: Object is searched in the current sheet only.
 - Block: Object is searched in all the sheets of schematic to which the current sheet belongs.
 - Here Down: This is a hierarchical search/find. The object is searched in the current schematic and its underlying hierarchy.
 - Project: This enables searching in all the schematics of current project.
 - **Documentation Improvements** - The following improvements have been made to the DxDesigner documentation set.
 - All DxDesigner procedure and reference information and is now provided in both Adobe PDF and HTML formats. The HTML is optimized for online viewing with your browser, and the PDF is optimized for hardcopy printing. These new documents are designed to present information in a more complete and contextual format. They include the following major documents, in addition to many other supporting documents:
 - o *DxDesigner User's Guide*
 - o *DxDataBook User's Guide*
 - o *PCB Interfaces Guide*
 - o *DxDesigner Administrator's Guide*
 - o *Scout User's Guide*
 - o *DxDataManager User's Guide*
 - o *Developing Scripts for DxDesigner Applications*
 - o *Simulation Netlists*
 - o *Getting Started with DxDesigner*
 - o *Creating Variant Designs from DxDesigner*
 - o *DxDesigner Attributes Glossary*
 - o *DxDesigner DOS Utilities*
 - o *DxDesigner Schematic Translators*
 - o *DxParts QuickStart*
 - o *Getting Started in Hierarchical Schematic Blocks*
 - o *WIR Database Reference Manual*
 - o *Viewbase Reference Manual*
 - All product documentation is consolidated into a single help system, called the InfoHub, that is HTML-based and viewable from your browser. You access the InfoHub, and all the installed product documentation (either HTML or PDF), through a single Help menu

-
- selection from the Dashboard, from DxDesigner, or (in MicroSoft Windows operating systems) from the Start menu. A new search feature in the InfoHub enables you to search all locally installed documentation and Supportnet documentation. Because the InfoHub locates all documentation in one place, it is easier to locate the needed information.
- The “Help” tab in the Project Navigator Tree window has been optimized to display help data more quickly, and it will display the same information as is displayed from the Help menu in DxDesigner.
- **Schematic Design and Editing Features.** Several enhancements have been made to the schematic editing capabilities in DxDesigner. They include:
- A new bus contents file is now available for defining the signals that are included in a bus. A short label can be placed on a bus that represents several buses and nets. For example, a bus can be named “CONTROL”, and in the bus contents file, the bus can be defined as containing the signals “READ, WRITE, OE, CNTL [7:0]”. The short name CONTROL can then be used to label the bus everywhere it appears in the design. This makes it easy to label a complex bus bundle, and change the contents of the bus everywhere it exists by simply changing the definition in the bus contents file. In addition, existing bus contents files from other projects or libraries can be included in the current project, making it easier to import the bus definitions that have already been used. Conflicting bus names are resolved in the user interface.
 - Hierarchical block generation has been improved. A schematic popup menu selection is now available that invokes a dialog for creating a hierarchical symbol from the active schematic sheet. All signals that contain hierarchical pin connectors (such as IN, OUT, or BI) are automatically added as pins to the symbol.
 - Functional Blocks, or FUBs, can now be resized in-place on a schematic. Select the corner of the FUB and stretch it to make it larger. This makes top-down design more flexible, since space for new signals can be easily created.
 - It is now possible to do a regular expression text search-and-replace across an entire design, including text within hierarchical blocks. Any type of text, including attributes, labels, and standard text can be modified. With this new feature, it is easier to rename a net everywhere it exists in a design.
 - A new “Place Device” dialog box can be used to place a part directly from the Central Library Parts Database (PDB). This is only used in the DxDesigner/Expedition flow when all parts are defined in the Central Library PDB.
- **Enterprise Configuration Enhancements.** To better support distributed design environments, we have extended the way that libraries and preferences can be specified for DxDesigner. In DxDesigner, whenever a new project is created, a copy of the viewdraw.ini file is placed in the new project folder. This makes it difficult to update existing projects when library paths or preference data change.

In this release, the viewdraw.ini file can be replaced with a new file called draw.ini. The draw.ini file only contains paths to other files, which contain the contents of the viewdraw.ini file. For example, you can create a file called “libraries.ini” and “preferences.ini”. These files may be put on a central server. The draw.ini file can be used to point to these files.

If you need to change the libraries or preferences, you can change them on the server and load the changes when invoking DxDesigner. Also, environment variables are supported in the paths in the draw.ini file, and in the library paths in the existing viewdraw.ini file. An example of a section of a simple draw.ini file is as follows:

```
read $CORP_PREF/preferences.ini
read division.ini WDIR
libsr //server1/standardparts/libs.ini
libsrw ./proj_libs.ini
```

In this example, the DxDesigner preferences (such as grid size, text size, etc.) may be read from a file called preferences.ini that is located where the \$CORP_PREF environment variable value indicates. Additional local preferences may be defined by following the WDIR environment variable search path and loading the division.ini file. Finally, the libraries for this project may be defined in the file called libs.ini that is located on server1 under the standardparts folder. User-defined libraries may then be located in a writable file called proj_libs.ini located in the project directory.

It is also possible to define a mapping file that maps variable names, or soft path names, to full hard path names. This provides added flexibility for defining the location of libraries in the viewdraw.ini file or the draw.ini file. A variable, such as \$LIBRARY, can be used instead of a hard path. The mapping file, typically located on the network under the your control, can then define the actual path to the library.

This capability can also be used to make it easier for UNIX users and Windows users to work in the same DxDesigner project. This is a very flexible and powerful capability that makes DxDesigner easier to configure in a distributed, enterprise-wide environment.

- **Partslister.** The Partslister has been enhanced to support multiple output formats. These formats include:
 - HTML
 - Microsoft Excel
 - Delimited Text (computer readable format suitable for integration with information systems)
 - The current formatted text output

In this release, the Partslister no longer runs the DxDesigner packager. This allows a parts list to be generated on designs that are not complete, or on designs that do not have packaging data on the symbols. In the new CDB-based Expedition integration, the DxDesigner packager

is not used, so the Expedition packager should be run prior to running the Partslister in order to get complete component data.

