



PADS® Layout Translator User's Guide

PADS 9.5

**© 1987-2012 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: supportnet.mentor.com/

Send Feedback on Documentation: supportnet.mentor.com/doc_feedback_form

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/trademarks.

Table of Contents

Chapter 1	
Introduction to the PADS Layout Translator	5
Chapter 2	
Translating CADSTAR Designs and Libraries	7
Supported File Types	7
Translation Notes	7
Library Translation Notes	7
Design Translation Notes	8
Translating Designs and Libraries	10
Combining Layout and Schematic Libraries into a Single PADS Library	12
CADSTAR to PADS Rule Mapping	13
CADSTAR to PADS Layer Type Mapping	16
CADSTAR to PADS Areas and Coppers Mapping	17
Chapter 3	
Translating OrCAD Designs and Libraries	19
Supported File Types	19
Translation Notes	19
Library Translation Notes	19
Design Translation Notes	19
Translating Designs and Libraries	21
Combining Layout and Schematic Libraries into a Single PADS Library	22
OrCAD To PADS Design Rule Mapping	23
OrCAD to PADS Layer Mapping	25
OrCAD to PADS Obstacle Mapping	26
Chapter 4	
Translating P-CAD Designs and Libraries	27
Supported File Types	27
Translation Notes	27
Library Translation Notes	27
Design Translation Notes	28
Translating Designs and Libraries	30
Combining Layout and Schematic Libraries into a Single PADS Library	33
P-CAD to PADS Rule Mapping	34
P-CAD to PADS Default Layer Mapping	35
P-CAD Object Type to PADS Rule Owner Mapping	36
Chapter 5	
Translating Altium/Protel Designs and Libraries	37
Supported File Types	37

Translation Notes	37
Library Translation Notes	37
Design Translation Notes	38
Altium Designer Translation Restrictions	39
Translating Designs and Libraries	41
Combining Layout and Schematic Libraries into a Single PADS Library	43
Altium/Protel to PADS Rule Owner Mapping	44
Altium/Protel to PADS Rule Mapping	45
..... Altium/Protel to PADS Layer Type and Name Mapping	48

Third-Party Information

End-User License Agreement

Chapter 1

Introduction to the PADS Layout Translator

The PADS Layout translator converts CADSTAR, OrCAD, P-CAD, Protel and Altium PCB designs and libraries to PADS Layout designs and libraries. You can translate:

- CADSTAR design files in both binary (*.pcb) and ASCII (*.cpa) formats generated in CADSTAR 5.0, 6.0, 7.0, 8.0.
- CADSTAR PCB Components, Parts and Documentation Symbols libraries in binary (*.lib) and ASCII (*.cpa) formats generated in CADSTAR 5.0, 6.0, 7.0, 8.0.
- OrCAD Layout design files in both binary (*.max) and ASCII (*.min) formats generated in OrCAD Layout 9.X, 10.X.
- OrCAD PCB libraries in both binary (*.llb) and ASCII (*.min) formats generated in OrCAD Layout 9.X, 10.X.
- P-CAD design files (*.pcb) in both binary and ASCII formats generated in P-CAD 2001, 2002, 2004, 2006.
- P-CAD PCB libraries in the binary format (*.lib) generated in P-CAD 2001, 2002, 2004, 2006.
- Protel 99 design files (*.pcb) in both binary and ASCII formats, including designs stored in Protel Design Database (*.ddb) project files.
- Protel 99 PCB library files (*.lib), including PCB libraries stored in Protel Design Database (*.ddb) project files.
- Altium design files (*.pcbdoc) in binary or ASCII format generated in Altium DXP, 2004, 2006, or in Altium Designer up to version 7.1 (Summer08).
- Altium PCB library (*.pcblib) files, including libraries stored in Integrated Libraries (*.intl), generated in Altium DXP, 2004, 2006, or in Altium Designer up to version 7.1 (Summer08).

Translated files are output in the binary file format of the PADS2005 SPac3 release.

See the appropriate chapter for translating your designs and libraries to PADS Layout:

[Translating CADSTAR Designs and Libraries](#)

[Translating OrCAD Designs and Libraries](#)

[Translating P-CAD Designs and Libraries](#)

[Translating Altium/Protel Designs and Libraries](#)

Chapter 2

Translating CADSTAR Designs and Libraries

Supported File Types

The PADS Layout translator converts the following categories of CADSTAR design and library files:

- CADSTAR design files in binary (*.pcb) or ASCII (*.cpa) format generated in CADSTAR 5.0, 6.0, 7.0, or 8.0.
- CADSTAR PCB Components, Parts and Documentation Symbols libraries in binary (*.lib) or ASCII (*.cpa) format generated in CADSTAR 5.0, 6.0, 7.0, or 8.0.

Tip: The PADS Layout translator does not convert CADSTAR Schematic Symbols libraries. Use the PADS Layout Schematic translator to translate these libraries.

Translation Notes

The following sections describe how differences between CADSTAR and PADS Layout are handled by the translator.

Library Translation Notes

PADS Layout Library Structure

A PADS Layout library consists of 4 files:

- Part Type (.pt9)
- PCB decal (.pd9)
- CAE Decal (.ld9)
- 2D Lines (.ln9)

The translator automatically creates these files from the single CADSTAR source file. The output Part Type and PCB Decal files contain data; the CAE Decal and 2D Lines files do not contain data—their data is generated by the PADS Schematic Translator.

CADSTAR Libraries

CADSTAR library files can be one of the following types: Parts, PCB Components, Schematic Symbols, and Documentation Symbols. Since part entries in the Parts library refer to symbols located in a separate PCB Components library, make sure that the associated PCB Components libraries are located in the same directory as the Parts library you are translating.

Design Translation Notes

Output File Names

The translator preserves the original file name while translating the file into PADS Layout unless the original file extension is .pcb. If the original file extension is .pcb, then the translator adds "_pads" to the original file name to get a translated design file name.

Design Rules Translation

See [CADSTAR to PADS Rule Mapping](#) for details of the mapping of CADSTAR design rules to PADS Layout.

Layer Name and Type Translation

Layer Name Translation

The translator gives you the option of preserving the original CADSTAR layer names, or changing them to the corresponding default PADS Layout layer names.

Note: If a CADSTAR layer has no equivalent layer type in PADS Layout, the translator uses the CADSTAR name in the translated file.

Layer Type Translation

See [CADSTAR to PADS Layer Type Mapping](#) for details of design layer mapping from CADSTAR to PADS Layout.

Areas and Coppers Translation

Since PADS Layout does not support hatched coppers, the translator flags hatched Figures and unattached Coppers as a warning in the log file, and translates them as regular pieces of drafting copper in PADS Layout.

Since there are no equivalents in PADS Layout for CADSTAR's Placement Area, Routing Area and Area Cutout, these Areas and Area Cutouts are ignored by the translator and an appropriate message is entered in the log-file.

See [CADSTAR to PADS Areas and Coppers Mapping](#) for information on how CADSTAR Areas and Coppers are translated to PADS Layout.

Component Outline Color

Component outlines may be colored differently when loaded in PADS Layout and PADS Router. PADS Router uses the Top Layer color to display the component outline, but PADS Layout uses the color of the layer where the component outline was originally placed.

Flooding/Hatching

Translated copper pour and plane area objects aren't automatically hatched when you load the translated design in PADS Layout. They are automatically hatched when you load the translated design in PADS Router.

Self-intersecting Copper Pours

Some copper pours that are self-intersecting, or that have areas too small for hatching, cannot be flooded/hatched by PADS Layout. The translator detects these coppers, flags them in the log-file as an error, and translates them as a 2D-line object in PADS Layout.

Figure Attributes

Since PADS Layout does not support attributes on geometry, keepouts, coppers and connections, CADSTAR's Figure, Area, Copper, Template and Connection attributes are not translated. If the **Translate figure and copper labels as free text** option is checked, the translator creates the attributes' values as free text objects.

Net Class Attributes

CADSTAR Net Class attributes are translated as net attributes for all nets included in the Net Class. CADSTAR Spacing Classes are translated as PADS Layout Net Classes. CADSTAR Net Spacing rules are translated as PADS Layout Net Class general and conditional rules.

Multi-line Labels and Attributes

A multiline free text object from a CADSTAR design becomes multiple single-line free text objects in PADS Layout. A multiline attribute from a CADSTAR design is translated as a single-line label or attribute to PADS Layout with concatenating multiple lines to single line.

Cross Hatch Patterns

Second Hatch Line parameters from CADSTAR design Hatching Assignments are ignored.

Dashed and Dotted Lines

Since PADS Layout does not support non-solid line styles, the translator emulates dashed and dotted lines as drawing shapes with a set of small line and arc segments.

Translating Designs and Libraries

Perform the following procedure to translate CADSTAR designs and libraries to PADS Layout.

1. **Start menu > Programs > Mentor Graphics SDD > PADS (version) > Translators > PADS Layout Translator**
2. Select the **CADSTAR** tab.
3. Populate the Files to translate file list:
 - a. In the Files to translate area, select the appropriate check box(es) to specify whether to translate designs, libraries, or both. Then click **Add**.
 - b. In the Select Input Design(s) dialog box, browse to and select the source file(s) you want to make available for translation. Then click **Open**. The selected files are added to the Files to translate file list. Files newly added to the list are checked (enabled for translation) by default.

Note: If an added file's format is not recognized, or is recognized as a PADS format, the file is flagged in the list and is not translated.

To remove a file from the list, select it and click **Remove**.

Important: If you are translating a CADSTAR Parts Library, all associated PCB Components libraries must be located in the same directory as the Parts library.

4. Set the output location for translated files:
 - a. In the Designs box, accept the default output location for translated design files, or click the **...** button to browse to and specify a different one.
 - b. In the Libraries box, accept the default output location for translated library files, or click the **...** button to browse to and specify a different one.

Tips:

- The default values are from the current FileDir (designs) and LibDir (libraries) entries in the PADS Layout powerpcb.ini file.
 - If the output file is open in PADS Layout, the file is not translated and a message is written to the output log.
5. In the Overwrite existing files drop-down list, select one of the following to specify how the translator should proceed if it needs to overwrite an existing file:

No—Leave the existing file intact; the source file is not translated.

