



Valor® ODB++ Inside for Cadence Allegro

Software Version 9.3.1

February 2013

© 1996—2013 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 HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

U.S. GOVERNMENT LICENSE RIGHTS: The software and documentation were developed entirely at private expense and are commercial computer software and commercial computer software documentation within the meaning of the applicable acquisition regulations. Accordingly, pursuant to FAR 48 CFR 12.212 and DFARS 48 CFR 227.7202, use, duplication and disclosure by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in the license agreement provided with the software, except for provisions which are contrary to applicable mandatory federal laws.

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the owner of the Mark, as applicable. 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.

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

Table of Contents

Chapter 1	
ODB++ Inside for Cadence Allegro	5
Translation Workflow	5
System Requirements	6
Chapter 2	
Exporting to ODB++	7
Using the Translator	7
Specifying File Options and Output Options	8
Specifying Parameters Specific to Cadence Allegro	10
Saving the Configuration	17
Editing the Matrix File	18
Chapter 3	
Viewing the Product Model in ODB++ Viewer	21
Opening ODB++ Viewer	21
File Menu Options	23
Actions Menu Options	24
View Menu Options	24
Help Menu Options	25
Board Viewer Right-Click Menu Options	25
Board Viewer Shortcuts	26
Viewing Component or Feature Properties	27
Setting Display Colors	30
Chapter 4	
System Administrator Notes	33
Running the Translator from Design Workbench	33
Setting Configuration Parameters	34
Using the config file to set Configuration Parameters	34
Configuration Parameters	34
Setting Environment Variables	39
Configuring a Thermal Model	39
Structure of a Thermal Model File	40
BNF (Backus-Naur Form) of Rules	40
Sample Thermal Model File	42
Generating the Extraction Files	42
comps_<product_model>.out	43
films_<product_model>.out	43
geoms_<product_model>.out	44
layers_<product_model>.out	44
nets_<product_model>.out	44

pads_<product_model>.out	45
pins_<product_model>.out	45
props_<product_model>.out	45
tech_<product_model>.out	45
Using Command Line Parameters	46
Supported Features	49
Skipping Extraction of Net Impedance Average	49
Placing Components Underneath Raised Components	50
Using Height Parameters for Keepout Areas	51
Back-Drill Information is Translated	52
Support for Mirrored Padstacks	52
Support for the COMPONENT KEEPOUT Class	52
Release Notes	53
Version 9.0 Features and Resolved Issues	53
Version 9.1 Features and Resolved Issues	54
Version 9.2 Features and Resolved Issues	55
Version 9.3 Features and Resolved Issues	55

End-User License Agreement

Chapter 1

ODB++ Inside for Cadence Allegro

ODB++ is a format designed to capture all CAD or EDA assembly and PCB fabrication information in a single, unified database.

ODB++ Inside for Cadence Allegro contains the BRD2ODB translator and the ODB++ Viewer.

- The translator converts Cadence Allegro board files to ODB++ V7.
- The translator supports Cadence Allegro version 11 - 16.5, and OrCAD Designer V16 and up.
- ODB++ Viewer displays the resulting ODB++ information, graphically. See [“Viewing the Product Model in ODB++ Viewer”](#) on page 21.

When Allegro is to be launched from the Allegro Design Workbench, environment variable **PCBDW_USER_PATH** must be set when ODB++ Inside is installed, as described in [“Running the Translator from Design Workbench”](#) on page 33.

The Cadence Allegro V1.6 APD family of products includes Cadence SiP. The translator can read **sip** files in addition to **brd** files and **mcm** files.

This version of the translator does not include the option to save as the earlier ODB++ V6. This functionality was removed so that there is no confusion over what should be sent to manufacturing. Manufacturers must use a software version capable of reading ODB++ V7 format. Mentor/Frontline applications such as Genesis work with a variation of the ODB++ format, but they can import and use the ODB++ V7 format.