- **Archive Utility Improvements.** The Archive utility is used to collect all the symbols that are referenced by a design and copy them to a new project folder, or compress them into a single file, so that the project can be easily moved or stored. The Archive command has a new user interface that allows you to select which files become part of the archive. It can also be executed from a command line, so it can more easily be incorporated into project management scripts.

An archive.ini file is supported. This file allows you to specify which files and designs should be added to the archive based on regular expression mapping. It can be used to provide consistent design archives that contain the required files and exclude the unnecessary files. The Archive utility also modifies the hard paths that exist in the Design Configuration (dproj) project file. Options are provided to generate a PDF view of the design, and to compress the archive into a single file.

- **Copy Schematic.** It is now possible to copy a section of a schematic from another project to your current schematic without adding the project to your library list. Using the file browser in Windows Explorer or Dashboard, select the schematic you want to open and double-click on it. When the schematic is open, select the circuitry you want to copy and copy it to the clipboard using the Edit > Copy menu command. You can then paste the circuitry into your current schematic using the Edit > Paste menu command. This new feature makes it easy to reuse schematic data from existing designs.
- **Defect Fixes.** A large number of high priority defects have been fixed in this release. An emphasis was placed on improving the performance and stability of DxDesigner. Please contact your customer support representative for status on specific issues.

Expedition

The following are the new features for Expedition:

- **HSR (High Speed Option) and Electrical Constraints**
 - Support Electrical net constraints (Series elements).
 - Netline Manipulation & delay/length recognizes series elements.
 - Pin-set Topologies for virtual pins in custom and partially-ordered nets.
 - Maximum Length on Restricted layers by Net and Class.
 - Layer restriction and trace width by pin-pairs.
 - Improved HSR Online Hazards reporting.
 - Improved performance for Formulas Online Hazards.

-
- Support for Flooded Signal layer in stackup.
 - **Flex Circuit Enhancements**
 - Can now slide mixed 45's and arc'd traces properly.
 - Multi-hug trace feature.
 - Group slide multiple hug traces.
 - Improved Assign Net Name feature.
 - **Advanced Draw Capability**
 - Cutouts in shapes for plane shapes, placement obstructs and draw objects.
 - Padstack Editor now has Draw Tool palette containing almost all Draw functions.
 - Sophisticated trim and extend command.
 - New Scale and Shape Offset commands to provide greater control over resizing draw objects.
 - Boolean draw operations - add/subtract shapes.
 - Improved Arc placement.
 - **AutoActive Enhancements**
 - Support for Package-Type to Package-Type clearances (CES constraints).
 - Support for HP and Sun Platforms (in addition to Windows and Linux).
 - Improved DxD schematic integration.
 - Improved Gloss on tuned diff-pairs.
 - Single plow and Multi-plow combined into single command - automatically recognize diff-pairs.
 - Advanced Plane Shape Support.
 - Batch DRC by Net.
 - Additional Plane to Trace, Pad and Via clearances.
 - Integration CES with Team PCB.

FablinkXE

Following are the new features in FablinkXE 2005:

- **PCB DFF Analysis** - Design For Fabrication (DFF) Analysis is a new product option that is tightly integrated with Expedition PCB to provide advanced fabrication and manufacturing rule DRC that reduces clean up cycles to and from the fabricators which improves quality, manufacturability and yield of PCB designs. It is targeted at those companies where controlling volume fabrication and manufacturing costs is critical to their business success. It puts DFF in the design space where it belongs and not as a post design process based on generated manufacturing data, DFF rule sets are defined and Hazards identified and resolved in the PCB database using the design data itself. It is available as an option to all configurations of Expedition PCB and Fablink XE.
- **PCB design file Support** - You can now open an Expedition PCB database directly in Fablink XE. Additional functionality such as Detail Views, searchable PDF output, Gerber In and Drill In can now all be accessed directly on an Expedition PCB database without consuming an Expedition PCB license. Previously Fablink XE functionality could only be accessed in a PCB database while acquiring an additional Fablink XE license.
- **IDF 3.0 Support** - Fablink XE now performs IDF 3.0 exports at the Panel Level. This enables integration between Fablink XE and Mechanical CAD tools.
- **Detail Views** - Fablink XE enables you to view detailed design content more clearly by employing an opaque “Bring to Front” style view instead of the traditional transparent view. This allows you to see exactly what they need to see without seeing all the other parts of the design. Individual Detail Views can now also be selected for plotting to a PDF file.
- **Panel Creation Improvements** - There are two significant areas where Fablink XE improves the panel creation process:
 1. The process of invoking Fablink XE from Expedition PCB for the first time when creating a panel design has been significantly improved and automated. All panel directories and associated data is created automatically. There is also an option to load a minimum data set (manufacturing Outline and Mounting Holes) when placing PCB's within a Panel design, this allows for the parallel preparation of the PCB design and the panel. Instead of having to wait for the PCB design to be complete before starting on the panel design, when the PCB design is complete, the full data set can be loaded just prior to generating the manufacturing output.
 2. Fablink XE now uses design layer validation to allow panel designs to ‘inherit’ the layer stackup of the first board placed on the panel. Previously, companies would have had to maintain a large library of panel templates that matched the layer stackup of all their PCB's. Now, Fablink XE simply matches the layer stackup of the panel to that of the first board.

HyperLynx 7.5

The following are the new features for HyperLynx 7.5:

- **Free-form schematic editor.** A significant upgrade to the LineSim® schematic editor, providing an optional way to create new or edit existing LineSim schematics. Full Windows support for Copy/Paste of multiple components and transmission-line segments—critical for complicated schematics.
- **Interactive board viewer.** The BoardSim® board viewer now supports point-and-click selection of nets and components, partial plane importing and viewing, interactive crosstalk coupling region viewing, and many more interactive enhancements.
- **Batch simulation.** A major upgrade for BoardSim users, including driver "round robin" support, automated flight time compensation, and sortable spreadsheet output.
- **Simulation enhancements.** Support for multiple power supplies, along with both die and pin probing.
- **HyperLynx File Editor.** All-new multi-purpose text editor for V7.5.
- **New IBIS models.** Over 10,000 new models.
- **HyperLynx GHz**
 - **Via Visualizer.** Significant for multi-gigabit signals, Via Visualizer, in BoardSim GHz, allows you to view the geometric and electrical details of an individual via or a differential via pair, in the board viewer.
 - **Free-form schematic editor.** Easily instantiate SPICE and scattering parameter (s-parameter) models.
 - **Eye Mask Library.** Built-in library of eye masks for a variety of popular SERDES technologies. By accessing one easy combo box, you can display the appropriate mask for any of approximately 20 bus/link types.