Yes—Overwrite the existing file.

Ask—Display a prompt asking whether to overwrite the existing file.

6. In the CADSTAR tab, select the check boxes to set translation options.

- Select the **Preserve net colors** check box to preserve the colors of CADSTAR nets for corresponding nets in PADS Layout.

Tip: Because PADS Layout net color applies to nets, pads, vias, and coppers, while CADSTAR net color applies only to unroutes, the translated design colors may not match the original design colors.

- Select the **Rename layers** check box to rename CADSTAR layer names to PADS Layout names. See [CADSTAR to PADS Layer Type Mapping](#) for information on CADSTAR to PADS layer name mapping.
- Select the **Translate figure and copper labels as free text** check box to preserve the CADSTAR figure, copper, template and connection attribute values as free text objects in PADS Layout. If this check box is unselected, attributes of these types will be ignored.
- Select the **Explode doc symbols** check box to allow exploding of CADSTAR document symbols to collections of line, arc and text objects. If the box is unchecked, document symbols are translated as component symbols in PADS Layout.

7. **(Optional)** Click the **Attributes Mapping** button to display the Attributes Mapping dialog, where you can add, delete and modify CADSTAR component and net attribute names, and map them to PADS Layout equivalents. (The translator uses the attributes mapping while translating part type, decal, component, net class, and pin attributes. The mapping is preserved in the translator's configuration ini-file.)

Tip: Attributes that are not listed are translated using the original name from the source design; they have an attribute type of Free Text.

- To add a new item to the attributes map:
 - a. Select the Component Attributes, Net Attributes, or Pin Attributes tab.
 - b. Type the CADSTAR attribute name in the **Name** box (or select it from the list directly below the box.)
 - c. Select the corresponding PADS attribute name in the **Map to** box.
 - d. Select the type of the attribute in the **Type** combo box and click the **Add** button.

Tip: To add an item with a new unit type, select **Measure** in the Type combo box, type the new unit name in the **Quantity** combo box, click the **Add** button and specify a unit abbreviation (for example, m or F) and a short unit

name (for example, meter or Farad) in the New Unit dialog box. Then click the **Add** button again.

When you are finished adding, deleting and modifying items, Click **OK** to close the Attributes Mapping dialog.

8. In the PADS Layout Translator dialog, click **Translate**. The displayed Translation Progress indicator shows both current file and overall job progress.

When translation is complete, the Translation Results dialog box displays the content of a log file created during translation. The log file contains the following information:

- Path of the translated files
- Number of errors and warnings
- Detailed description of each error or warning

Combining Layout and Schematic Libraries into a Single PADS Library

You may want to have a single PADS library that includes all the Part Type, PCB Footprint, and CAE Decal data. To create a single PADS library, perform the following procedure.

1. Using the Layout translator, translate the CADSTAR library.
2. Using the Schematic Translator, clear the Overwrite existing files check box, and translate the CADSTAR library. Save the schematic library to the same directory where you saved the library you translated in Layout.

Result: The Schematic translator does not overwrite Part Types created by the Layout translator, and it creates records in the schematic symbol library file (.ld9).

CADSTAR to PADS Rule Mapping

Table 2-1 maps the translation of design rules from CADSTAR to PADS Layout.

Note: CADSTAR design rules not listed in the table are not translated:

Table 2-1. CADSTAR to PADS Rule Mapping

CADSTAR		PADS Layout		Comment / Value Map
Category	Name	Category	Name	
Assignments / Design Rules	Comp Placement to Comp Placement	Clearance	Body to Body	Applies to Default rules from (Default) layer entries, All against Layer. Conditional rules from layer reassignment entries.
	Copper to Board	Clearance	Board to Copper	
	Copper to Copper	Clearance	Copper to Copper	
	Hole to Hole	Clearance	Drill to Drill	
	Pad to Board	Clearance	Board to Pad	
	Pad to Copper	Clearance	Copper to Pad Drill to Copper Text to Pad	
	Pad to Pad	Clearance	Pad to Pad Drill to Pad	
	Pad to SMD Pad	Clearance	SMD to Pad Drill to SMD	
	Pad to Via	Clearance	Pad to Via Drill to Via	
	Route to Board	Clearance	Board to Trace	
	Route to Copper	Clearance	Copper to Trace Text to Trace	
	Route to Pad	Clearance	Pad to Trace Drill to Trace	
	Route to Route	Clearance	Trace to Trace	
	Route to SMD Pad	Clearance	SMD to Trace	
	Route to Via	Clearance	Via to Trace	
SMD Pad to Board	Clearance	Board to SMD		

Table 2-1. CADSTAR to PADS Rule Mapping

CADSTAR		PADS Layout		Comment / Value Map
	SMD Pad to Copper	Clearance	Copper to SMD Text to SMD	
	SMD Pad to SMD Pad	Clearance	SMD to SMD	
	Via to Board	Clearance	Board to Via	
	Via to Copper	Clearance	Copper to Via Text to Via	
	Via to SMD Pad	Clearance	SMD to Via	
	Via to Via	Clearance	Via to Via	
Assignments / Net Spacing	First Spacing Class / Second Spacing Class / Spacing	Clearance	All clearance values, except Body to Body and Drill to Drill	For each spacing class, net class with the same name is created in PADS Layout. Applies to Default rules from the (Default) layer (Unclassed) - (Unclassed) entry, All against Layer Conditional rules from layer reassignment (Unclassed) - (Unclassed) entries, Net Class rules from (Default) layer <SpcClass> - (Unclassed) entries, Net Class against Net Class Conditional rules from <SpcClass1> - <SpcClass2> entries
Assignments / Routes	Optimal Width Min Width Max Width	Clearance	Trace Width	Applies to Default rules from the default Net Route Code, Net rules from the (Default) layer entries, Nets against Layer. Conditional rules from layer reassignments entries.

Table 2-1. CADSTAR to PADS Rule Mapping

CADSTAR		PADS Layout		Comment / Value Map
Defaults / Routes	Routing Layers	Routing	Selected layers	Applies to Default rules.
	Via Code	Routing	Selected vias	
Net attributes	net_max_length	High Speed	Maximum Length	Applies to Net rules.
	net_max_delay	High Speed	Maximum Delay	
	net_max_stub	High Speed	Maximum Stub length	
	net_shield	High Speed	Shielding	
	lengthen_size	High Speed	Parallelism Length	
	lengthen_space	High Speed	Parallelism Gap	
	net_diff_pair	Differential Pairs	Gap	

CADSTAR to PADS Layer Type Mapping

Table 2-2 maps the translation of CADSTAR design layer types to PADS Layout.

Tip: To ensure that CADSTAR non-electrical layers are properly translated, make certain that their layer subtypes are specified in the CADSTAR file.

Table 2-2. CADSTAR to PADS Layer Type Mapping

CADSTAR Layer Type	PADS Layout Layer Type
Electrical Layer	Electrical Layer (No Plane)
Powerplane Layer	Electrical Layer (CAM Plane). If a CADSTAR design contains a Split Power Plane object on this layer, then the translated layer becomes Split/Mixed plane.
Non-Electrical Layer Paste subtype	Non-electrical Layer of type Paste Mask, the Top or Bottom electrical layers should be associated with this layer.
Non-Electrical Layer Silkscreen subtype	Non-electrical Layer of type Silk-Screen, the Top or Bottom electrical layers should be associated with this layer.
Non-Electrical Layer Solder Resist subtype	Non-electrical Layer of type Solder Mask, the Top or Bottom electrical layers should be associated with this layer.
Non-Electrical Layer Assembly subtype	Non-electrical Layer of type Assembly, the Top or Bottom electrical layers should be associated with this layer.
Non-Electrical Layer Placement subtype	Non-electrical Layer 20 (or 120 for Max Layers). General type.
Non-Electrical Layer Other subtypes	Non-Electrical Layer of General type.
Documentation Layer	Non-Electrical Layer of General type.
Construction Layer	Substrate or Prepreg Thickness and Dielectric (Layer Thickness dialog box).

CADSTAR to PADS Areas and Coppers Mapping

Table 2-3 maps the translation of CADSTAR Areas and Coppers to PADS Layout.

Table 2-3. CADSTAR to PADS Areas and Coppers Mapping

CADSTAR	PADS Layout	Comment
Closed Figure filled with the Clear pattern	Closed polygon drawing shape on non-electrical layer	
Closed Figure filled with the Solid pattern	Fixed (non-poured) copper outline on non-electrical layer	
Closed Figure filled with the Hatching pattern		Flags as a warning in the log file
Cutout within a shape filled with the Clear pattern	Closed polygon drawing shape on non-electrical layer	
Cutout within a shape filled with the Solid or Hatching pattern	Copper Cut Out area	
Unattached non-closed Copper	Open copper path that is not filled	
Unattached Copper filled with the Clear pattern		
Unattached Copper filled with the Solid pattern	Fixed (non-poured) copper outline on electrical layer	
Unattached Copper filled with the Hatching pattern		Flags as a warning in the log file
Cutout within an unattached Copper	Copper Cut Out area	
Template	Copper pour area outline with zero hatch spacing	The Template Fill Style is ignored without a warning
Copper attached to a Template filled with the Clear pattern	Poured copper outline with the maximum allowed hatch spacing	Flags as a warning in the log file
Copper attached to a Template filled with the Solid pattern	Poured copper outline with zero hatch spacing	
Copper attached to a Template filled with the Hatching pattern	Poured copper outline	

Table 2-3. CADSTAR to PADS Areas and Coppers Mapping

CADSTAR	PADS Layout	Comment
Keepout Area	Keepout with the Placement restrictions	The Area Fill Style and Line Style are not used in PADS Layout and ignored without a warning
Placement Area with the Height specified	Keepout with the Placement restrictions and specified Component Height	
No Routes Area	Keepout with the Trace and Copper restrictions	
No Vias Area	Keepout with the Via and Jumper restrictions	
Routing Area or Placement Area with no Height specified	<not translated>	Flags as a warning in the log file
Cutout within an Area	<not translated>	Flags as a warning in the log file

Chapter 3

Translating OrCAD Designs and Libraries

Supported File Types

The PADS Layout translator converts the following types of OrCAD design and library files:

- OrCAD Layout design files in binary (*.max) and ASCII formats (*.min) generated in OrCAD Layout 9.X, 10.X.
- OrCAD PCB libraries in the binary (*.llb) and ASCII formats (*.min) generated in OrCAD Layout 9.X, 10.X.