Translation Workflow	5
System Requirements	6

Translation Workflow

To translate a Mentor Graphics Board Station design to ODB++, perform the following tasks:

Task	Information
Check that the system meets the requirements	“System Requirements” on page 6
Set up the Environment	“System Administrator Notes” on page 33
Export the design to ODB++	“Exporting to ODB++” on page 7

Task	Information
View the resulting ODB++ information	“Viewing the Product Model in ODB++ Viewer” on page 21

System Requirements

The translator can be run on the following operating systems:

Operating System	Operating System Versions
Windows XP	32 bit SP2, SP3 64 bit SP3
Windows Vista	Business SP2 (32 bit, 64 bit) Ultimate SP2 (32 bit, 64 bit)
Windows 7	Professional (32 bit, 64 bit)
Windows 2003 server	32 bit
Windows 2008 server	32 bit
Red Hat Enterprise Linux	V4.4, V4.6, V5.x, V6 (32bit, 64bit) X86 64bit OSs that can run X86 32bit executables
SUSE Linux Enterprise Server	V10 X86 SP3, V11 (desktop and server editions - 32 and 64 bit)
Solaris	10

Chapter 2

Exporting to ODB++

Using the Translator	7
Specifying File Options and Output Options	8
Specifying Parameters Specific to Cadence Allegro.....	10
Saving the Configuration	17
Editing the Matrix File	18

Using the Translator

You use the ODB++ Inside for Cadence Allegro translator to specify parameters and to run the translation to export a Cadence Allegro design to ODB++.

Procedure

1. Perform the following tasks as necessary:
 - o “Setting Configuration Parameters” on page 34.
 - o “Setting Environment Variables” on page 39.
 - o “Configuring a Thermal Model” on page 39.
 - o “Generating the Extraction Files” on page 42.
2. Open ODB++ Inside in one of these ways. See “Using Command Line Parameters” on page 46.
 - o From within Cadence Allegro, using the menu option.
 - o In Windows, the translator can be activated from a line mode command:
Command line: “%ALLEGRO_BRD2ODB%/brd2odb.exe” -gui
 - o In UNIX, type the following command: \$ALLEGRO_BRD2ODB/brd2odb -gui
3. Specify the information described in “Specifying File Options and Output Options” on page 8.
4. If necessary, perform the following tasks:
 - o “Specifying Parameters Specific to Cadence Allegro” on page 10
 - o “Editing the Matrix File” on page 18

5. If you will be using these parameters for subsequent translations, see [“Saving the Configuration”](#) on page 17.
6. Click **Run** to perform the translation.

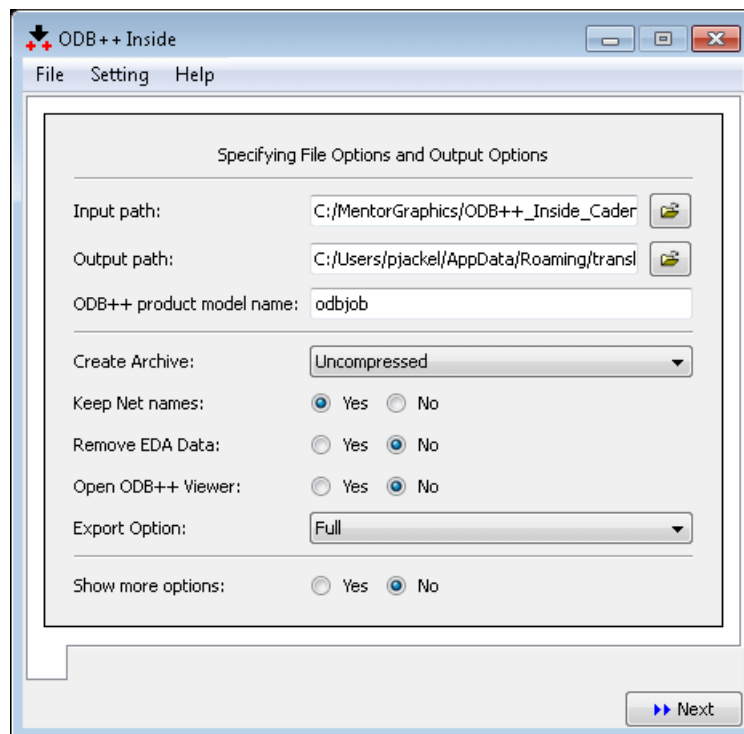
Specifying File Options and Output Options


You must provide input and output paths and output options needed by the translator.