For a detailed list of new features, refer to the What's New topic in online Help after software installation.

ICX

The following are the new features for ICX:

- **CES > ICX constraint integration.** From within CES, constraint classes, electrical nets and electrical constraints can be sent to ICX. Constraints include MST, TShape, Star and Htree topologies, differential pair assignments, length, time of flight and simulated delays, maximum crosstalk, static overshoot (high and low), ringback (high and low) and monotonicity. With CES open, cross-selection is also available between CES, ICX and Expedition or DxDesigner.

For additional information review the documentation within the ICX
_TOOLKIT\extras\ces2icx directory.

ICX Pro™ Explorer

The following are the new features for ICX Pro Explorer:

- **Complete What-if Analysis Environment.** ICX Pro™ Explorer enables pre-schematic what-if signal integrity analysis by exploring and testing various design options. You can vary topology, termination, IBIS model selection, board stackup, characteristic impedance and many more design parameters. Simulations with these varying parameters are called sweeps and may be used to determine the optimum design implementation. Design sensitivity to manufacturing tolerances and device variances can be fully explored.
- **Solution Space Exploration.** Performing simulation sweeps enables engineers to evaluate circuit performance over a range of operating conditions. Solution Space Exploration delivers that capability in an easy to use three-dimensional visual form that clearly shows under which conditions acceptable circuit performance is achieved. These conditions can then be expressed as design constraints to drive the physical implementation of the board.
- **Constraint Definition.** The Constraint Template Editor built into ICX Pro™ Explorer provides the same look and feel as CES, enabling designers to capture constraints based upon simulation results. Once established, constraints may be applied to nets or classes of nets directly in CES for use in the current project, or they may be saved as generic constraint templates to be used in other designs.
- **Proven Simulation Technology.** ICX Pro™ Explorer uses the same proven simulation technology as ICX®. This proven technology supports IBIS v4.1, including EBDs, SPICE, VHDL-AMS and S-Parameter based models. With ICX® simulation technology, designers can mix and match model formats, allowing them to use the right models, the first time.
- **Integration into a CES Design Flow.** Used standalone, or integrated into a CES design flow, ICX Pro™ Explorer provides signal integrity exploration for constraint definition. The intellectual property that is captured as design constraints may be re-used design after design.
- **QualityIBIS™.** With over 13,500 IBIS models (many created from actual measured devices), the QualityIBIS™ library provides you with a comprehensive library of accurate IBIS models for use in ICX Pro™ Explorer.

I/O Designer

The following are the new features for I/O Designer:

- **Import PCB Design Wizard.** I/O Designer 2005 SPac1 allows you to import an existing PCB design from Design Architect, DxDesigner or Design Capture/DesignView. The Import PCB Design Wizard guides you through the entire import process.
- **Update Wizard.** I/O Designer 2005 SPac1 provides an enhanced update mechanism called the Update Wizard. The wizard allows you to merge signal assignments if there are any conflicts. The wizard also displays a summary of what has been updated.
- **HPUX operating system support.** I/O Designer 2005 SPac1 is available on the HPUX platform.
- **Configurable power and ground PCB signal names.** I/O Designer 2005 SPac1 allows you to choose the PCB signal names for power and ground pins for symbol export.
- **Quartus II 5.0.** I/O Designer 2005 SPac1 includes support for Altera Quartus II 5.0.
- **DMS Corporate Library Integration.** There is a new option available called “Import symbols from DMS”, which is used to retrieve the appropriate component data from DMS and load it into I/O Designer.
- **Access Altera DRC's directly from I/O Designer.**
- **Support for devices from Altera, Xilinx and Actel, including Xilinx Spartan3e, Altera Hardcopy II, and Actel ProASIC3.**

Librarian Flow Manager

The successful Librarian Flow Manager (LFM) product for the EN release prompted the development of a similar product for the Expedition 2005 SPac1 flow. Based entirely on the latest web technologies, there is no dependency on WorkXpert. This uses a fully integrated Web-based Part Request System, which can be used with a web browser. Moreover, the request management is integrated into an extended version of Library Manager (run as “Librarian Flow Manager” from the Start Menu or LibrarianFlowManager from the command line), so that this process becomes automatic and seamless for the librarian. The major benefits are:

- **Create a part request in DMS, Librarian Flow Manager, or in a web browser.**
- **Notification and quick response are ensured by DMS notification mechanisms and the integration into the librarian’s usual working environment (that is, Library Manager).**

-
- **Improved communication.** You can log onto your web browser (or in DMS directly) and see the status of a request.

The Librarian Flow Manager extension to Library Manager is supported by DMS LFM EE addon Op SW (PN 225515). This license requires the Expedition Library Manager license and the DMS Libr. Mgr. Addon Ap SW (PN 226377). The web-based Part Request system is supported by Part Request Manager Ap SW-SL (PN 225517).

Library Manager

The following are the new features for Library Manager 2005 SPac1:

- **New Library Manager interface.** Library Manager 2005 SPac1 supports a new object-centric interface supporting both Dx-Expedition and DC-Expedition.
- **HPUX and Solaris Support.** Library Manager 2005 SPac1 supports the Solaris and HP-UX operating systems (in addition to Linux and Windows).
- **Navigator Tree.** Library Manager's new interface utilizes a navigator tree that provides hierarchical tree view of the library objects contained in the Central Library, organized by partition or type. This object-centric view clearly shows the relationship and association between library objects without having to invoke the different object editors. In addition to displaying the different library object types, the navigator tree provides the following features:
 - Create new part, symbol and cell partitions.
 - Create new library objects within part, symbol and cell partitions and padstack types.
 - View the following associations between objects:
 - o Part entries can be expanded to show the associated cell(s) and symbol(s).
 - o Cell entries can be expanded to show the associated part(s) and padstack(s).
 - o Symbol entries can be expanded to show the associated part(s).
 - o Padstack entries can be expanded to show the associated cell(s), pad(s) and hole(s).
 - Edit Library Objects - Library objects can be edited directly from the navigator tree by either double-clicking the object or using the popup menu Edit command. This opens the object for edit within the appropriate editor.
 - Find Command - This command enables searches for all object types including parts, symbols, cells, padstacks, pads and holes.
- **Output PDF.** The Output PDF command has been enhanced to support DxDesigner-Expedition Central Libraries.