Translation Notes

The following sections describe how differences between OrCAD and PADS Layout are handled by the translator.

Library Translation Notes

PADS Layout Library Structure

A PADS Layout library consists of 4 files:

- Part Type (.pt9)
- PCB decal (.pd9)
- CAE Decal (.ld9)
- 2D Lines (.ln9)

The translator automatically creates these files from the single OrCAD source file. The output Part Type and PCB Decal files contain data; the CAE Decal and 2D Lines files do not contain data—these files are generated by the PADS Schematic Translator.

Design Translation Notes

Output File Names

The translator uses the original OrCAD file name as the name of the newly-translated PADS Layout file.

Design Rules Translation

See [OrCAD To PADS Design Rule Mapping](#) for details of the mapping of OrCAD design rules to PADS Layout.

Layers Translation

The translator gives you the option of preserving the original OrCAD layer names or changing them to the corresponding default PADS Layout layer names.

See [OrCAD to PADS Layer Mapping](#) for details of the mapping of OrCAD design layers to PADS Layout.

Obstacles Translation

See [OrCAD to PADS Obstacle Mapping](#) for details of the mapping of OrCAD obstacles to PADS Layout.

Track Translation

If a track object has no assigned nets in OrCAD, it is translated as a 2-d line object in PADS Layout ; otherwise, it becomes a trace in PADS Layout.

Cross Hatch Patterns

OrCAD Line Hatch patterns are translated to PADS Layout as Cross Hatching. Hatch angle patterns are translated to either orthogonal or diagonal patterns.

Individual Character Rotations

The translator splits texts having defined individual character rotations into multiple different-orientated single-character text objects.

Translating Designs and Libraries

Perform the following procedure to translate OrCAD designs and libraries to PADS Layout.

1. **Start menu > Programs > Mentor Graphics SDD > PADS (version) > Translators > PADS Layout Translator**
2. Select the **OrCAD** tab.
3. Populate the Files to translate file list:
 - a. In the Files to translate area, select the appropriate check box(es) to specify whether to translate designs, libraries, or both. Then click **Add**.
 - b. In the Select Input Design(s) dialog box, browse to and select the source file(s) you want to make available for translation. Then click **Open**. The selected files are added to the Files to translate file list. Files newly added to the list are checked (enabled for translation) by default.

To remove a file from the list, select it and click **Remove**.
4. Set the output location for translated files:
 - a. In the Designs box, accept the default output location for translated design files, or click the ... button to browse to and specify a different one.
 - b. In the Libraries box, accept the default output location for translated library files, or click the ... button to browse to and specify a different one.

Tips:

- The default values are from the current FileDir (designs) and LibDir (libraries) entries in the PADS Layout powerpcb.ini file.
 - If the output file is open in PADS Layout, the file is not translated and a message is written to the output log.
5. In the Overwrite existing files drop-down list, select one of the following to specify how the translator should proceed if it needs to overwrite an existing file:
 - No**—Leave the existing file intact; the source file is not translated.
 - Yes**—Overwrite the existing file.
 - Ask**—Display a prompt asking whether to overwrite the existing file.
 6. In the Translation options area, select the OrCAD tab, and then select the check boxes to set translation options.
 - Select the **Preserve net colors** check box to preserve the colors of OrCAD nets for corresponding nets in PADS Layout.

Tip: Because PADS Layout net color applies to nets, pads, vias, and coppers, while OrCAD net color applies only to unrouted, the translated design colors may not match the original design colors.

- Select the **Rename layers** check box to rename OrCAD layer names to PADS Layout names. See [OrCAD to PADS Layer Mapping](#) for information on OrCAD to PADS layer name mapping.

Tip: If an OrCAD layer has a non-default name, the translator preserves the non-default name in the translated file even if the Rename layers check box is selected.

- **Select the Flood copper pours check box to flood all copper pours on the translated board. If this check box is not selected, only copper pour outlines and cutouts are preserved.**
 - Select the **Explode doc symbols** check box to allow exploding of OrCAD document symbols to collections of line, arc and text objects. If the box is unchecked, document symbols are translated as component symbols in PADS Layout.
7. In the PADS Layout Translator dialog, click **Translate**. The displayed Translation Progress indicator shows both current file and overall job progress.

When translation is complete, the Translation Results dialog box displays the content of a log file created during translation. The log file contains the following information:

- Path of the translated files
- Number of errors and warnings
- Detailed description of each error or warning

Combining Layout and Schematic Libraries into a Single PADS Library

You may want to have a single PADS library that includes all the Part Type, PCB Footprint, and CAE Decal data. To create a single PADS library, perform the following procedure.

1. Using the Layout translator, translate the OrCAD library.
2. Using the Schematic Translator, clear the Overwrite existing files check box, and translate the OrCAD library. Save the schematic library to the same directory where you saved the library you translated in Layout.

Result: The Schematic translator does not overwrite Part Types created by the Layout translator, and it creates records in the schematic symbol library file (.ld9).

OrCAD To PADS Design Rule Mapping

Table 3-1 maps the translation of design rules from OrCAD to PADS Layout.

Table 3-1. OrCAD Rule Mapping

OrCAD		PADS Layout		Comment/Value map
Category	Name	Category	Name	
Route Spacing	Track to Track	Clearance	Trace to Trace	Applies to Default rules with most common value and All against layer conditional rules with layer specific values.
	Track to Via	Clearance	Via to Trace	
	Track to Pad	Clearance	Pad to Trace Drill to Trace SMD to Trace	
	Via to Via	Clearance	Via to Via	
	Via to Pad	Clearance	Pad to Via Drill to Via SMD to Via	
	Pad to Pad	Clearance	Pad to Pad Drill to Pad Drill to SMD SMD to SMD SMD to Pad	
Net Properties	Min Width Conn Width Max Width	Clearance	Trace Width	Applies to Default rules with most common value and Net rules with net specific values.
	Net Width by Layer	Clearance	Trace Width Recommended	Applies to Nets against Layer Conditional rules.
	Net Spacing by Layer	Clearance	Trace to Trace	Applies to Nets against Layer Conditional rules.
	Layers Enabled for Routing	Routing	Selected layers	Applies to Net rules.
	Weight	Routing	Priority	
	Routing Enabled	Routing	Auto route	
	Retry Enabled	Routing	Allow ripup	
Shove Enabled	Routing	Allow shove		

Table 3-1. OrCAD Rule Mapping

OrCAD		PADS Layout		Comment/Value map
Layer Strategy / Sweep Win/Comp	Routing Enabled	Routing	Selected layers	Applies to Default rules.
	Primary Direction	Layers Setup	Routing Direction	
Fanout Settings	Fanout power/gnd	Fanout	Nets/Plane	
	Fanout signals	Fanout	Nets / Signal	
	IC fanout direction	Fanout	Direction	
	Maximum fanout distance	Fanout	Fanout length / Maximum	

OrCAD to PADS Layer Mapping

The translator preserves the original OrCAD unique layer name unless the "Rename layers" check box is turned on. If this option is checked, the translator renames the default OrCAD layer nickname (three-letter abbreviation for the layer name) to the corresponding default PADS Layout layer name as shown in [Table 3-2](#).

Notes:

- Routing inner layers (IN1 through IN9, I10, I11, I12) are translated as additional nonplane electrical layers.
- Plane layers are translated as Split/Mixed electrical layers,
- Documentation layers are translated as additional General nonelectrical layers.

Table 3-2. OrCAD to PADS Layer Renaming

OrCAD Name	PADS Layout Name
TOP	Top
BOT	Bottom
PWR	Power Plane
GND	Ground Plane
SST	Silkscreen Top
SSB	Silkscreen Bottom
AST	Assembly Drawing Top
ASB	Assembly Drawing Bottom
SPT	Paste Mask Top
SPB	Paste Mask Bottom
SMT	Solder Mask Top
SMB	Solder Mask Bottom
DRD	Drill Drawing

OrCAD to PADS Obstacle Mapping

Table 3-3 maps the translation of OrCAD Obstacles to PADS Layout.

Table 3-3. OrCAD to PADS Obstacle Mapping

OrCAD	PADS Layout	Comment
Free track	Open copper path that is not filled	
Solid Copper area	Fixed (non-poured) copper outline	
Copper area filled with hatched lines		Flags as a warning in the log-file
Anti-copper	Copper Pour Cut Out	
Board outline	Board outline	
Via keepout	Keepout with the Via and Jumper restrictions	
Route-via keepout	Keepout with the Trace, Copper, Via and Jumper restrictions	
Route keepout	Keepout with the Trace and Copper restrictions	
Detail	Opened polygon drawing shape	
Comp height keepin	<nothing>	Flags as a warning in the log-file
Comp height keepout	Keepout with the Placement restrictions and specified Component Height	
Comp group keepin	<nothing>	Flags as a warning in the log-file
Comp group keepout	<nothing>	Flags as a warning in the log-file
Place outline	<nothing>	
Insertion outline	<nothing>	
Copper pour	Copper pour area outline	

Chapter 4

Translating P-CAD Designs and Libraries

Supported File Types

The PADS Layout translator converts the following types of P-CAD design and library files:

- P-CAD design files in binary and ASCII formats (*.pcb) generated in P-CAD 2001, 2002, 2004, 2006.
- P-CAD PCB libraries in the binary format (*.lib) generated in P-CAD 2001, 2002, 2004, 2006.

Translation Notes

The following sections describe how differences between P-CAD and PADS Layout are handled by the translator.

Library Translation Notes

PADS Layout Library Structure

A PADS Layout library consists of 4 files:

- Part Type (.pt9)
- PCB decal (.pd9)
- CAE Decal (.ld9)
- 2D Lines (.ln9)

The translator automatically creates these files from the single P-CAD source file. The output Part Type and PCB Decal files contain data; the CAE Decal and 2D Lines files do not contain data—these files are generated by the PADS Schematic Translator.

