



PADS2007 Release Highlights

**© 2007 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.

Table of Contents

Introduction	3
Release Goals	3
Improved PADS Configuration, Installation	3
All PADS Products	4
ECO Enhancements	4
Alphanumeric Pin Number Improvements	4
Part Type Editor Enhancements	5
File Locking.....	5
GUI Customization.....	5
New Macro Language	5
PADS Logic	6
ECO Enhancements – Rules Comparison	6
Dialogs, Menu Consolidation	6
Allow Single Attribute Editing in Attribute Label Properties Dialog	6
Launch Change Part Type Command from Part Type Label Properties	7
Launch Bus Properties Command from Bus Name Properties dialog.....	7
PADS Layout	8
RF Design in PADS Layout	8
DXF Import in Decal Editor for RF Component Creation	8
Via Array Enhancements.....	8
Artwork Improvements for Copper Shapes - Square and Chamfered Corners	8
Copper Creation.....	9
Solder Mask/Paste Mask Output CAM	9
Generate Drafting Shapes from Terminals	10
Drafting Shape Generation Dialog	10
Complex Solder Dams.....	11
Differential Pair Routing Improvements	11
New Text Commands and Uncombine Operation Improvement	11
Pin Number Visibility in PADS Layout, Router, Decal Previewer.....	11
DxDesigner.....	15
Alternate Decal Support	15
ECO Enhancements – Rules Comparison	15
Alphanumeric Pin Number Improvements	16
DxDesigner to PADS Router Cross Probing.....	16
PXR (Packaged Cross Reference) File	16
Added DxDesigner Link Tool as Primary Integration Tool.....	16
LineSim Integration with DxDesigner.....	17
DxSim.....	17
HyperLynx OCUR Release	19
Documentation	19
HTML Help System and InfoHub	19
Licensing and Installation.....	20
Updated Licenses.....	20
Licensing	20
CD and License Installation	20
New Install Location	21
Windows Vista	21
Windows XP Professional or Windows XP Home	21
Windows 2000.....	21
Support Information	21

PADS2007 Release Highlights

Introduction

Welcome to the PADS2007 Release Highlights. This release introduces exciting new functionality in the area of RF design, tighter integration between DxDesigner and PADS Layout, and defect fixes.

As with all releases, it also includes a significant number of defect fixes, see the [Release Notes](#) on the web.

Release Goals

This release continues to deliver improvements in the area of startup and overall usability—particularly for users new to PCB design, and delivers some long requested product enhancements. Usability improvements have been made by requirements gathered from PADS User Validations, customer requests through our Service Request program, and PADS User Group meetings.

Improved PADS Configuration, Installation

Improvements in installation and configuration have been made, particularly for DxDesigner/PADS customers, including easier PADS configuration setup and creation of PADS projects.

In ViewPCB, creating a PADS Project initializes the Netlist Format/Vendor to PADS 2007, and PCB Configuration File to pads2007.cfg.

DxDesigner Link has been added to the Tools menu so that it can be used for netlisting and crossprobing to PADS Layout.

All PADS Products

ECO Enhancements

ECOGEN has been updated for PADS Layout 2007 to include comparison of design rules. This new feature is enabled using command line switches to ECOGEN and also by dialog options from within Logic, Layout and DxDesigner. As with previous releases, this feature is provided as a “stand-alone” executable as well as being embedded in PADS Layout, PADS Logic and DxDesigner. Support for net class and pin-pair group comparison is also added, however, pin-pair groups are not currently supported by either DxDesigner or PADS Logic.

Different subsets of the PADS Layout design rules are supported by different flows. ECOGEN is configurable to allow these different subsets to be properly supported for each flow. Both the rule type and hierarchy levels are configurable.

- **DxDesigner to PADS flow**—the hierarchy levels supported are: Default, Net Class, and Net rules. The rule types supported are General and Diff-pair but not Conditional. The DxDesigner flow does not support layer specific rules, so layer specific settings within Routing and Diff-Pair rules are not supported.
- **PADS Logic to Layout flow**—the hierarchy levels supported are: Default, Net Class, and Net rules. The rule types supported are General, Conditional, and Diff-pair. The Logic flow supports layer specific settings in the rule types that it supports.