-
- **Modify Pins.** A new Modify Pin Dialog has been added to the Tools pull-down menu, enabling librarians to quickly modify pin names or pin numbers of symbols and cells associated to parts.
 - **DxDataBook Configuration and Parametric Data Editing.** Library Manager 2005 SPac1 supports creating and editing of DxDataBook configurations and the editing of parametric data within the associated database. This functionality from Library Studio has been incorporated into Library Manager to enable the librarian to edit the configuration and parametric data with Library Manager, without invoking another tool. Here are a few highlights of this functionality:
 - The DxDataBook configuration can be setup and edited from Library Manager via the Edit DxDataBook Configuration command available on the main Central Library node within the navigator tree.
 - Once configured, DxDataBook properties, tables and parametric data can be edited via popup commands on each Part partition within the navigator tree which matches the libraries in DxDataBook.
 - Parametric Data for individual parts can be edited via the Edit parametric data command which is available via the popup menu. The parametric data edited in Library Manager is written back to the DxDataBook database.
 - **DxDesigner Symbol Wizard.** The DxDesigner Symbol Wizard can now be run from Library Manager via the Symbol Wizard command, which is available on symbol partition node popup menu.
 - **New DxDesigner Symbol Editor.** A new DxDesigner symbol editor has been incorporated into Library Manager. The editor provides comparable capability to DxDesigner/ViewDraw and reduces the active project dependencies between Library Manager and DxDesigner.
 - **Cell Editor Features**
 - **Rule Areas in Cells** - Rule Areas can now be defined in cells which can be mapped to Rule Areas in the PCB design.
 - **Associative Dimensioning** - A variety of Associative Dimensioning commands are now available in the Cell Editor which can be used to dimension specific object types in a cell. The dimensions can be defined on Assembly, Silkscreen or user-defined layers.
 - **Interactive DRC** - You now have the ability to temporarily turn OFF Interactive DRC within the Cell Editor.
 - **Cutouts** - The Subtract command can now be used to define one or more "cutouts" within shapes for specific closed draw objects. For some objects, the outer shape must be filled in order to contain cutouts, however, non-filled objects such as the Placement Outline can contain cutouts which will affect the area for the Height specification.
 - **Draw Enhancements** - In addition to Cutout support, the Trim, Extend, Join, Merge & Subtract commands have been enhanced in the 2005 SPac1 release. Also, arcs can be

-
- inserted during the definition of Polylines and Polygons using the right mouse button popup menu.
- **Cell Editor support for Padstack Enhancements** - The Cell Editor has been enhanced to support Padless holes, Padstack Layer Overrides, and Padstack User Layers.
 - **Mechanical and Drawing cells** - Conductive shapes and Trace objects can now be defined in Mechanical cells. Drawing cells can now contain Trace objects.
 - **Jumper Cells** - Silkscreen and Assembly Outline data can now be added to Jumper cells.
 - **Padstack Editor Features**
 - **User Layers** - Padstack definitions can now include Pads on User-defined layers (as defined in Setup Parameters).
 - **Layer Overrides** - Padstack definitions for Through padstacks (Through Pin, Mounting Holes, Vias, etc..) can now include inner-layer overrides (different pads for specified layers).
 - **Padless Holes** - Through Pin and Mounting Hole padstacks can now be defined using [No Pad] for Mount, Internal, and Opposite side pads.
 - **Draw Enhancements** - Several new Autoactive draw-mode commands were added to the Custom Pad editor in the 2005 SPac1 release to better support custom pad and drill symbol creation. The commands include: Move, Scale, Trim, Extend, Join, Merge, and Subtract. Also, arcs can be inserted during the definition of Polylines and Polygons using the right mouse button popup menu.
 - **X,Y Coordinate Readout** - The Custom Pad Editor now includes an X,Y coordinate readout at the top of the window.
 - **Part Editor Automation** - Support for Read-only Automation has been added to the Part Editor.
 - **Visual IBIS Editor** - The ICX Pro Visual IBIS Editor can now be launched from the Library Manager dialog.

Quiet Expert

The following lists the new features in Quiet Expert:

- **Results Classification and Sorting.** Major improvements have been made to facilitate better handling and classification of analysis results. Mechanisms now exist to classify severity of results and sort the violations according to their severity. You can also add comments to each violation, further increasing the layer of communication between inter-department teams. Refer to Chapter 1 in the *Quiet Expert User's Manual* for more information.

-
- **History Function.** Previous analysis run results can be saved and compared to current analysis results to insure previous violations have been fixed properly. Refer to Chapter 1 in the *Quiet Expert User's Manual* for more information.
 - **Added DRCs.** Five new DRCs have been added to the analysis set. These DRCs are in the areas of decoupling capacitor analysis, area fill analysis, I/O filtering, differential pair checking, and return path checking. Refer to the online help and Chapter 3 in the *Quiet Expert User's Manual* for more information.
 - **Advanced API Additions.** Seventeen new APIs have been added to the scan set. Most are in the area of analyzing complex paths, area fills and shielding. Refer to Chapter 2 in the *Quiet Expert Reference Manual* for more information.
 - **Automated Flow Support.** Automated translation flows for Expedition and Allegro are now included in the tool. Property passing has also been implemented to assist in automation of design information from the CAD system to Quiet Expert. Refer to Chapter 2 in the *Quiet Expert User's Manual* for more information.
 - **Board Viewer Enhancements.** The board viewer has been enhanced to allow transparent viewing of area fills. This allows you to “see” nets or other structures that exist below an area fill. Additionally, reference designators now appear on the board viewer to assist in the investigation of violations. Refer to Chapter 1 in the *Quiet Expert User's Manual* for more information.
 - **Various DR Fixes.** There are numerous DR fixes to both the Quiet Expert analysis engine and translation process. These can be found in the Release Notes on SupportNet for more information.
 - **Crossprobing.** Quiet Expert now supports crossprobing to the Expedition toolset (PCBExpedition, PCBPlanner and PCBViewer) for nets and parts. To enable cross-tool crossprobing, refer to the Additional Release Notes and Workarounds section in this document.
 - **Printable Documentation Added.** Printable PDF versions of the *Quiet Expert User's Manual* are available for this release. An InfoHub has been added for web-based searching and SupportNet access from Quiet Expert. Refer to the Documentation System Improvements topic in this section for more information.