Design Translation Notes

Output File Names

The translator preserves the original file name while translating the file into PADS Layout unless the original file extension is .pcb. If the original file extension is .pcb, then the translator adds "_pads" to the original file name to get a translated design file name.

Design Rules Translation

See [P-CAD Object Type to PADS Rule Owner Mapping](#) for details of the mapping of P-CAD design rule owners to PADS Layout.

See [P-CAD to PADS Rule Mapping](#) for details of the mapping of P-CAD design rules to PADS Layout.

Layers Translation

Default Layers Translation

The translator gives you the option to preserve the original P-CAD layer names, or to change them to the corresponding default PADS Layout layer names.

See [P-CAD to PADS Default Layer Mapping](#) for details of the default mapping of P-CAD design layers to PADS Layout.

Customized Layers Translation

You can also customize layer mappings for individual layers by specifying the P-CAD layer name and the PADS Layout layer number.

Pads Translation

- P-CAD Ellipse, Target, and Mounting Hole pads are translated as circle/oval shaped pads in PADS Layout.
- P-CAD Thermal and NoConnect pads are translated as corresponding Thermal and Antipad pad shapes.
- P-CAD 8-sided regular polygon pads are translated as chamfered rectangle pads in PADS Layout.
- P-CAD 4-sided regular polygon pads are translated as square pads in PADS Layout.

Component Outline Color

Component outlines may be colored differently when loaded in PADS Layout and PADS Router. PADS Router uses the Top Layer color to display the component outline, but PADS Layout uses the color of the layer where the component outline was originally placed.

Self-intersecting Copper Pours

Some copper pours that are self-intersecting, or that have areas too small for hatching, cannot be flooded/hatched by PADS Layout. The translator detects these coppers, flags them in the log-file as an error, and translates them as a 2D-line object in PADS Layout.

Flooding/Hatching

Translated copper pour and plane area objects are not automatically hatched when you load the translated design in PADS Layout. They are automatically hatched when you load the translated design in PADS Router.

Line keepouts

A P-CAD widthless Line Keepout is translated as a closed filled polygon keepout 1 mil wide in PADS Layout.

Free pads

Free pads in the P-CAD design are translated as dummy components with no geometry and a single pad stack in PADS Layout.

Copper Ties

P-CAD Copper Ties electrically connect two or more nets while maintaining the ability to route them separately. These Copper Ties are translated in PADS Layout as regular pieces of drafting copper with no net.

Board Outline and Board Cutouts

- P-CAD board cutouts are translated as 2-D lines in PADS Layout.
- Board outlines created by the P-CAD 2006 Place Board Outline tool are unsupported and not translated.

Locked library file

When you open a library in the P-CAD Pattern Editor or add a library into the Library Setup dialog list in the P-CAD PCB and have P-CAD PCB open, P-CAD locks that library file; therefore, the translator cannot even open this file not to mention about translating. Usually, the

translator indicates that the library is locked by displaying the exclamation mark icon in the files list. If you see this, close the P-CAD Pattern Editor or PCB application or remove the library from the Library Setup dialog.

Room attributes

Since PADS Layout does not support P-CAD Rooms, room attributes and rules are translated as component attributes for all components included in the room.

Multiple Stroke Fonts

PADS Layout does not support multiple stroke fonts. Multiple stroke font families in the original P-CAD file are all converted to the Romansim stroke font family in PADS Layout.

P-CAD System (TrueType) fonts are translated as corresponding System fonts in PADS Layout.

Multiple Symbol Representations

P-CAD supports multiple graphical representations (Patterns) of a pcb symbol; PADS Layout does not. Multiple pattern graphics for a single P-CAD pcb symbol are translated into PADS Layout as multiple symbols, each identified by its corresponding P-CAD pattern graphic's name suffix.

Tip: The default P-CAD pattern graphic (usually "Primary") is named without the suffix.

Multi-line Labels and Attributes

A multiline free text object from a P-CAD design becomes multiple single-line free text objects in PADS Layout. A multiline attribute from a P-CAD design is translated as a single-line label or attribute to PADS Layout with concatenating multiple lines to single line.

Translating Designs and Libraries

1. **Start menu > Programs > Mentor Graphics SDD > PADS (version) > Translators > PADS Layout Translator**
2. Select the **P-CAD** tab.
3. Populate the file list:
 - a. In the Files to translate area, select the appropriate check box(es) to specify whether to translate designs, libraries, or both. Then click **Add**.
 - b. In the Select Input Design(s) dialog box, browse to and select the source file(s) you want to make available for translation. Then click **Open**. The selected files are added

to the Files to translate file list. Files newly added to the list are checked (enabled for translation) by default.

To remove a file from the list, select it and click **Remove**.

4. Set the output location for translated files:
 - a. In the Designs box, accept the default output location for translated design files, or click the ... button to browse to and specify a different one.
 - b. In the Libraries box, accept the default output location for translated library files, or click the ... button to browse to and specify a different one.

Tips:

- The default values are from the current FileDir (designs) and LibDir (libraries) entries in the PADS Layout powerpcb.ini file.
 - If the output file is open in PADS Layout, the file is not translated and a message is written to the output log.
5. In the Overwrite existing files drop-down list, select one of the following to specify how the translator should proceed if it needs to overwrite an existing file:
 - No**—Leave the existing file intact; the source file is not translated.
 - Yes**—Overwrite the existing file.
 - Ask**—Display a prompt asking whether to overwrite the existing file.
 6. In the Translation options area, select the P-CAD tab, and then select the check boxes to set translation options.
 - Select the **Preserve net colors** check box to preserve the colors of P-CAD nets for corresponding nets in PADS Layout.
 - Select the **Rename layers** check box to rename P-CAD layer names to PADS Layout names. See [P-CAD to PADS Default Layer Mapping](#) for information on P-CAD to PADS layer name mapping.
 7. **(Optional)** Click the Layer Mapping button to customize the layer mapping. In the P-CAD Layer Mapping dialog box, for each layer whose mapping you want to customize:
 - a. Enter the P-CAD layer **name**.
 - b. Enter the PADS Layout layer **number**.
 - c. When you are finished adding layers, Click **OK**.
 8. **(Optional)** Click the **Attributes Mapping** button to display the Attributes Mapping dialog, where you can add, delete and modify P-CAD component and net attribute names, and map them to PADS Layout equivalents. (The translator uses the attributes

mapping while translating part type, decal, component, net class, and pin attributes. The mapping is preserved in the translator's configuration ini-file.)

Tips:

- Attributes that are not listed are translated using the original name from the source design; they have an attribute type of Free Text.
- If you do not want to translate an attribute, select the **Do not translate** check box.
- If the name of a file type is case sensitive, select the **Case sensitive** check box.
- To add a new item to the attributes map:
 - a. Select the Component Attributes or Net Attributes tab.
 - b. Type the P-CAD attribute name in the **Name** box (or select it from the list directly below the box.)
 - c. Select the corresponding PADS attribute name in the **Map to** box.
 - d. Select the type of the attribute in the **Type** combo box and click the **Add** button.

Tip: To add an item with a new unit type, select **Measure** in the Type combo box, type the new unit name in the **Quantity** combo box, click the **Add** button and specify a unit abbreviation (for example, m or F) and a short unit name (for example, meter or Farad) in the New Unit dialog box. Then click the **Add** button again.

When you are finished adding, deleting and modifying items, Click **OK** to close the Attributes Mapping dialog.

9. In the PADS Layout Translator dialog, click **Translate**. The displayed Translation Progress indicator shows both current file and overall job progress.

When translation is complete, the Translation Results dialog box displays the content of a log file created during translation. The log file contains the following information:

- Path of the translated files
- Number of errors and warnings
- Detailed description of each error or warning

Combining Layout and Schematic Libraries into a Single PADS Library

You may want to have a single PADS library that includes all the Part Type, PCB Footprint, and CAE Decal data. To create a single PADS library, you will use both the Layout and Schematic Translators, in the order described in the following procedures.

To create a single PADS library:

1. Using this Translator, translate the P-CAD library.
2. Using the Schematic Translator, clear the Overwrite existing files check box, and translate the P-CAD library.
3. Save the schematic library to the same directory where you saved the library you translated in Layout.

Result: The Schematic translator does not overwrite Part Types created by the Layout translator, and it creates records in the schematic symbol library file (.ld9).

P-CAD to PADS Rule Mapping

Table 4-1 maps rule translation from P-CAD to PADS Layout. Any P-CAD rules not listed in the table are translated as attributes of the corresponding object.

Table 4-1. P-CAD to PADS Layout Rule Mapping

P-CAD		PADS Layout		Comment / Value map
Category	Name	Category	Name	
Clearance	LineToLineClearance	Clearance	Trace to Trace Copper to Trace Text to Trace	Applies to: Default rules from Design attributes, Net rules from Net attributes, Net Class rules from Net Class attributes, Classes against Classes Conditional rules from Class To Class attributes, All against Layer Conditional rules from Layer attributes
Clearance	ViaToLineClearance	Clearance	Via to Trace Copper to Via Text to Via	
Clearance	ViaToViaClearance	Clearance	Via to Via	
Clearance	PadToLineClearance	Clearance	Pad to Trace SMD to Trace Copper to Pad Copper to SMD Drill to Track Text to Pad	
Clearance	ViaToPadClearance	Clearance	Pad to Via SMD to Via Drill to Via	
Clearance	PadToPadClearance	Clearance	Pad to Pad SMD to Pad SMD to SMD Drill to Pad	
Clearance	Clearance	Clearance	Default clearance value	
Physical	Width	Clearance	Trace Width	
Clearance	BoardEdgeClearance	Clearance	Board to Trace Board to SMD Board to Via Board to Pad	Applies to: Default rules from Design attributes, Net rules from Net attributes, Net Class rules from Net Class attributes
Clearance	HoleToHoleClearance	Clearance	Drill to Drill	Applies to: Default rules from Design attributes
Clearance	SilkscreenClearance	Clearance	Body to Body	

Table 4-1. P-CAD to PADS Layout Rule Mapping

P-CAD		PADS Layout		Comment / Value map
Category	Name	Category	Name	
Net	MaxNetLength	High Speed	Maximum Length	Applies to: <i>Default rules</i> from Design attributes, <i>Net rules</i> from Net attributes, <i>Net Class rules</i> from Net Class attributes
Net	MinNetLength	High Speed	Minimum Length	
Router	NoAutoRoute	Routing	Auto Route flag	
Router	Ripup	Routing	Allow Ripup flag	

P-CAD to PADS Default Layer Mapping

When P-CAD layers are translated, the original layer name is preserved unless you select the Rename layers check box. If you choose to rename layers, layers are renamed as shown in [Table 4-2](#).