Alphanumeric Pin Number Improvements

Improvements have been made to support alphanumeric pin numbers in PADS Library, PADS Logic and PADS Layout. Improvements have been made in particular for the DxDesigner to PADS flow by removing the added complexity in DxDesigner for handling PADS specific problems with alphanumeric pin numbers. The cross reference between alphanumeric pin numbers and the footprint pins are stored in the PCB Decal (footprint) itself instead of the Part Type. This should also simplify creation of parts in that you no longer have to worry about pin sequence number assignment.

NOTE: For backward compatibility or for customers who want to continue it, using the PPN files for alphanumeric pin numbers and using a file mapping table in the Part Type is still supported.

For more detailed information, see the Alphanumeric Pins Transition Guide (pads_alphanumeric_tg.pdf) under \release_documents on CD 1 and the download image.

Part Type Editor Enhancements

Enhancements have been made to the Part Type editor in both Logic and Layout. It is now possible to Copy & Paste pin information and part attributes from Excel into the editor. This is a major productivity enhancement for creating large parts such as FPGAs.

File Locking

File locking has been added for all PADS products, thus when you are the first to open a design file, you become the owner of that design; the design file becomes locked for reading and writing by other users. If a design file is opened by one user and then a different user attempts to open it, the application opens the design file in read-only mode for the second user and displays a message that the file is already in use and locked for editing. The message includes the name of the design owner and the computer where the design owner locked the design.

The design file cannot be updated by other users while the design owner has the design file open. The 'Save' command is enabled but it launches the 'Save As' command, i.e. you can save the opened design to another file (which is not locked).

GUI Customization

You can customize toolbars, menu bar(s), dropdown and shortcut menus, as well as assign keyboard and mouse shortcuts using the Customize dialog. You can also rearrange toolbar buttons without calling the Customize dialog, by dragging the required button while pressing the Alt button. All parts of the customization state are saved into the current workspace. This allows you to change the whole set of toolbars, menus, shortcuts etc. because the customized Workspace is automatically loaded when starting a new session.

New Macro Language

A new macro language is available for PADS Logic and PADS Layout. The architecture and language resembles those used in PADS Router. Unlike the old macro language, the new macro language is human readable. For instance, the following macro lines represent the *Area Select* operation in PADS Layout:

```
Application.ExecuteCommand("Area Select", 0.28717in, 0.73746in)
MainView.MouseMove(0.28717in, 0.72825in, "L")
MainView.MouseMove(0.39163in, 0.57156in, "L")
MainView.MouseMove(0.52374in, 0.41487in, "L")
MainView.MouseMove(0.54217in, 0.39336in, "L")
Application.ExecuteCommand("Select", 0.54217in, 0.39336in)
Application.ExecuteCommand("Area Complete", 0.54217in, 0.39336in)
```

For more detailed information, see the Macro chapter in the Help.

PADS Logic

ECO Enhancements – Rules Comparison

ECOGEN has been updated for PADS Layout 2007 to include comparison of design rules. This new feature is enabled using command line switches to ECOGEN, as well as dialog options from within Logic, Layout and DxDesigner.

Different subsets of the PADS Layout design rules are supported by different flows. ECOGEN is configurable to allow these different subsets to be properly supported for each flow. Both the rule type and hierarchy levels are configurable.

For the PADS Logic to PADS Layout flow, the hierarchy levels supported are: Default, Net Class, and Net rules. The rule types supported are General, Conditional, and Diff-pair. The Logic flow supports layer specific settings in the rule types; however Pin Pair or Pin Pair group rules are not supported by PADS Logic.

Dialogs, Menu Consolidation

For greater efficiency, several dialog menus have been combined:

- Net Properties and Net Name Label Properties
- Reference Designator Rename and Label Properties

Allow Single Attribute Editing in Attribute Label Properties Dialog

Prior to the 2005 release, a double-click on an attribute label would bring up the attribute editor dialog. Now it brings up the attribute label properties dialog. The new dialog is the label properties dialog with an edit box added to allow editing of the selected attribute value. This provides the best of both worlds since you are normally only interested in editing the selected attribute and not all the attributes for the part.

Launch Change Part Type Command from Part Type Label Properties

The Part Type Label Properties dialog shows the current part type name and provides a **Change Type** button to call the existing Change Part Type dialog. All the controls in the **Part Type** group are now grayed out if multiple part type labels are selected. The Part Type Label Properties command remains as the command launched by double-clicking on a part type label.