If you will be using these parameters for subsequent translations, see [“Saving the Configuration”](#) on page 17.

Procedure

1. Open ODB++ Inside as described in [“Using the Translator”](#) on page 7.



2. Specify these file options. Click  to browse to a folder.

File Option	Explanation
Input path	The input path of the Allegro design.
Output path	The path for the ODB++ output.
Output product model name	The name of the ODB++ product model to be created.

3. Select the actions to be performed by the translator.

Output Option	Explanation
Create Archive	Controls the format of the ODB++ output. <ul style="list-style-type: none"> • Uncompressed (default) • Tar — Compresses the ODB++ folders into a tared file. • Tar and GZIP — Compresses the ODB++ folders into a tared and zipped file.
Keep Net names	Controls whether net names are renamed numerically or are kept as their original names.
Remove EDA Data	Removes component/package data.
Open ODB++ Viewer	Opens the ODB++ Viewer application to display the imported design. See “Viewing the Product Model in ODB++ Viewer” on page 21
Export Option	Controls how much data is exported to ODB++: <ul style="list-style-type: none"> • Full—all information in the design Export Fabrication • FAB—Exports layers and data options for fabrication: <ul style="list-style-type: none"> - Physical nets - output for net points - Outer copper layers - Silk Screen layers - Solder Paste layers - Solder Mask layers - Drill / Rout layers - Document layers - Inner layers • ASSY—Exports layers and data options for assembly: <ul style="list-style-type: none"> - Components /Packages & Logical nets - components + logical nets (net nodes/net attributes/net properties) - Physical nets - output for net points - Outer copper layers - Silk Screen layers - Solder Paste layers - Solder Mask layers - Drill / Rout layers - Document layers

4. If you want to set additional parameters, select **Show more options** and click **Next**. See [“Specifying Parameters Specific to Cadence Allegro”](#) on page 10.

5. If you want to run the translation without setting additional parameters, click **Run**.

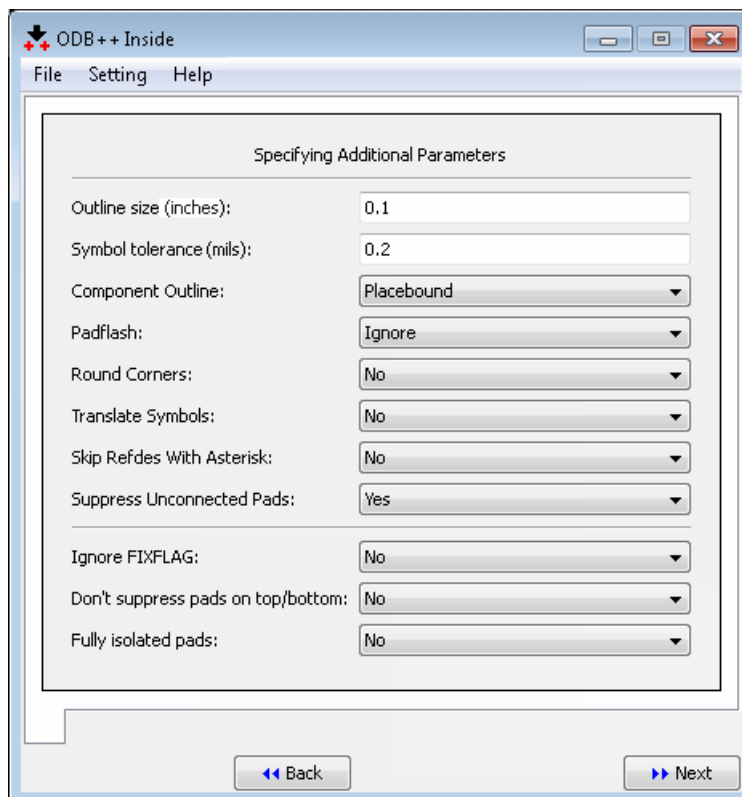
Specifying Parameters Specific to Cadence Allegro

You can specify parameters specific to Cadence Allegro.