Table 4-2. P-CAD to PADS Layout Layer Mapping

P-CAD Name	PADS Layout Name
Top	Top
Bottom	Bottom
Top Paste	Paste Mask Top
Bot Paste	Paste Mask Bottom
Top Mask	Solder Mask Top
Bot Mask	Solder Mask Bottom
Top Silk	Silkscreen Top
Bot Silk	Silkscreen Bottom
Top Assy	Assembly Drawing Top
Bot Assy	Assembly Drawing Bottom

Tips:

- Custom signal layers are translated as additional nonplane electrical layers.
- Custom nonsignal layers are translated as additional General nonelectrical layers.
- Plane layers are translated as Split/Mixed electrical layers.

P-CAD Object Type to PADS Rule Owner Mapping

Design rules are set per object; therefore, the objects to which rules apply must also map. PADS Layout does not support P-CAD rooms when they are specified as a rule scope in the source P-CAD design.

[Table 4-3](#) maps rule owner translation from P-CAD to PADS Layout.

Table 4-3. P-CAD Object Type to PADS Layout Rule Owner Mapping

P-CAD Type	PADS Layout Type
Design	Default
Net Class	Net Class
Net	Net
Class to Class	Classes against Classes
Layer	All against Layer

Tip: If layers are specified in the P-CAD rule scope, then the rule translates to PADS Layout as a conditional rule.

Chapter 5

Translating Altium/Protel Designs and Libraries

Supported File Types

The PADS Layout translator converts the following types of Altium/Protel design and library files:

- Protel 99 design files (*.pcb) in the binary format as well as in the ASCII format, including designs stored in Protel Design Database (*.ddb) project files.
- Protel 99 PCB library files (*.lib), including PCB libraries stored in Protel Design Database (*.ddb) project files.
- Altium DXP/2004/2006/Altium Designer up to version 7.1 (Summer08) design files (*.pcbdoc) in the binary format as well as in the ASCII format.
- Altium DXP/2004/2006/Altium Designer up to version 7.1 (Summer08) PCB library files (*.pcbplib), including libraries stored in Integrated Libraries (*.intl).lib).

Translation Notes

The following sections describe how differences between Altium/Protel and PADS Layout are handled by the translator.

Library Translation Notes

PADS Layout Library Structure

A PADS Layout library consists of 4 files:

- Part Type (.pt9)
- PCB decal (.pd9)
- CAE Decal (.ld9)
- 2D Lines (.ln9)

The Layout translator automatically creates these files from the single Altium/Protel source file. The output Part Type and PCB Decal files contain data; the CAE Decal and 2D Lines files do not contain data—these files are generated by the PADS Schematic Translator.

Design Translation Notes

Output File Names

The translator preserves the original file name while translating a file into PADS Layout unless the original file extension is .pcb. If the original file extension is .pcb, the translator adds "_pads" to the original file name to get a translated design file name.

If an Altium/Protel source file is stored in a MS Access database, the translator uses the stored MS Access name as the name for the resulting PADS Layout design or library.

Padstack Translation

- Altium/Protel octagonal pads are translated as chamfered rectangle pads to PADS Layout.
- Square holes are not supported in PADS. They are translated as round holes with diameter equal to the square hole size.

Component Outline Color

Component outlines may be colored differently when loaded in PADS Layout and PADS Router. PADS Router uses the Top Layer color to display the component outline, but PADS Layout uses the color of the layer where the component outline was originally placed.

Part Types

Since Altium/Protel PCB does not have the concept of Part Types, the translator creates a PADS Layout part type object for each PCB Decal used in a design or a library. If the schematic library reference is specified in Protel DXP, it is used as the part type name; otherwise the original PCB Decal name is used, unless the "Use Comment attribute as part type name" check box is selected.

When the "Use Comment attribute as part type name" check box is checked the translator determines whether that attribute value is appropriate to be used as a part type name. If the comment attribute value has any of the following characteristics, it will not be used as a part type name:

- It is a floating-point number with an appended value unit name ("mils", "Hz" etc.)
- It contains white-spaces or lower-case characters.
- It is too long or too short.
- It has any other attribute inconsistent with what the PADS product might use as a part type name.

Tip: The "Use Comment attribute as part type name" checkbox does not apply to the translation of libraries. There are no attributes in libraries.

Track Translation

If a track object has no assigned nets in Altium/Protel, it is translated as a 2-d line object in PADS Layout ; otherwise, it becomes a trace in PADS Layout.

Self-intersecting Copper Pours

Some copper pours that are self-intersecting, or that have areas too small for hatching, cannot be flooded/hatched by PADS Layout. The translator detects these coppers, flags them in the log-file as an error, and translates them as a 2D-line object in PADS Layout.

Flooding/Hatching

Translated copper pour and plane area objects are not automatically hatched when you load the translated design in PADS Layout. They are automatically hatched when you load the translated design in PADS Router.

PADS Layout does not support Altium/Protel vertical and horizontal hatching styles; they are translated as pane hatching.

Solid regions

Solid regions in the Altium/Protel design are translated as solid coppers in PADS Layout.

Free pads

Free pads in the Altium/Protel design are translated as dummy components with no geometry and a single pad stack in PADS Layout.

Multiple Stroke Fonts

PADS Layout supports only one stroke font. All fonts in the original Altium file are converted to the single PADS Layout font.

Altium Designer Translation Restrictions

The following items are not translated when translating Altium designs in any format:

- Embedded Board Arrays (board panels), which don't exist in PADS Layout
- Solid regions defined as Board cutouts
- Solid regions defined as Polygon cutouts

- Solid regions defined as Keepouts

The following restrictions apply when translating Altium designs in ASCII format only:

- Only the Simple and Top-Middle-Bottom padstack modes are fully supported. “Full stack mode” (defining pads for individual layers) is translated as Top-Inner-Bottom layer padstacks. (The Inner pad is taken from the first inner layer.)
- The following items are not translated:
 - Slotted and square holes
 - Rounded rectangle pads
 - Solid region objects

Translating Designs and Libraries

Perform the following procedure to translate Altium/Protel designs and libraries to PADS Layout.

1. **Start menu > Programs > Mentor Graphics SDD > PADS (version) > Translators > PADS Layout Translator**
2. Select the **Protel/Altium** tab.
3. Populate the Files to translate file list:
 - a. In the Files to translate area, select the appropriate check box(es) to specify whether to translate designs, libraries, or both. Then click **Add**.
 - b. In the Select Input Design(s) dialog box, browse to and select the source file(s) you want to make available for translation. Then click **Open**. The selected files are added to the Files to translate file list. Files newly added to the list are checked (enabled for translation) by default.

Tip: Since Protel 99 project files and Altium DXP integrated libraries may use an MS Access database as file storage, the file list may be represented as a two level hierarchy tree, where the top level is a file path to the source Altium/Protel DDB file, and the second level is a list of .pcb and .lib files stored in this DDB or IntLib file.

To remove a file from the list, select it and click **Remove**.

4. Set the output location for translated files:
 - a. In the Designs box, accept the default output location for translated design files, or click the ... button to browse to and specify a different one.
 - b. In the Libraries box, accept the default output location for translated library files, or click the ... button to browse to and specify a different one.

Tips:

- The default values are from the current FileDir (designs) and LibDir (libraries) entries in the PADS Layout powerpcb.ini file.
 - If the output file is open in PADS Layout, the file is not translated and a message is written to the output log.
5. In the Overwrite existing files drop-down list, select one of the following to specify how the translator should proceed if it needs to overwrite an existing file:

No—Leave the existing file intact; the source file is not translated.

Yes—Overwrite the existing file.

Ask—Display a prompt asking whether to overwrite the existing file.

6. In the Translation options area, select the Protel tab, and then select the check boxes to set translation options.
 - Select the **Preserve net colors** check box to preserve the colors of Protel nets for corresponding nets in PADS Layout.

Tip:The colors in the translated design may not match the original design colors because, while Protel net color applies only to unrouted, PADS Layout net color applies to nets, pads, vias, and coppers.
 - Select the **Rename layers** check box to rename Protel layer names to PADS Layout names. See [Altium/Protel to PADS Layer Type and Name Mapping](#) for information on Protel to PADS layer name mapping.

Tip:If an Altium/Protel layer has a non-default name, the translator preserves the non-default name in the translated file even if the Rename layers check box is selected.
 - Select the **Merge adjacent coppers** check box to enable merging of multiple adjacent coppers in the Protel source design into a single copper in the resulting PADS Layout design.

Tip: This feature increases PADS Layout graphics performance.
7. In the PADS Layout Translator dialog, click **Translate**. The displayed Translation Progress indicator shows both current file and overall job progress.

When translation is complete, the Translation Results dialog box displays the content of a log file created during translation. The log file contains the following information:

- Path of the translated files
- Number of errors and warnings
- Detailed description of each error or warning

Combining Layout and Schematic Libraries into a Single PADS Library

You may want to have a single PADS library that includes all the Part Type, PCB Footprint, and CAE Decal data. To create a single PADS library, you will use both the Layout and Schematic Translators, in the order described in the following procedures.

To create a single PADS library:

1. Because Altium/Protel has separate files for schematic and library files, first use this Translator to translate the PCB library.
2. Using the Schematic Translator, translate the schematic library, following the procedure in Schematic help topic "Translating Libraries".

Tip: The Translator places translations of the two types of files in separate folders.