Launch Bus Properties Command from Bus Name Properties dialog

The Bus Name Properties dialog shows the current bus name and provides a **Bus...** button to call the existing Bus Properties dialog. All the controls in the **Bus** group are unavailable if multiple bus labels are selected. The Bus Name Properties command remains as the command launched by double-clicking on a bus name.

PADS Layout

RF Design in PADS Layout

DXF Import in Decal Editor for RF Component Creation

The PADS Decal editor has now been updated to import DXF shape and line information. A new button is added to the Drafting Toolbar in the decal editor to provide access to the *Add From DXF* command. When invoked, the DXF file is checked for appropriate content and a dialog lists items to add to the decal. It converts most line geometries in a DXF file into equivalent line items in the decal editor. For example, an open polygon drawn defined within a DXF file becomes an open polygon in the decal editor. The specific DXF geometries supported for import from DXF into the decal editor include: POINT, LINE, ARC, CIRCLE, TRACE, SOLID, 3DFACE, POLYLINE, and LWPOLYLINE (AutoCAD R14). Supported geometries also include BLOCKS with hierarchy.

Via Array Enhancements

New enhancements have been made for via creation and placement requirements for RF design, typically for shielding crucial traces and/or filling copper pour areas. This functionality is accessible through two new commands – Via Shield and Via Stitch.

Via Shield (Channel/co-planar wave guides)—A via shield can be constructed around a routed net or pin pair to prevent or limit noise susceptibility. A via shield can be a collection of vias in a copper or copper pour area. They are arranged in an array or along the perimeter to provide greater conductivity between planes across more than one layer. The via shield command works for nets, pin pairs, coppers, or copper pour and shield it with vias according to the parameters defined in the *Via Pattern* tab of the *Preferences* dialog.

Via Stitch—Fills coppers or hatch outlines with vias.

Artwork Improvements for Copper Shapes - Square and Chamfered Corners

Improvements have been made for creating precise outline and corner geometry for copper shapes and traces. The prior method for displaying and plotting copper and traces restricted precision because the traces and copper outlines were drawn with a round pen/aperture equal to the line width of the trace or copper outline. The corners produced by this method were always rounded and it was difficult to precisely produce the required shape. In PADS Layout, the ability to create precise corner geometry for copper shapes has been added. Also, commands

have been provided to create copper based on traces you select, which allows selected traces to be replaced by copper shapes.

Routing and clearance checking—Copper outlines used to replace traces represent exact corner geometry subject to the chosen copper outline pen width, and are recognized by the DRC routines so that trace to object clearance rules are used instead of copper to object clearances. The Router sees the copper as a fixed obstacle so no violations are created in DRC ON mode. The copper does not push and shove like traces.

Square Corners—Orthogonal corners have a square shape. Diagonal corners that are not acute, have a diagonal outer angle instead of an arc. Acute corners are always chamfered, because the outer corner could protrude a great distance. The original trace is unrouted after the copper is generated.

Chamfered Corners—You can choose whether orthogonal corners should be chamfered or square. Acute corners are always chamfered. Since users typically have a formula that determines the distance between the inner corner and the outer chamfered edge as a ratio of trace width, you are allowed to specify this parameter. Alternatively, you may specify the length of the chamfered edge as a ratio of the trace width.

Displaying and CAM Output (Plotting) of Copper Traces—The copper representing the traces is displayed and plotted with real square or chamfered shapes. To avoid generating a large amount of hatch lines in the Gerber output, you should choose RS274X output.

Copper Creation

You create the copper with square and chamfered corners either by drawing the copper path (open copper) or by converting existing traces to copper. There is an existing command that creates an open copper path that has rounded corners just like a trace. A new command based on this existing command, is provided that allows you to specify the square or chamfered corner options and converts the open copper path (poly-line) into a closed polygon. You draw the open copper path as a normal open copper shape, but when you finish the command, the open copper is converted to a polygon with the square or chamfered corners. When you complete the path, a dialog is displayed allowing you to specify properties for the copper.

Solder Mask/Paste Mask Output CAM

Prior to PADS2007, CAM output for Solder Mask could only over/undersize pads for all components by the same amount - the CAM output can only either oversize or undersize by the same number. This is a problem for BGA type components that may need to have openings different from other parts (for example, BGA component pads may need to be undersized by 5 mils, while other component pads need to be oversized by 4 mils).