XtremePCB

The following are the new features for XtremePCB:

- **CES integration**
- **Supports PCB Browser. Realtime updates in read-only browser.**
- **Crossprobing to Design Capture.**
- **Pin and Gate swapping with back annotation.**

-
- **Support for HP, Sun and Linux Platforms (in addition to Windows).**

Authorization Codes

To use the latest releases, you must be on support contracts for these products as of March, 2006. For more information about "Exact Access" authorization code formats, see the explanation on SupportNet at:

http://www.mentor.com/supportnet/process_policy/exact_access.html

You may download your site's existing authorization codes from SupportNet at:

<http://www.mentor.com/supportnet/licenses>

For additional information on licensing, refer to the *Licensing Mentor Graphics Software* manual.

Installation Information

This release uses the Mentor Graphics Standard Installation program. For additional information on installation, refer to *Managing Mentor Graphics PCB Systems Software* manual and the help system within the installation software. You can view this manual at the top level of the CD and on SupportNet.

Refer to the Additional Release Notes and Workarounds section later in this document for information about any installation issues.

Support Information

If you have questions about this software release, please log in to SupportNet. You may search the KnowledgeBase with thousands of technical solutions or open a Service Request online at:

<http://www.mentor.com/supportnet>

If you do not have a SupportNet login, you may easily request one by filling out the short form:

<http://www.mentor.com/supportnet/quickaccess/SelfReg.do>

For phone support in the United States or Canada, please call 1-800-547-4303. For phone support in other locations, please contact your local sales office or distributor. All Customer Support contacts can be found on our web site at:

http://www.mentor.com/supportnet/support_offices.html

As mentioned previously in the New Features section above, InfoHubs are HTML information centers that organize your sets of documentation and provide information links into SupportNet. InfoHubs

provide a combination of HTML- and PDF-based user, reference, process, and release information. Refer to that section for InfoHub set up information.

Changes to the default setting for Internet Explorer on Windows XP SP2 prevents the InfoHub from functioning. A fix for this issue is provided in the Additional Release Notes and Workarounds section that follows.

For additional InfoHub release notes and any other workarounds for this release, refer to the Expedition Flow 2005 SPac1 Release Notes manual on SupportNet.

Additional Release Notes and Workarounds

This section contains additional release notes and workarounds for issues found late in the release process for this software release. For a detailed list of all other release notes and any workarounds for this release, refer to the Expedition Flow 2005 SPac1 Release Notes manual on SupportNet.

Is there a significant change for all UNIX and Linux customers using DataBook or DxLibraryStudio?

With the 2005 release, there will be a significant change for all our UNIX and Linux customers who use DxDataBook or DxLibraryStudio. In our previous releases, in order for these products to connect to a standard ODBC (Open Database Connectivity) compliant database, we provided a special set of third party ODBC drivers. In the 2005 release, these drivers have been replaced with Mentor-developed ODBC drivers and direct database connections. All UNIX and Linux customers who have been using the third party ODBC drivers will need to modify their setup to use this new Mentor provided solution. The new drivers will support the following database connection:

Oracle 9.X

The drivers may work with other versions of Oracle, but we will only be qualifying DxDataBook and DxLibraryStudio on this major version. We are also introducing an ODBC interface into our 2005 Library Manager product that will use these new drivers. Connections to other vendor's databases will no longer be supported on UNIX or Linux.

Windows-based customers will not have to make any changes to their existing configuration. DxDataBook and DxLibraryStudio will continue to work as before. When making edits to the DxDataBook configuration that required an ODBC connection (such as adding a new table), a new data source (DSN) will need to be added.

The web server interface will still be supported on UNIX, Linux, and Windows platforms. A Windows IIS web server is required to publish the database data that DxDataBook reads.

This new ODBC interface will enable Mentor Graphics to provide a high performance and simple database connection for our UNIX and Linux customers. Please refer to your online help file for

information on how to set up the new environment. Mentor's Customer Support organization is also ready to help with any transition issues that may arise.

What privileges are required to run the configurator?

Beginning with the 2005 release, all environment variables set by the configurator are set as System variables rather than User variables. (The configurator is an application that sets up your Windows environment. For more information, see Chapter 4 in the *Managing Mentor Graphics PCB Systems Software* manual.) The result of this change is that tool installations are configured for all users of a given computer. Users who will be installing the tools or running the configurator are required to have administrator privileges on the computer; power user privileges are not sufficient.

Are there any issues running InfoHub from Internet Explorer on Windows XP Service Pack2?

Microsoft has made a change to the default settings for Internet Explorer in Windows XP SP2 that disables Javascript from running for locally installed HTML content, thus preventing the InfoHub from functioning. To fix this issue, select **Tools > Internet Options > Advanced**. Scroll down to the Security section, and select "Allow active content to run in files on My Computer". Click **OK**. This is not an issue for Mozilla or Netscape users.

Where can I find application program interface documentation for DxDesigner?

For object, method, property, and event information, please refer to the following online help file within your EXP2005 SPac1 installation:

```
<install directory>\2005EXP\SDD_HOME\wv\help\vdauto.hlp
```

For DxDesigner addins automation, please refer to dxdbauto.hlp and dxdmauto.hlp, which are also stored at this location. For all other DxDesigner API documentation, please refer to the DxDesigner InfoHub.