3. Copy the resulting PCB library file (.pd9) to the folder into which you saved the translated schematic library.
4. Because part types are translated by the Schematic translator, and the Layout translator generates part type data based on the PCB decal names, verify that the PCB decals are assigned to the part types correctly.

Result: The Altium/Protel PCB library typically does not include part type information, so the Layout translator creates part types based on PCB decal names.

Altium/Protel to PADS Rule Owner Mapping

Design rules are set per object; therefore, the objects to which rules apply must also map. PADS Layout does not support all Altium/Protel object types.

Table 5-1. Owner Mapping

Altium/Protel Type	PADS Layout Type
Whole board	Default
Footprint	Decal
Component	Component
Net Class	Net Class
Net	Net
From-to Class	Pin Pair Group
From-to	Pin Pair

Object Types Specified as Rule Scope

The translator does not support the following object types when they are specified as a rule scope in the Altium/Protel design:

- pad class
- component class
- via specification
- pad specification
- footprint-pad
- pad

Tips:

- If layers are specified in the Protel rule scope, then the rule translates as a conditional rule.
- The translator ignores disabled Protel rules.

Altium/Protel to PADS Rule Mapping

The following table maps rules translation from Altium/Protel to PADS Layout. Any Altium/Protel rules not listed in the table are not translated because no equivalent exists in PADS Layout. Any rules that are not translated are noted in the .log file.

Table 5-2. Rule Mapping

Altium/Protel		PADS Layout		Notes/Value Map
Category	Name	Category	Name	
Routing	Width Constraint	Clearance rule	Trace Width	
Routing	Clearance Constraint	Clearance rule		The exact PADS Layout rule depends on the object the rule is assigned to in Altium/Protel (rule scope).
Routing	Routing Layers	1. Layers setup dialog box settings 2. Routing rule	1. Routing direction 2. Layer Biasing	1. Protel "o'clock" values are mapped to "Any" in PADS Layout. Routing directions are supported in PADS Router. 2. If Protel type is "Not used" or "Fanout," the layer is removed from Selected Layers (Setup > Design Rules > Default > Routing).
Routing	Routing Corners	Design Preferences dialog box settings	Miters	45 degree = Diagonal Rounded = Arc
Routing	Routing Priority	Routing rule	Priority	

Table 5-2. Rule Mapping

Altium/Protel		PADS Layout		Notes/Value Map
Routing	Routing Topology	Routing rule	Topology type	Shortest, Starburst, Horizontal, Vertical = Minimized Daisy-Simple = Serial Source Daisy- MidDriven = Mid-driven Daisy- Balanced = Parallel source
Routing	Routing VIA style	Routing rule	Selected vias	
Routing	SMD to Corner Constraint	Clearance rule	Same Net/SMD to Corner	
Manufacturing	Polygon Connect Style Power Plane Connect Style	Clearance rule	Copper to via, pad, SMD	This rule is the thermal pad parameters in PADS Layout.
Manufacturing	Power Plane Clearance	Clearance	Copper to via, pad, SMD	
Manufacturing	Test point style	DFT Audit dialog box settings	Minimum pad probing size	Top = PCB Top Side Testpoint grid size = Test Point grid
High Speed	Daisy Chain Stub Length	High Speed rule	Stub Length	
High Speed	Length Constraint	High Speed rule	Length	
High Speed	Matched Length	High Speed rule	Match Length	Tolerance = Tolerance
High Speed	Parallel Segment	High Speed rule	Parallelism	Same Layer = Parallelism Adjacent Layers = Tandem
High Speed	Vias under SMD	Pad Entry rule	Via at SMD	

Table 5-2. Rule Mapping

Altium/Protel		PADS Layout		Notes/Value Map
Signal Integrity	Flight Time	High Speed rule	Delay	Min = 0 Max = Min (Flight Time 1, Flight Time 2)
Signal Integrity	Impedance Constraint	High Speed rule	Impedance	
Signal Integrity	Supply Nets	Attributes	PowerGround and Voltage attributes	This Protel rule is translated as PADS Layout attributes assigned to nets and net classes specified in the Protel rule scope.
Placement	Component Clearance	Clearance	Body to Body	Applies to Default rules only.

Altium/Protel to PADS Layer Type and Name Mapping

When Altium/Protel layers are translated, the original layer name is preserved unless you select the Rename layers check box. If you choose to rename layers, layers are renamed as shown in [Table 5-3](#).

Table 5-3. Altium/Protel to PADS Layer Renaming

Altium/Protel Name	PADS Layout Name
Top Layer	Top
Bottom Layer	Bottom
Top Paste	Paste Mask Top
Bottom Paste	Paste Mask Bottom
Top Solder	Solder Mask Top
Bottom Solder	Solder Mask Bottom
Top Overlay	Silkscreen Top
Bottom Overlay	Silkscreen Bottom

Tip: If a default Altium/Protel layer name has been changed to a different name, the translator preserves the changed name even if the Rename layers check box is selected.

Layer types are also translated when you rename layers. Altium/Protel layer types are mapped to PADS Layout layer types as shown in [Table 5-4](#). PADS Layout layer types are set in the Layers Setup dialog box (Setup menu > Layer Definition).

Table 5-4. Altium/Protel to PADS Layer Type Mapping

Altium/Protel Type	PADS Layout Type	Notes
Signal layer	Electrical layer (No Plane)	
Internal plane	Electrical layer (CAM Plane)	If the Protel layer contains a split plane object, it is translated to a Split/Mixed Plane in PADS Layout.
Mechanical	Nonelectrical layer (General)	
Top Overlay	Nonelectrical layer (Silkscreen Top)	The top electrical layer is associated with this layer.
Bottom Overlay	Nonelectrical layer (Silkscreen Bottom)	The bottom electrical layer is associated with this layer.

Table 5-4. Altium/Protel to PADS Layer Type Mapping

Altium/Protel Type	PADS Layout Type	Notes
Top Solder	Nonelectrical layer (Solder Mask Top)	The top electrical layer is associated with this layer.
Bottom Solder	Nonelectrical layer (Solder Mask Bottom)	The bottom electrical layer is associated with this layer.
Top Paste	Nonelectrical layer (Paste Mask Top)	The top electrical layer is associated with this layer.
Bottom Paste	Nonelectrical layer (Paste Mask Bottom)	The bottom electrical layer is associated with this layer.
Drill Drawing	Nonelectrical layer (Drill Drawing)	
Drill Guide	Nonelectrical layer	
Keep Out	All layers (Layer 0)	2D lines from this Protel layer are applied to the board outline object on Layer 0, and to Keepout areas (<All Layers>) in PADS Layout.
Multi Layer	All layers (Layer 0)	
Top Dielectric Properties	Top coating properties (Thickness)	The properties are assigned as Thickness to the Top layer in PADS Layout.
Bottom Dielectric Properties	Bottom coating properties	The properties are assigned as Thickness to the Bottom layer in PADS Layout.

Third-Party Information

This section provides information on open source and third-party software that may be included in the PADS Layout Translator product.

- This software application may include BOOST version 1.46.0 third-party software. BOOST version 1.46.0 is distributed under the terms of the BOOST Software License version 1.0 and is distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND, either express or implied. See the license for the specific language governing rights and limitations under the license. You can view a copy of the license at: your_Mentor_Graphics_documentation_directory/legal/boost_1.0.pdf.

End-User License Agreement

The latest version of the End-User License Agreement is available on-line at:
www.mentor.com/eula

IMPORTANT INFORMATION

USE OF ALL SOFTWARE IS SUBJECT TO LICENSE RESTRICTIONS. CAREFULLY READ THIS LICENSE AGREEMENT BEFORE USING THE PRODUCTS. USE OF SOFTWARE INDICATES CUSTOMER'S COMPLETE AND UNCONDITIONAL ACCEPTANCE OF THE TERMS AND CONDITIONS SET FORTH IN THIS AGREEMENT. ANY ADDITIONAL OR DIFFERENT PURCHASE ORDER TERMS AND CONDITIONS SHALL NOT APPLY.

END-USER LICENSE AGREEMENT ("Agreement")

This is a legal agreement concerning the use of Software (as defined in Section 2) and hardware (collectively "Products") between the company acquiring the Products ("Customer"), and the Mentor Graphics entity that issued the corresponding quotation or, if no quotation was issued, the applicable local Mentor Graphics entity ("Mentor Graphics"). Except for license agreements related to the subject matter of this license agreement which are physically signed by Customer and an authorized representative of Mentor Graphics, this Agreement and the applicable quotation contain the parties' entire understanding relating to the subject matter and supersede all prior or contemporaneous agreements. If Customer does not agree to these terms and conditions, promptly return or, in the case of Software received electronically, certify destruction of Software and all accompanying items within five days after receipt of Software and receive a full refund of any license fee paid.

1. ORDERS, FEES AND PAYMENT.

- 1.1. To the extent Customer (or if agreed by Mentor Graphics, Customer's appointed third party buying agent) places and Mentor Graphics accepts purchase orders pursuant to this Agreement ("Order(s)"), each Order will constitute a contract between Customer and Mentor Graphics, which shall be governed solely and exclusively by the terms and conditions of this Agreement, any applicable addenda and the applicable quotation, whether or not these documents are referenced on the Order. Any additional or conflicting terms and conditions appearing on an Order will not be effective unless agreed in writing by an authorized representative of Customer and Mentor Graphics.
- 1.2. Amounts invoiced will be paid, in the currency specified on the applicable invoice, within 30 days from the date of such invoice. Any past due invoices will be subject to the imposition of interest charges in the amount of one and one-half percent per month or the applicable legal rate currently in effect, whichever is lower. Prices do not include freight, insurance, customs duties, taxes or other similar charges, which Mentor Graphics will state separately in the applicable invoice(s). Unless timely provided with a valid certificate of exemption or other evidence that items are not taxable, Mentor Graphics will invoice Customer for all applicable taxes including, but not limited to, VAT, GST, sales tax and service tax. Customer will make all payments free and clear of, and without reduction for, any withholding or other taxes; any such taxes imposed on payments by Customer hereunder will be Customer's sole responsibility. If Customer appoints a third party to place purchase orders and/or make payments on Customer's behalf, Customer shall be liable for payment under Orders placed by such third party in the event of default.
- 1.3. All Products are delivered FCA factory (Incoterms 2000), freight prepaid and invoiced to Customer, except Software delivered electronically, which shall be deemed delivered when made available to Customer for download. Mentor Graphics retains a security interest in all Products delivered under this Agreement, to secure payment of the purchase price of such Products, and Customer agrees to sign any documents that Mentor Graphics determines to be necessary or convenient for use in filing or perfecting such security interest. Mentor Graphics' delivery of Software by electronic means is subject to Customer's provision of both a primary and an alternate e-mail address.