Now, in **CAM output for Solder Mask – Options**, you have the ability to specify oversize or undersize values for individual components or decals. Other components that do not have

variations use the default over/undersize number as always. These special over/undersize attributes can be assigned to components either on the decal or component level, and to vias. They are named *CAM.Solder mask.adjust* and *CAM.Paste mask.adjust* and set as visible system attributes. Solder mask and Paste mask pads for the components are generated using these attributes.

Combining two solutions – solder mask pads in pad stacks and component/decal attributes – Layout uses the following order of priorities:

- Pad stack information
- Component value of over/under size attribute
- Decal value of over/under size attribute
- CAM value of over/under size parameter

Additional Methods of Creating Drafting Shapes in the Decal

In some cases, the task of mask creation had been complicated, such as when solder mask over/undersized a pad on an electrical layer for unusual copper shapes; this required new methods of creating copper/copper cut outs.

Now you can shrink or grow any selected pin and store the resulting polyline as one of several drafting shapes. A copper or copper cut out can be used either as a free copper object or as a custom pad of the pin. An oversized keepout on an electrical layer in the decal is also useful to prevent the associated pin from routing, flooding, or from undesired fanout creation.

Generate Drafting Shapes from Terminals

With a terminal selected in the PCB Decal Editor, you can now right-click and click **Generate Drafting Shape** option. The option is available if one or more terminals (pins) are selected. The option activates a *Generate Drafting Shape* dialog.

Drafting Shape Generation Dialog

A new *Generate Drafting Shape* dialog is available (activated by the *Generate Drafting Shape* option) to request several parameters describing the object – 2D Line, Copper, Copper Cut Out, or Keepout.

Negative numbers in the Oversize/Undersize value field are treated as an undersize value, and can have current units either metric or mils. All selected pins can be expanded or contracted by that value, and combined with a polyline which can be saved as a drafting object on the selected layer. A zero value can be used to create a copy of the pad shape, which allows you to create a solder mask or solder dam, combining them into one object, and either associating it

with the selected pin or leaving it as a free decal copper. Complex pad shapes are processed by the command.

Complex Solder Dams

To create a complex solder dam, both pad stack information and copper/cut outs can be used. For example, you can create copper, surround a group of pins, then create a set of cut outs using the cut out generation dialog. Cutouts can then be combined with copper, which can be either associated with some of the pins or used as a free decal copper. To suppress output of other pins to the layer, you can set a round zero size pads for the pins on the layer.

Differential Pair Routing Improvements

Improvements include the functions “Complete routing on fanouts,” and “Start routing on pins with multiple connections attached.” Improvements include:

- General pad entry
- Complete routing on fan-outs
- Prior issues with completion for partially routed differential pairs

New Text Commands and Uncombine Operation Improvement

With text selected, you can now right-click and click Uncombine or Select Shape. If text objects are combined with 2D line shapes and other texts, both items are editable. The Uncombine command uncombines text from 2D line shapes, while the Select Shape command selects shapes. Both commands are unavailable in the case of multi-selection or if the text is not combined to any shape.

Pin Number Visibility in PADS Layout, Router, Decal Previewer

Provides the ability to see pin numbers overlaid on the pin graphics while in PADS Layout board and decal editors, and PADS Router while manually routing.

Usability Improvement - Autorouter Launching in Background Mode

The behavior of the autorouter launched in background mode has been updated to bring the routed design into the current session of Layout. In PADS Router, Monitor now opens showing routing statistics. The resulting .pcb file is loaded into the same instance of PADS Layout, which launches the autorouter. In PADS Router Monitor, the checkboxes “Load the resulting file” are selected and unavailable.

Ellipse Translation from DXF File

Improvements have been made in how DXF ellipse drafting objects are imported. To translate them more precisely, the system interprets them as polylines, and gives you fine control over tolerances of this interpretation.

Route Length Monitor in Bus Route Operation

A new *Ctrl+PageUp* shortcut toggles the Length Monitor on and off during Bus Route operation.

Conversion of Drafting Objects to Board Cut Out

The Drafting Properties dialog has been updated to support conversion of drafting objects as board cutouts. If the current design doesn't contain a board outline, the *Type* list of the Drafting Properties dialog doesn't contain this type as a choice.

Component Coppers/Cut Outs Display Improvement in Layout Mode

In prior versions of PADS Layout, the Decal Editor allowed combining coppers and cut outs into one object and displayed the cut outs as copper voids, but didn't allow association of such coppers with pins. Being assembled into a decal all combined coppers and cut outs were exploded when displayed since they had never been combined.