If you will be using these parameters for subsequent translations, see [“Saving the Configuration”](#) on page 17.

Procedure

1. Open the **ODB++ Inside** wizard as described in [“Using the Translator”](#) on page 7.
2. Enter the file options and output options as described in [“Specifying File Options and Output Options”](#) on page 8.
3. Select **Show more options** and click **Next**.

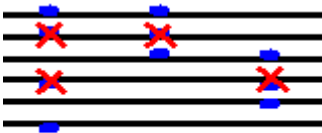
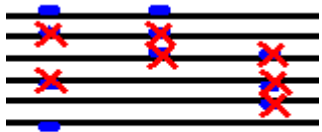



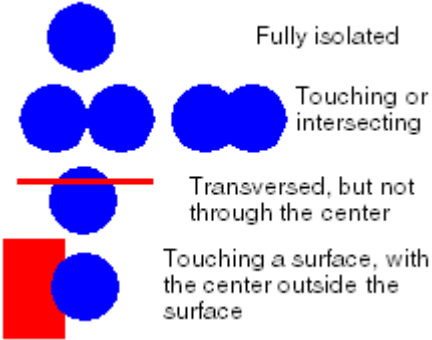
4. Enter the appropriate parameters:

Parameter	Explanation
Outline size (inches)	When creating negative plane layers, the size of the frame is the value of this parameter. For accurate translation this value should match the -o option in the Cadence Allegro artwork program. If these two parameters differ, the frame will be created according to the value in Outline size . The value is in inches.
Symbol tolerance (mils)	The system compares shapes that are input, with symbols previously input in the same session, and with standard and semi-standard system symbols. <ul style="list-style-type: none"> • 0 — only if the input shape exactly matches a system symbol, is the system symbol used. If it does not match, the input shape is used ‘as is’ without change. • positive value — the input shape is compared to system symbols within the tolerance specified. If it can be matched, the system symbol is used. Use this parameter as appropriate for the type of file you expect to input. The lower the tolerance the more critical the system is in judging that shapes are equivalent. The value is specified in mils.
Component Outline	Sets the component outline according to the classes ASSEMBLY_TOP / ASSEMBLY_BOTTOM of package geometry or according to the sub-classes PLACE_BOUND_TOP / PLACE_BOUND_BOTTOM . <ul style="list-style-type: none"> • Place Bound—(recommended) when place bound shapes are available they will be used for the component outline; otherwise the limits of the assembly features will be used. • Assembly—a heuristic algorithm is used, which tries to determine the actual component outline from the collection of data on the ASSEMBLY_TOP / ASSEMBLY_BOTTOM class. Note that using this option might result in an unexpected component outline, because the data defining it is expected to be complete in terms of ODB++, that is, a well defined closed polygon.
Padflash	Allegro pad definitions can have Padflash codes that effectively override the pad size information for the padstack. This information is extracted into the 12th. field of the pad extract file (pads_[brd name].out). For instance, on fiducials, a designer defines a padstack called FID120RD40RD that appears in Allegro as a 120 mil diameter pad with a 120 mil diameter solder mask. It also has a Padflash definition of RD40 . <ul style="list-style-type: none"> • Ignore (default) — the Padflash field is ignored and instead, the pad size is used. So, during translation the example padstack would be constructed of 120 mil diameter pads. • Substitute (recommended) — the name in the Padflash field will be used in conjunction with the thermal models file to determine what is placed at the location. In this case, the Padflash name RD40 would determine the actual fiducial on the copper layer based on the current Thermal Model.