What are the recommended Central Library permissions?

To prevent individuals from making changes to a Central Library but still allow Read access for viewing library data, the Central Library must be loaded on a Windows NTFS or Linux/UNIX OS disk. All files, whether they are on Windows or Linux/UNIX, should have "Write" permission except for individual library files, which can be write-protected as desired to control access.

Note: Windows users can use the Properties/Security dialog box and Linux/UNIX users can use the chmod command to adjust Central Library permissions to corporate guidelines.

Files that can be permission-controlled are:

- Common property (centlib.prp) file
- Padstack library (padstackDB.psk) file located in the Layout folder

-
- Layout template and Panel template files
 - Files in the CellDBLibs, IBISModels, PartDBLibs, and SymbolLibs subdirectories.
 - Parts Manager database (partsmanager.mdb) file (DC/DV to Expedition flow only)

All other files in the central library should remain “Everyone = Change” (read-write) to ensure proper operation of the library. The Windows share that allows access to the central library directory from remote machines must have permissions set to “Everyone = Full Control”. The share permissions are superseded by the individual file permissions.

Are libraries created in EXP2005 SPac1 backward compatible with ISD2004?

Libraries created in EXP2005 are not backward compatible to previous versions. Libraries created in previous versions and migrated to EXP2005 cannot be used in prior released versions.

Can I use Librarian Flow Manager – DxDesigner – Expedition with Librarian Flow Manager –F1?

There are data model differences between the Librarian Flow Manager for DxDesigner/Expedition included in this release and the Librarian Flow Manager - F1 (for Design Architect/Board Station) that was included in the ISD2004 SPac2 release so they cannot be used with the same DMS database. If you have separate DMS databases for each flow, then you can use both products.

Can I configure DMS to support spaces in my Component Part Numbers and Cell catalog names in DMS Librarian and Librarian Flow Manager?

The default configuration of the DMS Data Model sets special character restrictions based on being able to support all Mentor Graphics products. If you are only using the Expedition 2005 SPac1 Flow, then you can remove some of these restrictions to have more flexibility. Refer to Chapter 2 in the *DMS Librarian User's Guide* for DMS configuration changes to support spaces.

Are there any known issues with resolution of library paths in the tools?

Yes. The following are the known issues:

- **Relative path is getting changed to full path in ini files (ex: viewdraw.ini)** - This should not cause any problem in schematic operations, but when copying the design to another location, library paths in ini files of the target location need to be examined and corrected. The workaround is to define the location map for the project and use soft prefixes to define library paths.
- **ViewDraw/Dashboard are resolving hard to soft path incorrectly if one soft path is a sub-string of other soft path** - No workaround exists. Make sure that no soft path is a substring of another soft path. For Example, '\$EPDLIB' & '\$EPDLIB_BACKUP'

-
- **Library paths in viewdraw.ini are not accessible** - Check the viewdraw.ini (or libsr/libswrw ini file referenced in draw.ini). Are the soft paths mentioned in the ini file properly defined in the location map file? Is the location map getting read from the correct/expected location?
 - **Library path in ini files contains \$HOME** - There should not be any problem if \$HOME is fixed in your environment. If you are changing your \$HOME very often, define \$HOME in the location map file and use the force option so that the environment value of \$HOME does not override the value defined in the location map file. Use "MGC_LOCATION_MAP_2 force" as the header in your location map file. You cannot use force for individual entries in the location map file.

Are there any issues with defining MGLS_LICENSE_FILE and LM_LICENSE_FILE?

Yes. Beginning with this release, the Configurator sets MGLS_LICENSE_FILE instead of LM_LICENSE_FILE to define the location of your licenses. If you already have defined LM_LICENSE_FILE to point to your licenses, this results in duplicate entries. Under some conditions, these duplicate entries may cause the tool to invoke more slowly. For more information on this issue and how to address it, see technote mg59067 (available on SupportNet).

How do I enable Inter-tool crossprobing between Quiet Expert and Expedition?

For the EXP2005 SPac1 release, Quiet Expert now supports crossprobing to the Expedition toolset (PCBExpedition, PCBPlanner and PCBViewer) for nets and parts. To enable cross-tool crossprobing, perform the following actions:

1. For Windows machines, edit the shortcut or command line to change the /syscfg option from "QuietExpert" to "QE_Expedition", "QE_Planner", or "QE_Viewer". This enables the application to search and connect to PCB Expedition, Planner, or Viewer, respectively. Quiet Expert can connect to only one tool at any time.
2. For UNIX platforms, three additional invoke scripts are provided in addition to the standard quietexpert script. These scripts are all located in the <install_location>/2005EXP/SDD_HOME/common/<platform>/bin directory and are named qe_expedition, qe_planner and qe_viewer.
3. To set up the applications for crossprobing, first invoke the desired Expedition tool and open the design of interest. Then invoke Quiet Expert using one of the methods described above. An indication in the workbench log tab shows that Quiet Expert has connected to the specified tool. Please note that Quiet Expert should already have the default project defined prior to invocation or the tool will not properly connect (reference MGC defect 100290486). This can be done by first invoking the tool standalone and setting the appropriate project as active and then closing the tool. This ensures that both tools are using the same data for connection and cross-selection.

Crossprobing can be toggled on and off using the crossprobe radio buttons located in the application toolbar. These buttons control both crossprobing within the application between the DRC results window and the boardviewer, and also between Quiet Expert and external tools as described above.

When the application is installed and is invoked for the first time, crossprobing is enabled by default. This default can then be controlled using the setting in the Tools > Options dialog. Please note that when changing options using the Tools > Options dialog, the setting reverts to an "OFF" mode and the default must be manually reset (reference MGC defect 100290490).

Will .vfm forms continue to work in this release?

Beginning with the EXP2005.05 release, DxDesigner no longer supports the .vfm format for forms. However, forms in the .efm format are still supported. You should rewrite any existing .vfm forms, which were used in prior releases, to .efm format for use in this and subsequent releases of DxDesigner.

What version of Allegro does ICX 3.5 support?

ICX 3.5 supports Allegro 15.2.

Are there any backward compatibility issues with I/O Designer?