To improve copper/cut out functionality, the internal data representation of decals is improved by keeping combined objects. Both free coppers and coppers associated with pins are able to contain combined cut outs and are displayed in components as coppers with voids.

Improvements in CAM Output of Copper

In prior implementations, the CAM Output for coppers was managed by *Use fill mode* option of Photo Plotter Advanced Setup dialog. Behavior controlled by this option is as follows:

- If the option is not set, copper is hatched by horizontal lines and output as a set of lines.
- If the option is set and copper *doesn't* have associated voids, the copper is output as a filled polygon, or if the option is set and copper *does* have associated voids, it is cut by horizontal lines into several polygons. These polygons are output as filled polygons using G36/G37 codes. Both methods may produce a huge number of additional lines and dramatically increase the output file.

To improve the output, PADS Layout supports two formats of filled polygons (coppers) in the CAM document. The formats are the same as described in the previous paragraph, except the copper with voids is converted to a simple connected polygon using a minimal number of additional lines. *Use fill mode* option is operable only in the RS274X format.

New *Through/Partial* Column in the Drill Chart

A new *Show Through/Partial column* check box has been added to the Drill Drawing Options dialog. When it is selected, one additional *Through/Partial* column appears in the table. If both through and partial drills have the same size, it is shown in the table in two rows and the total number of drills of this size is distributed between these rows. Check boxes of the *Through/Partial* row are used to manage output of corresponding drills to the document. They are selected by default.

If the *Show Through/Partial column* check box is cleared, the corresponding column of the table is hidden and both through and partial drills of the same size are combined into one row.

Drill Table Improvement - Preserve Tolerance Values after Regeneration of Drill Symbol Table

Both Regenerate and Augment operations of the Drill Symbol Table preserve already-assigned tolerance values for the drill sizes presented both in the original and regenerated tables. The default tolerance value is assigned to any new drill size.

Modeless Commands for Switching Current Design Units

Modeless commands for switching design units are supported both in Layout and Decal Editor modes. They are:

- UM for mils
- UMM for millimeters (metric)
- UI for inches

PADS Router also supports these modeless commands. In addition, Router supports the UUM command for microns.

DxDesigner

Alternate Decal Support

PADS Layout part type definitions allow the specification of a list of legal footprints (including a default footprint) for a part number. DxDesigner users can now assign an alternate footprint and pass that footprint assignment to PADS Layout through the netlist ASCII file or through an ECO file produced by ECOGEN.

This is accomplished using the DxDesigner PCB interface, which supports the use of an ALT_PKG_LST attribute for specifying a list of alternate footprints. ALT_PKG_LST support also allows DxDesigner librarians to specify the list of allowable footprints for any particular part number, which are passed to PADS Layout through the *<design>.p* ASCII file during forward annotation. This is an improvement over past functionality, which only passed the list of footprints actually assigned to the part number on the schematic at any given time.

If the current active document in DxDesigner is a symbol, the PADS Decal Browser enables you to update the PKG_TYPE or ALT_PKG_LST based on selections in the Decal Browser.

Tools>Check PADS Decal Pin Numbers—This application checks all of the component pin numbers in the design against the pin numbers of the PADS Decals based on the PKG_TYPE and ALT_PKG_LST attribute value.

ECO Enhancements – Rules Comparison

ECOGEN has been updated for PADS Layout to include comparison of design rules. This new feature is enabled using command line switches to ECOGEN, as well as dialog options from within DxDesigner, PADS Logic, and PADS Layout.

Different subsets of the PADS Layout design rules are supported by different flows. ECOGEN is configurable to allow these different subsets to be properly supported for each flow. Both the rule type and hierarchy levels are configurable.

For the DxDesigner to PADS flow, the hierarchy levels supported are: Default, Net Class, and Net rules. The rule types supported are General and Diff-pair, **but not Conditional**. The DxDesigner flow doesn't support layer specific rules, so layer specific settings within Routing and Diff-Pair rules are not supported nor are Pin Pair and Pin Pair Group rules supported.

Also, for existing designs in DxDesigner where the Constraint Editor has been used for passing rules, there is a utility, **Tools>Merge PADS CNS File**, which merges data from a previous version of the .cns file to the new pads2007.cns file. This is especially important if you have created classes which get stored in the .cns file.