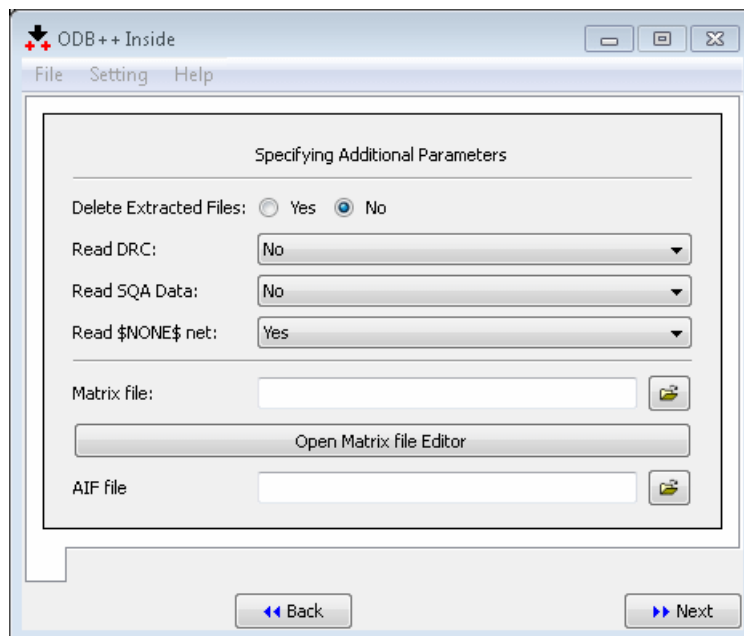
Parameter	Explanation
Round Corners	<p>Indicates whether corners should be rounded.</p> <ul style="list-style-type: none"> • No — (default) process precise (square) corners. • Yes — round corners of polygons (contours).
Skip Refdes With Asterisk	<p>Controls whether components with names containing an asterisk (*) should be translated.</p> <ul style="list-style-type: none"> • No — all components are translated. (default) • Yes — components with names containing an asterisk are not translated. • Part — the translation excludes components whose RefDes contains an asterisk (*) but includes their pad and drill features.
Suppress Unconnected Pads	<p>Controls whether unconnected pads are included in the translated design if the Allegro design contains the following:</p> <ul style="list-style-type: none"> • In the films file, SUPPRESS_UNCONNECTED_PADS = Yes. • In the pads extract file, FIXFLAG = o (optional). If FIXFLAG = f (fixed), it can be ignored by selecting Ignore FIXFLAG. <p>This sets configuration parameter eda_cadence_suppress. Pads on negative layers are never suppressed.</p> <ul style="list-style-type: none"> • Yes — Unconnected pads are not translated. Options Ignore FIXFLAG, Don't suppress pads on top/bottom, and Fully isolated pads are available to control which pads are suppressed, unless you are using options in Cadence Allegro V16.2 to control how to treat unconnected pads, and you access the translator from within Allegro. • No (default) — Unconnected pads are translated. <p>Related line mode command switches are iff, bb, fi, and ups. See “Using Command Line Parameters” on page 46.</p>

5. If you have entered **Suppress Unconnected Pads = Yes**, enter these parameters:

Parameter	Explanation
Ignore FIXFLAG	Ignores the setting of FIXFLAG in the pads extract file.
Don't suppress pads on top/bottom	<p>Controls whether unconnected pads at the top and bottom of a drill are suppressed. Available only if Suppress Unconnected Pads = Yes.</p> <ul style="list-style-type: none"> • Yes — Unconnected pads at the top and bottom of a drill are not suppressed. (All unconnected pads other than those on the top and bottom of a drill are suppressed) <div style="text-align: center;">  <p>TH blind buried</p> </div> <ul style="list-style-type: none"> • No (default) — Unconnected pads at the top and bottom of a drill are suppressed. (All unconnected pads other than those on the top and bottom layers of a board are suppressed.) <div style="text-align: center;">  <p>TH blind buried</p> </div>

Parameter	Explanation
Fully isolated pads	<p>Controls which pads are considered to be unconnected. Available only if Suppress Unconnected Pads = Yes.</p> <ul style="list-style-type: none"> • Yes—Only single pads, touching no other feature on the layer, are considered to be unconnected.  <ul style="list-style-type: none"> • No—(default) All of these pads are considered to be unconnected: <ul style="list-style-type: none"> - a single totally isolated pad - two pads touching or intersecting - a pad transversed by a trace not through its center - a pad touching a surface where its center is not inside the surface. 