2. **GRANT OF LICENSE.** The software installed, downloaded, or otherwise acquired by Customer under this Agreement, including any updates, modifications, revisions, copies, documentation and design data ("Software") are copyrighted, trade secret and confidential information of Mentor Graphics or its licensors, who maintain exclusive title to all Software and retain all rights not expressly granted by this Agreement. Mentor Graphics grants to Customer, subject to payment of applicable license fees, a nontransferable, nonexclusive license to use Software solely: (a) in machine-readable, object-code form (except as provided in Subsection 5.2); (b) for Customer's internal business purposes; (c) for the term of the license; and (d) on the computer hardware and at the site authorized by Mentor Graphics. A site is restricted to a one-half mile (800 meter) radius. Customer may have Software temporarily used by an employee for telecommuting purposes from locations other than a Customer office, such as the employee's residence, an airport or hotel, provided that such employee's primary place of employment is the site where the Software is authorized for use. Mentor Graphics' standard policies and programs, which vary depending on Software, license fees paid or services purchased, apply to the following: (a) relocation of Software; (b) use of Software, which may be limited, for example, to execution of a single session by a single user on the authorized hardware or for a restricted period of time (such limitations may be technically implemented through the use of authorization codes or similar devices); and (c) support services provided, including eligibility to receive telephone support, updates, modifications, and revisions. For the avoidance of doubt, if Customer requests any change or enhancement to Software, whether in the course of receiving support or consulting services, evaluating Software, performing beta testing or otherwise, any inventions, product

improvements, modifications or developments made by Mentor Graphics (at Mentor Graphics' sole discretion) will be the exclusive property of Mentor Graphics.

3. **ESC SOFTWARE.** If Customer purchases a license to use development or prototyping tools of Mentor Graphics' Embedded Software Channel ("ESC"), Mentor Graphics grants to Customer a nontransferable, nonexclusive license to reproduce and distribute executable files created using ESC compilers, including the ESC run-time libraries distributed with ESC C and C++ compiler Software that are linked into a composite program as an integral part of Customer's compiled computer program, provided that Customer distributes these files only in conjunction with Customer's compiled computer program. Mentor Graphics does NOT grant Customer any right to duplicate, incorporate or embed copies of Mentor Graphics' real-time operating systems or other embedded software products into Customer's products or applications without first signing or otherwise agreeing to a separate agreement with Mentor Graphics for such purpose.
4. **BETA CODE.**
 - 4.1. Portions or all of certain Software may contain code for experimental testing and evaluation ("Beta Code"), which may not be used without Mentor Graphics' explicit authorization. Upon Mentor Graphics' authorization, Mentor Graphics grants to Customer a temporary, nontransferable, nonexclusive license for experimental use to test and evaluate the Beta Code without charge for a limited period of time specified by Mentor Graphics. This grant and Customer's use of the Beta Code shall not be construed as marketing or offering to sell a license to the Beta Code, which Mentor Graphics may choose not to release commercially in any form.
 - 4.2. If Mentor Graphics authorizes Customer to use the Beta Code, Customer agrees to evaluate and test the Beta Code under normal conditions as directed by Mentor Graphics. Customer will contact Mentor Graphics periodically during Customer's use of the Beta Code to discuss any malfunctions or suggested improvements. Upon completion of Customer's evaluation and testing, Customer will send to Mentor Graphics a written evaluation of the Beta Code, including its strengths, weaknesses and recommended improvements.
 - 4.3. Customer agrees to maintain Beta Code in confidence and shall restrict access to the Beta Code, including the methods and concepts utilized therein, solely to those employees and Customer location(s) authorized by Mentor Graphics to perform beta testing. Customer agrees that any written evaluations and all inventions, product improvements, modifications or developments that Mentor Graphics conceived or made during or subsequent to this Agreement, including those based partly or wholly on Customer's feedback, will be the exclusive property of Mentor Graphics. Mentor Graphics will have exclusive rights, title and interest in all such property. The provisions of this Subsection 4.3 shall survive termination of this Agreement.
5. **RESTRICTIONS ON USE.**
 - 5.1. Customer may copy Software only as reasonably necessary to support the authorized use. Each copy must include all notices and legends embedded in Software and affixed to its medium and container as received from Mentor Graphics. All copies shall remain the property of Mentor Graphics or its licensors. Customer shall maintain a record of the number and primary location of all copies of Software, including copies merged with other software, and shall make those records available to Mentor Graphics upon request. Customer shall not make Products available in any form to any person other than Customer's employees and on-site contractors, excluding Mentor Graphics competitors, whose job performance requires access and who are under obligations of confidentiality. Customer shall take appropriate action to protect the confidentiality of Products and ensure that any person permitted access does not disclose or use it except as permitted by this Agreement. Customer shall give Mentor Graphics written notice of any unauthorized disclosure or use of the Products as soon as Customer learns or becomes aware of such unauthorized disclosure or use. Except as otherwise permitted for purposes of interoperability as specified by applicable and mandatory local law, Customer shall not reverse-assemble, reverse-compile, reverse-engineer or in any way derive any source code from Software. Log files, data files, rule files and script files generated by or for the Software (collectively "Files"), including without limitation files containing Standard Verification Rule Format ("SVRF") and Tcl Verification Format ("TVF") which are Mentor Graphics' proprietary syntaxes for expressing process rules, constitute or include confidential information of Mentor Graphics. Customer may share Files with third parties, excluding Mentor Graphics competitors, provided that the confidentiality of such Files is protected by written agreement at least as well as Customer protects other information of a similar nature or importance, but in any case with at least reasonable care. Customer may use Files containing SVRF or TVF only with Mentor Graphics products. Under no circumstances shall Customer use Software or Files or allow their use for the purpose of developing, enhancing or marketing any product that is in any way competitive with Software, or disclose to any third party the results of, or information pertaining to, any benchmark.
 - 5.2. If any Software or portions thereof are provided in source code form, Customer will use the source code only to correct software errors and enhance or modify the Software for the authorized use. Customer shall not disclose or permit disclosure of source code, in whole or in part, including any of its methods or concepts, to anyone except Customer's employees or contractors, excluding Mentor Graphics competitors, with a need to know. Customer shall not copy or compile source code in any manner except to support this authorized use.
 - 5.3. Customer may not assign this Agreement or the rights and duties under it, or relocate, sublicense or otherwise transfer the Products, whether by operation of law or otherwise ("Attempted Transfer"), without Mentor Graphics' prior written consent and payment of Mentor Graphics' then-current applicable relocation and/or transfer fees. Any Attempted Transfer without Mentor Graphics' prior written consent shall be a material breach of this Agreement and may, at Mentor Graphics' option, result in the immediate termination of the Agreement and/or the licenses granted under this Agreement. The terms of this Agreement, including without limitation the licensing and assignment provisions, shall be binding upon Customer's permitted successors in interest and assigns.