Alphanumeric Pin Number Improvements

The PADS alphanumeric pin assignment scheme has been greatly simplified for DxDesigner to PADS Layout users. In order to properly support alphanumeric pin information in DxDesigner in the past, information was stored in both the schematic and a separate ASCII .ppn file. The new scheme no longer requires the use of a .ppn file. In addition, alphanumeric pin information in the PADS Layout library was stored in the Part Type, with a cross reference to numeric pin assignments used in the PCB Decal (or footprint) itself. Now, instead of storing them in the Part Type section of the library, alphanumerics are assigned directly in the PCB Decal. You need to ensure that alternate decals have alphanumeric pin numbers as well, otherwise, there will be import/update errors.

NOTE: For backward compatibility or for customers who want to continue it, using the PPN files for alphanumeric pin numbers and using a file mapping table in the Part Type is still supported.

A library converter is available to auto-generate each variation of a decal used by a part, based on the pin information stored in the Part Type. This converts all PADS Libraries from Ver. 4 (ex. lib.pt4) to Ver. 7 (ex. lib.pt7). It is only necessary to convert library files once.

DxDesigner to PADS Router Cross Probing

There is currently cross probing functionality between DxDesigner and PADS Layout through the DxDesigner Cross Prober Addin and DxDesigner Link. In order to match PADS Logic functionality, cross probing has been added between DxDesigner and PADS Router. Adding a PADS Router module to the DxDesigner Cross Prober addin allows you to launch or connect to a PADS Router session from DxDesigner and enables bi-directional cross probing of components, nets and pins.

PXR (Packaged Cross Reference) File

The DxDesigner Cross Prober Addin requires the presence of a .PXR file. The .PXR file contains the mapping of schematic gates and nets to their physical counterparts. This file is updated whenever the schematic is packaged and whenever forward or back annotation (pcb fwd and pcb bck) are run. Since it is possible to edit the schematic without updating the .PXR file and Router/Layout databases, any added or renamed components and nets don't cross probe correctly until the PXR file and PADS Layout database are updated through forward annotation, and PADS Router has been launched from the updated PADS Layout database.

Added DxDesigner Link Tool as Primary Integration Tool

The DxDesigner Link provides dynamic cross probing between DxDesigner and PADS Layout much like the DxDesigner Cross Prober. The DxDesigner Link cross prober provides two

dialogs: a two-way selection list identical to the DxDesigner Cross Prober Dialog, and a Placement list allowing selection of placed or unplaced components. The interface allows you to target the 2007 PADS Layout ASCII format, and supports PADS2007 Layout functionality, particularly the new PCB configuration and constraint definition files, and rule change management.

LineSim Integration with DxDesigner

DxDesigner is now integrated with LineSim via a new interface. This link provides a means for engineers to select and analyze critical signals. The main objectives are:

- Automate net schematic creation in LineSim by importing net data from DxDesigner.
- Provide an interface that enables you to perform “what-if” and topology exploration

The interface:

- Updates and retains signal integrity model assignments in DxDesigner made within LineSim
- Updates DxDesigner schematics or provides you with data to update schematic with new termination requirements
- Allows you to easily re-run simulations with any updated stack-up and t-line properties

You can select electrical nets in your DxDesigner schematics (including any appropriate net attribute information, component values and pin/part models already defined) with one-button operation, and have that circuitry automatically loaded into LineSim for signal integrity simulation. The interface is designed to assist you in back-annotating any new data for net termination. Results from simulation in LineSim assists you in determining routing topology constraints and net length/delay constraints. However, the results from LineSim simulation are not suitable for direct interpretation for constraint synthesis, so you need to enter any derived constraint data manually into DxDesigner.

DxSim

DxSim is an addition to the DxDesigner family and provides an integrated, easy to use, analog design environment. From within this single environment you have access to all the tools required to take the circuit from concept through simulation through to PCB layout.

Building upon the proven DxDesigner schematic editor, DxSim provides a series of features that speed the complete analog design cycle.

By using libraries that support both processes there is no need to re-enter the schematic, thus significantly shortening the overall development cycle and reducing opportunities for error.

Symbol drag n drop—You can quickly incorporate new devices into the design. This allows you to carry-out extensive what-if permutations without having to commit new components to a library. By simply *dragging* a Spice model file from the windows browser into the schematic window.

Modeling—DxSim also allows you to create new or edit existing Spice models using a context sensitive editor. These can then be stored in the model library for future use by other designers.

Stimulus—Stimulus can be quickly applied to the circuit using the *add source* graphical user interface. Through the dialog, you have access to all standard stimuli such as *sine*, *pulse*, *piecewise linear (PWL) signals*, etc.