Design files created in previous versions and migrated to I/O Designer 2005 SPac1 cannot be used in prior released versions.

Are any platforms new for this release?

Yes. Linux WS 4 and Windows Server 2003 are now supported. For more information, see Appendix G: Red Hat Enterprise Linux WS 4 (RHEL4), x86 and Appendix J: Windows Server 2003.

What version of Exceed does Quiet Expert support in this release?

Version 9.0 of Exceed terminal emulation software is supported. We do not support versions of Exceed earlier than 7.1 and we do not support XVision for Mainwin-ported products.

Is client/server setup available for the Windows platform?

Yes. The Server Setup for Client installs is used to install the software to a server-class system where multiple Windows client machines actually runs the software across the network.

The System Administrator should install the desired applications onto the server and select the Server Setup for Client Installs > Client/Server Setup option at the same time. The Client/Server Setup option installs a file called ClientInstallSetup.exe onto the server during installation.

To set up the clients, map a network drive to the location on the server where the applications were installed and run ClientInstallSetup.exe in the dialog that displays. Enter the path to the location where the applications are installed on the server.

Supported Platforms

Overall Notes

- Specified patches below are minimum levels. Later versions of the patches are valid, supported configurations.
- Except as noted, all products are supported on all platforms.

Appendix A: Solaris 8

Solaris 8 (July 2001) with Recommended Patch Bundle 03/16/2004

Recommended OS Patches:

- 108131-17
- 108434-15
- 108528-29
- 108576-46 (if Expert3D Graphics Card is installed)
- 108606-34 (if M64 Graphics Card is installed)
- 108987-13
- 109147-38 *
- 109154-20 (if GFX PGX32 Graphics Card is installed)
- 109607-01
- 111111-03
- 111310-01
- 112396-02
- 112564-07 (if XVR-1000 Graphics Card is installed)

*** This version of patch 109147 must be acquired independently of the above specified patch bundle and is required for successful operation of the product.**

Kernel Configuration: Default

Processor: Sun Ultra10 333MHz Minimum, SunBlade 2000 900 MHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Swap Space: 2X the amount of RAM

/tmp Space: 512 MB Minimum, 1GB Recommended

Appendix B: Solaris 9

Recommended OS Patches:

- Solaris 9 with Recommended Patch Cluster 04/2004 or later
- 112565-08 (if XVR-1000 Graphics Card is installed)
- 112622-xx (if PGX64 Graphics Card is installed)

Kernel Configuration: Default

Memory: 512 MB Minimum, 1GB Recommended

Processor: Sun Ultra10 333MHz Minimum, SunBlade 2500 1+GHz Recommended

Swap Space: 2X the amount of RAM

/tmp Space: 512 MB Minimum, 1GB Recommended

Appendix C: Solaris 10

Solaris 10 – FCS Version

Recommended OS Patches:

- None

Kernel Configuration: Default

Memory: 512 MB Minimum, 1GB Recommended

Processor: SunBlade 2000 900MHz Minimum, SunBlade 2500 1+GHz Recommended

Swap Space: 2X the amount of RAM

/tmp Space: 512 MB Minimum, 1GB Recommended

Appendix D: HP-UX 11.0 (PA-RISC 2.0)

Recommended OS Patches:

- QPK1100 – Quality Pack for HP-UX 11.00, March 2003 (or newer)
- PHSS_30965 – ld(1) and linker tools cumulative patch

Kernel Configuration: Require "CAE/ME/EE Engineering Workstation 64-bit" Tuned Kernel Parameters Set, and optionally set the following parameters:

Maxssize: 401604608

Maxsiz_64bit: 401604608

Note: These are optional parameter settings to make the best use of the virtual address space for a process. In the event that an application exhausts its virtual memory space, these settings can be reduced a minimum of 83570688 (0x4FB3000) until the application can be accommodated. Note that these settings are systemwide and therefore can affect other applications in the system.

Processor: HP C3600 552MHz Minimum, HP C3750 875MHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Swap Space: 2X the amount of RAM

/tmp Space: 512 MB Minimum, 1GB Recommended

HP-UX 11 Kernel Reconfiguration

Caution: Exit all applications before performing the following steps. Your system automatically reboots when you complete the last step.

1. At the UNIX superuser command prompt, enter the following to display the System Administration Manager dialog box: **Sam**
2. Double-click the **Kernel Configuration** icon.
3. Double-click the **Configurable Parameters** icon to display the Kernel Configuration dialog box.
4. From the **Actions** pulldown menu, select **Apply Tuned Parameters Set** to display the Tuned Kernel Parameters Set dialog box.
5. Select **CAE/ME/EE Engineering Workstation** in the list and click **OK**.

6. From the **File** pulldown menu, select **Exit** to close the Kernel Configuration dialog box. The Create a New Kernel dialog box displays.

7. Select **Create a New Kernel Now**.

Note: If you haven't already done so, exit all other applications because the next step reboots your system.

8. Click **OK**.

Appendix E: HP-UX 11i (PA-RISC 2.0)

Recommended OS Patches:

- PHNE_25644 - 11i Cumulative ARPA Transport Patch
- PHSS_30966 – ld(1) and linker tools cumulative patch
- GOLDAPPS11i – Gold Applications Patches for HP-UX 11i, March 2003 (or newer)
- GOLDBASE11i – Gold Base Patches for HP-UX 11i, March 2003 (or newer)

Kernel Configuration: Require "CAE/ME/EE Engineering Workstation 64-bit" Tuned Kernel Parameters Set, and optionally set the following parameters:

Maxssize: 401604608

Maxsiz_64bit : 401604608

Note: These are optional parameter settings to make the best use of the virtual address space for a process. In the event that an application exhausts its virtual memory space, these settings can be reduced to a minimum of 83570688 (0x4FB3000) until the application can be accommodated. Note that these settings are systemwide and therefore can affect other applications in the system.

Processor: HP C3600 552MHz Minimum, HP C3750 875MHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Swap Space: 2X the amount of RAM

/tmp Space: 512 MB Minimum, 1GB Recommended

HP-UX 11i Kernel Reconfiguration

Caution: Exit all applications before performing the following steps. Your system automatically reboots when you complete the last step.