5.4. The provisions of this Section 5 shall survive the termination of this Agreement.

6. **SUPPORT SERVICES.** To the extent Customer purchases support services, Mentor Graphics will provide Customer updates and technical support for the Products, at the Customer site(s) for which support is purchased, in accordance with Mentor Graphics' then current End-User Support Terms located at <http://supportnet.mentor.com/about/legal/>.
7. **AUTOMATIC CHECK FOR UPDATES; PRIVACY.** Technological measures in Software may communicate with servers of Mentor Graphics or its contractors for the purpose of checking for and notifying the user of updates and to ensure that the Software in use is licensed in compliance with this Agreement. Mentor Graphics will not collect any personally identifiable data in this process and will not disclose any data collected to any third party without the prior written consent of Customer, except to Mentor Graphics' outside attorneys or as may be required by a court of competent jurisdiction.
8. **LIMITED WARRANTY.**
 - 8.1. Mentor Graphics warrants that during the warranty period its standard, generally supported Products, when properly installed, will substantially conform to the functional specifications set forth in the applicable user manual. Mentor Graphics does not warrant that Products will meet Customer's requirements or that operation of Products will be uninterrupted or error free. The warranty period is 90 days starting on the 15th day after delivery or upon installation, whichever first occurs. Customer must notify Mentor Graphics in writing of any nonconformity within the warranty period. For the avoidance of doubt, this warranty applies only to the initial shipment of Software under an Order and does not renew or reset, for example, with the delivery of (a) Software updates or (b) authorization codes or alternate Software under a transaction involving Software re-mix. This warranty shall not be valid if Products have been subject to misuse, unauthorized modification or improper installation. MENTOR GRAPHICS' ENTIRE LIABILITY AND CUSTOMER'S EXCLUSIVE REMEDY SHALL BE, AT MENTOR GRAPHICS' OPTION, EITHER (A) REFUND OF THE PRICE PAID UPON RETURN OF THE PRODUCTS TO MENTOR GRAPHICS OR (B) MODIFICATION OR REPLACEMENT OF THE PRODUCTS THAT DO NOT MEET THIS LIMITED WARRANTY, PROVIDED CUSTOMER HAS OTHERWISE COMPLIED WITH THIS AGREEMENT. MENTOR GRAPHICS MAKES NO WARRANTIES WITH RESPECT TO: (A) SERVICES; (B) PRODUCTS PROVIDED AT NO CHARGE; OR (C) BETA CODE; ALL OF WHICH ARE PROVIDED "AS IS."
 - 8.2. THE WARRANTIES SET FORTH IN THIS SECTION 8 ARE EXCLUSIVE. NEITHER MENTOR GRAPHICS NOR ITS LICENSORS MAKE ANY OTHER WARRANTIES EXPRESS, IMPLIED OR STATUTORY, WITH RESPECT TO PRODUCTS PROVIDED UNDER THIS AGREEMENT. MENTOR GRAPHICS AND ITS LICENSORS SPECIFICALLY DISCLAIM ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY.
9. **LIMITATION OF LIABILITY.** EXCEPT WHERE THIS EXCLUSION OR RESTRICTION OF LIABILITY WOULD BE VOID OR INEFFECTIVE UNDER APPLICABLE LAW, IN NO EVENT SHALL MENTOR GRAPHICS OR ITS LICENSORS BE LIABLE FOR INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES (INCLUDING LOST PROFITS OR SAVINGS) WHETHER BASED ON CONTRACT, TORT OR ANY OTHER LEGAL THEORY, EVEN IF MENTOR GRAPHICS OR ITS LICENSORS HAVE BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN NO EVENT SHALL MENTOR GRAPHICS' OR ITS LICENSORS' LIABILITY UNDER THIS AGREEMENT EXCEED THE AMOUNT RECEIVED FROM CUSTOMER FOR THE HARDWARE, SOFTWARE LICENSE OR SERVICE GIVING RISE TO THE CLAIM. IN THE CASE WHERE NO AMOUNT WAS PAID, MENTOR GRAPHICS AND ITS LICENSORS SHALL HAVE NO LIABILITY FOR ANY DAMAGES WHATSOEVER. THE PROVISIONS OF THIS SECTION 9 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.
10. **HAZARDOUS APPLICATIONS.** CUSTOMER ACKNOWLEDGES IT IS SOLELY RESPONSIBLE FOR TESTING ITS PRODUCTS USED IN APPLICATIONS WHERE THE FAILURE OR INACCURACY OF ITS PRODUCTS MIGHT RESULT IN DEATH OR PERSONAL INJURY ("HAZARDOUS APPLICATIONS"). NEITHER MENTOR GRAPHICS NOR ITS LICENSORS SHALL BE LIABLE FOR ANY DAMAGES RESULTING FROM OR IN CONNECTION WITH THE USE OF MENTOR GRAPHICS PRODUCTS IN OR FOR HAZARDOUS APPLICATIONS. THE PROVISIONS OF THIS SECTION 10 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.
11. **INDEMNIFICATION.** CUSTOMER AGREES TO INDEMNIFY AND HOLD HARMLESS MENTOR GRAPHICS AND ITS LICENSORS FROM ANY CLAIMS, LOSS, COST, DAMAGE, EXPENSE OR LIABILITY, INCLUDING ATTORNEYS' FEES, ARISING OUT OF OR IN CONNECTION WITH THE USE OF PRODUCTS AS DESCRIBED IN SECTION 10. THE PROVISIONS OF THIS SECTION 11 SHALL SURVIVE THE TERMINATION OF THIS AGREEMENT.
12. **INFRINGEMENT.**
 - 12.1. Mentor Graphics will defend or settle, at its option and expense, any action brought against Customer in the United States, Canada, Japan, or member state of the European Union which alleges that any standard, generally supported Product acquired by Customer hereunder infringes a patent or copyright or misappropriates a trade secret in such jurisdiction. Mentor Graphics will pay costs and damages finally awarded against Customer that are attributable to the action. Customer understands and agrees that as conditions to Mentor Graphics' obligations under this section Customer must: (a) notify Mentor Graphics promptly in writing of the action; (b) provide Mentor Graphics all reasonable information and assistance to settle or defend the action; and (c) grant Mentor Graphics sole authority and control of the defense or settlement of the action.

- 12.2. If a claim is made under Subsection 12.1 Mentor Graphics may, at its option and expense, (a) replace or modify the Product so that it becomes noninfringing; (b) procure for Customer the right to continue using the Product; or (c) require the return of the Product and refund to Customer any purchase price or license fee paid, less a reasonable allowance for use.
- 12.3. Mentor Graphics has no liability to Customer if the action is based upon: (a) the combination of Software or hardware with any product not furnished by Mentor Graphics; (b) the modification of the Product other than by Mentor Graphics; (c) the use of other than a current unaltered release of Software; (d) the use of the Product as part of an infringing process; (e) a product that Customer makes, uses, or sells; (f) any Beta Code or Product provided at no charge; (g) any software provided by Mentor Graphics' licensors who do not provide such indemnification to Mentor Graphics' customers; or (h) infringement by Customer that is deemed willful. In the case of (h), Customer shall reimburse Mentor Graphics for its reasonable attorney fees and other costs related to the action.
- 12.4. THIS SECTION 12 IS SUBJECT TO SECTION 9 ABOVE AND STATES THE ENTIRE LIABILITY OF MENTOR GRAPHICS AND ITS LICENSORS FOR DEFENSE, SETTLEMENT AND DAMAGES, AND CUSTOMER'S SOLE AND EXCLUSIVE REMEDY, WITH RESPECT TO ANY ALLEGED PATENT OR COPYRIGHT INFRINGEMENT OR TRADE SECRET MISAPPROPRIATION BY ANY PRODUCT PROVIDED UNDER THIS AGREEMENT.
13. **TERMINATION AND EFFECT OF TERMINATION.** If a Software license was provided for limited term use, such license will automatically terminate at the end of the authorized term.
- 13.1. Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement immediately upon written notice if Customer: (a) exceeds the scope of the license or otherwise fails to comply with the licensing or confidentiality provisions of this Agreement, or (b) becomes insolvent, files a bankruptcy petition, institutes proceedings for liquidation or winding up or enters into an agreement to assign its assets for the benefit of creditors. For any other material breach of any provision of this Agreement, Mentor Graphics may terminate this Agreement and/or any license granted under this Agreement upon 30 days written notice if Customer fails to cure the breach within the 30 day notice period. Termination of this Agreement or any license granted hereunder will not affect Customer's obligation to pay for Products shipped or licenses granted prior to the termination, which amounts shall be payable immediately upon the date of termination.
- 13.2. Upon termination of this Agreement, the rights and obligations of the parties shall cease except as expressly set forth in this Agreement. Upon termination, Customer shall ensure that all use of the affected Products ceases, and shall return hardware and either return to Mentor Graphics or destroy Software in Customer's possession, including all copies and documentation, and certify in writing to Mentor Graphics within ten business days of the termination date that Customer no longer possesses any of the affected Products or copies of Software in any form.
14. **EXPORT.** The Products provided hereunder are subject to regulation by local laws and United States government agencies, which prohibit export or diversion of certain products and information about the products to certain countries and certain persons. Customer agrees that it will not export Products in any manner without first obtaining all necessary approval from appropriate local and United States government agencies.
15. **U.S. GOVERNMENT LICENSE RIGHTS.** Software was developed entirely at private expense. All Software is commercial computer software within the meaning of the applicable acquisition regulations. Accordingly, pursuant to US FAR 48 CFR 12.212 and DFAR 48 CFR 227.7202, use, duplication and disclosure of the Software by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in this Agreement, except for provisions which are contrary to applicable mandatory federal laws.
16. **THIRD PARTY BENEFICIARY.** Mentor Graphics Corporation, Mentor Graphics (Ireland) Limited, Microsoft Corporation and other licensors may be third party beneficiaries of this Agreement with the right to enforce the obligations set forth herein.
17. **REVIEW OF LICENSE USAGE.** Customer will monitor the access to and use of Software. With prior written notice and during Customer's normal business hours, Mentor Graphics may engage an internationally recognized accounting firm to review Customer's software monitoring system and records deemed relevant by the internationally recognized accounting firm to confirm Customer's compliance with the terms of this Agreement or U.S. or other local export laws. Such review may include FLEXlm or FLEXnet (or successor product) report log files that Customer shall capture and provide at Mentor Graphics' request. Customer shall make records available in electronic format and shall fully cooperate with data gathering to support the license review. Mentor Graphics shall bear the expense of any such review unless a material non-compliance is revealed. Mentor Graphics shall treat as confidential information all information gained as a result of any request or review and shall only use or disclose such information as required by law or to enforce its rights under this Agreement. The provisions of this Section 17 shall survive the termination of this Agreement.
18. **CONTROLLING LAW, JURISDICTION AND DISPUTE RESOLUTION.** The owners of certain Mentor Graphics intellectual property licensed under this Agreement are located in Ireland and the United States. To promote consistency around the world, disputes shall be resolved as follows: excluding conflict of laws rules, this Agreement shall be governed by and construed under the laws of the State of Oregon, USA, if Customer is located in North or South America, and the laws of Ireland if Customer is located outside of North or South America. All disputes arising out of or in relation to this Agreement shall be submitted to the exclusive jurisdiction of the courts of Portland, Oregon when the laws of Oregon apply, or Dublin, Ireland when the laws of Ireland apply. Notwithstanding the foregoing, all disputes in Asia arising out of or in relation to this Agreement shall be resolved by arbitration in Singapore before a single arbitrator to be appointed by the chairman of the Singapore International Arbitration Centre ("SIAC") to be conducted in the English language, in accordance with the Arbitration Rules of the SIAC in effect at the time of the dispute, which rules are deemed to be incorporated by reference in this section. This section shall not

restrict Mentor Graphics' right to bring an action against Customer in the jurisdiction where Customer's place of business is located. The United Nations Convention on Contracts for the International Sale of Goods does not apply to this Agreement.

19. **SEVERABILITY.** If any provision of this Agreement is held by a court of competent jurisdiction to be void, invalid, unenforceable or illegal, such provision shall be severed from this Agreement and the remaining provisions will remain in full force and effect.
20. **MISCELLANEOUS.** This Agreement contains the parties' entire understanding relating to its subject matter and supersedes all prior or contemporaneous agreements, including but not limited to any purchase order terms and conditions. Some Software may contain code distributed under a third party license agreement that may provide additional rights to Customer. Please see the applicable Software documentation for details. This Agreement may only be modified in writing by authorized representatives of the parties. Waiver of terms or excuse of breach must be in writing and shall not constitute subsequent consent, waiver or excuse.

Rev. 100615, Part No. 246066