Simulation—The simulation specific commands are accessed from the additional DxSim toolbar. The DxSim toolbar guides you, step-by-step, through the simulation process. Using the Mentor Graphics' *EASE* simulation technology, DxSim provides fast and accurate analysis.

Simulation capabilities:

- DC
- AC
- Transient
- Frequency and time domain analysis
- Monte carlo
- Sensitivity analysis*
- Stress analysis - Smoke analysys*
- Worst Case Analysis*

*Note: These features are currently available through the advanced option TAB.

Simulation—The integrated waveform process provides you with a flexible tool where you can view waveforms from single or multiple simulation runs. This capability coupled with the waveform calculator allows you to compare different design scenarios and to predict circuit performance.

DxSim key features:

- Add-on to DxDesigner
- Easy-of-use.
- Single environment to create schematic, generate models, and simulate

-
- Same schematic to drive PCB and simulation
 - Simulation walk-through toolbar
 - Model organisation
 - *Drag-n-drop* model-to-symbol generation
 - Databook integration
 - Comprehensive and interactive waveform post-processing functions with a wide range of display formats
 - PSpice model conversion to native format

HyperLynx OCUR Release

Out-of-cycle Update Releases (OCURs) are fully qualified release candidates that are waiting for an approved release vehicle to be included in for global customer consumption. The difference between an OCUR release and a global release is the scope of the distribution. OCURs are supported in a customer production environment and are always rolled in the next available global release.

The new HyperLynx build number is 385.

Documentation

HTML Help System and InfoHub

The format of help is changing from .chm files to HTML help. Instead of being limited to using the operating system browser, the help launches your default browser to display the help.

A new information storage/retrieval scheme known as InfoHub is implemented for the 2007 release. InfoHub provides a new method to search through all local files, and provides stand-alone navigation, access, and search of all installed documentation. You can access and read product documentation without opening the tools. You can use whatever browser you prefer. InfoHub also enables access to HTML and PDF formats of all product documentation.

Licensing and Installation

Updated Licenses

An updated license is required to access the new RF design functionality. If you received a 2007 license file prior to the release of PADS2007, you need to obtain a new license file that contains '**pwradvedtrf**' to enable new RF functionality.

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

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

Licensing

To move towards a more Mentor-licensing compliant installation, this installation only sets a MGLS_LICENSE_FILE variable. All Mentor products are designed to first search for a MGLS_LICENSE_FILE variable before the LM_LICENSE_FILE variable (and other typical Flexlm locations). MGLS_LICENSE_FILE can have multiple paths. For example:

C:\Documents and Settings\All Users\ApplicationData\pads\2007\licensefile.dat;

C:\Documents and Settings\All Users\ApplicationData\pads\2005_2\licensefile.dat.

If your system has only an LM_LICENSE_FILE variable present, the system also receives a new MGLS_LICENSE_FILE variable. The location to where the license file is copied during a Nodelocked/Dongle based install has changed. This location is now "C:\Documents and Settings\All Users\ApplicationData\pads\2007\licensefile.dat". This is also the value to which the MGLS_LICENSE_FILE variable is set.

Tip: The "Application Data" folder under the "C:\Documents and Settings\All Users" folder is hidden by default. To be able to view the directory in Windows "Explorer" or "My Computer" you need to turn on the "Show hidden files and folders" option in the "View" tab of the "Folder Options" application (in the Windows Control Panel).

CD and License Installation

All PADS Flow products are now included on two installable CDs or a single download. As before, with new installations of PADS products, the license file can determine which products to install from the distribution CD or CD image you downloaded.

New Install Location

The PADS2007 installation now installs the PADS products to the C:\MentorGraphics\2007PADS directory.

The install now sets up a library path, and no longer prompts for an alternate library path.

Windows Vista

Windows Vista is not supported at this time. Do not install the PADS software products on Windows Vista.

Windows XP Professional or Windows XP Home

Microsoft Internet Explorer 7.0 is not supported.

Additional OS Patches—Service Pack 1a or Service Pack 2 (plus any Web Updates or Hotfixes to this service pack available through Microsoft). **Tip:** 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.

Memory—512 MB Minimum, 1GB Recommended

Virtual Memory—2X the amount of RAM

Windows 2000

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

Memory—512 MB Minimum, 1GB Recommended

Virtual Memory—2X the amount of RAM

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