1. At the UNIX superuser command prompt, enter the following to display the System Administration Manager dialog box: **Sam**
2. Double-click the **Kernel Configuration** icon.
3. Double-click the **Configurable Parameters** icon to display the Kernel Configuration dialog box.
4. From the **Actions** pulldown menu, select **Apply Tuned Parameters Set** to display the Tuned Kernel Parameters Set dialog box.
5. Select **CAE/ME/EE Engineering Workstation** in the list and click **OK**.
6. From the **File** pulldown menu, select **Exit** to close the Kernel Configuration dialog box. The Create a New Kernel dialog box displays.
7. Select **Create a New Kernel Now**.

Note: If you haven't already done so, exit all other applications because the next step reboots your system.

8. Click **OK**.

Appendix F: Red Hat Enterprise Linux WS 3 (RHEL3), x86 compatible

There is no support for Red Hat Enterprise ES or AS.

This release supports only the x86 (32-bit) version of RHEL3. It has not been fully tested on the AMD64 or IntelEM64T (64-bit) versions. Customers running AMD64-based systems or Intel Pentium4 or Xeon based systems with EM64T capabilities should still use the x86 version of RHEL3. This release supports both the base release of RHEL3 and Update 4. Other RHEL 3 update releases (2, 3, & 5) are expected to run, but have not been fully tested.

OS Version:

- `$ uname -rs`
 - Linux 2.4.21-4.EL (Base RHEL 3)
 - Linux 2.4.21-27.EL (RHEL 3 Update 4)

Required File Sets:

Select the following options on the Package Groups offered from install:

Desktops Package Group: X Window System, GNOME Desktop Environment

Applications Package Group: Graphical Internet

Development Package Group: Legacy Software Development

System Package Group: Printing Support

Windowing System:

```
$ /bin/rpm -q -f /usr/bin/gdm
```

gdm-2.4.1.6-2 (Base RHEL 3)

gdm-2.4.1.6.5 (RHEL 3 Update 4)

```
$ /bin/rpm -q -f /usr/X11R6/bin/X
```

XFree86-4.3.0.35.EL (Base RHEL 3)

Xfree86-4.3.0.78.EL (RHEL 3 Update 4)

Minimum Required OS Patches: None

Processor: Intel Pentium 3 900MHz Minimum, Intel Pentium 4 2+GHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Swap Space: 2X the amount of RAM

Kernel Parameters: No changes required.

Appendix G: Red Hat Enterprise Linux WS 4 (RHEL4), x86 compatible

This release supports only the x86 (32-bit) version of RHEL4. Customers running AMD64-based systems or Intel Pentium4 or Xeon based systems with EM64T capabilities should still use the x86 version of RHEL4.

This release supports both the base release of RHEL4 and Update 2. Other RHEL 4 update releases are expected to run, but have not been fully tested.

OS Version:

- `$ uname -rs`
 - Linux 2.6.9-5.EL #1 (Red Hat Enterprise Linux 4 Base Version)
 - Linux 2.6.9-11.EL #1 (Red Hat Enterprise Linux 4 Update 2)

Required File Sets:

Select the following options on the Package Groups offered from install:

Desktops Package Group: X Window System, GNOME Desktop Environment

Applications Package Group: Graphical Internet

Development Package Group: Legacy Software Development

System Package Group: Printing Support

Windowing System:

`$ /bin/rpm -q -f /usr/bin/gdm`

gdm-2.6.0.5-6 (RHEL 4 Base Version)

gdm-2.6.0.6-7 (RHEL 4 Update 2)

`$ /bin/rpm -q -f /usr/X11R6/bin/X`

xorg-x11-6.8.1-23 (RHEL 4 Base version)

xorg-x11-6.8.2-1.EL.13.15 (RHEL 4 Update 2)

Minimum Required OS Patches: None

Processor: Intel Pentium 3 900MHz Minimum, Intel Pentium 4 2+GHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Swap Space: 2X the amount of RAM

Kernel Parameters: No changes required.

IT Administration Notes:

Enable TCP connections for X via the gdm.conf file using the following instructions:

-
1. From the desktop (after OS install), edit /etc/X11/gdm/gdm.conf and change the line that reads

```
#DisallowTCP=true to  
DisallowTCP=false
```

2. Reboot System.

Optionally disable firewall and SELinux (This option is needed for XtremePCB, and may be needed for other system level administrative activities.)

1. Applications->SystemSettings-> Security Level
 - Firewall Options tab:
Security Level: Disable firewall
 - SELinux tab
Uncheck "Enabled (Modification Requires Reboot)"
2. Reboot System

Appendix H: Windows 2000

Additional OS Patches: Service Pack 4 (plus any Web Updates or Hotfixes to this service pack available through Microsoft)

Processor: Intel Pentium 3 900MHz Minimum, Intel Pentium 4 2+GHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Virtual Memory: 2X the amount of RAM

Appendix I: Windows XP Professional

Additional OS Patches:

Either of the following two configurations is supported:

- Service Pack 2
- Service Pack 1a (plus any Web Updates or Hotfixes to this service pack available through Microsoft). **Note:** There is no benefit to installing SP1a if you are already running Windows XP SP1. Microsoft does not recommend that you install SP1a if you are already running SP1.

Microsoft Windows XP Pro Service Pack 2 (optionally load all current updates via Windows Update)

Processor: Intel Pentium 3 900MHz Minimum, Intel Pentium 4 2+GHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Virtual Memory: 2X the amount of RAM

Appendix J: Windows Server 2003

Additional OS Patches:

The following configuration is supported:

- Microsoft Windows Server 2003, Standard Edition Service Pack 1 with all current updates via Windows Update.

While there is no known issue with running Windows Server 2003, Enterprise Edition or Windows Server 2003, Datacenter Edition, the product has not been tested with these versions, and therefore is officially unsupported.

Windows Server 2003 x64 Editions and Windows Server 2003 for Itanium-based Systems are not supported.

Processor: Intel Pentium 3 900MHz Minimum, Intel Pentium 4 2+GHz Recommended

Memory: 512 MB Minimum, 1GB Recommended

Virtual Memory: 2X the amount of RAM