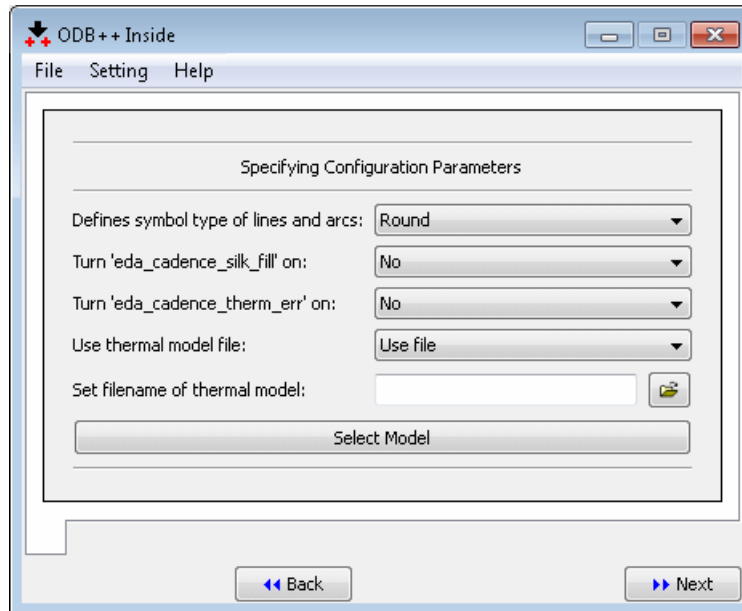
6. Click Next.



7. Enter the appropriate parameters:

Parameter	Explanation
Delete Extracted Files	Controls whether temporary extract files created during translation are deleted.
Read DRC	Adds DRC features to physical DRC layers. DRC layers are added as BOARD document layers. Some can be merged into one physical layer depending on configuration parameter settings.
Read SQA Data	Indicates that Signal Quality Analysis data should be read (default). If SQA data is not needed, you can clear this option. The Signal Quality data layer is not created nor is the tech file read. The translation takes less time.
Read \$NONE\$ net	Controls whether to assign features with no net to the \$NONE\$ net. <ul style="list-style-type: none"> • Yes—assign features with no net to the \$NONE\$ net (default) • No—do not assign features with no net to the \$NONE\$ net.
Matrix File	The matrix file of the product model. Click Open Matrix file Editor to edit an existing matrix if one exists, or a matrix generated from the product model as it will be translated. See “Editing the Matrix File” on page 18.
AIF File	HDI net information can be translated, and can be used to perform HDI net validation. By default, an AIF file residing in the same folder as the <i>out</i> files is used during translation. If your AIF file is located in a different folder, specify the location in the AIF File box. This location is used for subsequent translations even if there is an AIF file in the same folder as the <i>out</i> files, so be sure and change this location for subsequent translations if necessary. Make sure that you are using the current Skill script. Before opening Cadence Allegro to export information, copy the current script to the Cadence directory from this location: <i><installation folder>\all\eda\cadence\set_allegro</i>

8. Click Next.



9. Enter the appropriate parameters.

Parameter	Explanation
Symbol type for lines and arcs	Defines the symbol type for lines and arcs that are part of the step profile polygon. <ul style="list-style-type: none"> • Round—the symbol type is round. • Square—the symbol type is square.
Turn eda_cadence_silk_fill on	Fills surfaces on Allegro silk screen layers. Setting this parameter sets configuration parameter eda_cadence_silk_fill .
Turn eda_cadence_thermal_error on	The translator aborts with a message listing the padstack / thermal names that did not have a match in the models file. Setting this parameter sets configuration parameter eda_cadence_thermal_error .
Use thermal model file	Controls whether a thermal file is used. <ul style="list-style-type: none"> • Default—Use a default model that uses direct connect and no thermals. • Use file—Use a model stored in a thermal model file.
Set file name of thermal model	The file to be used when Use thermal model file = Use file . Click Select Model to select the model in the file.

Related topics

- [“Configuring a Thermal Model”](#) on page 39.
- [“Editing the Matrix File”](#) on page 18.

Saving the Configuration

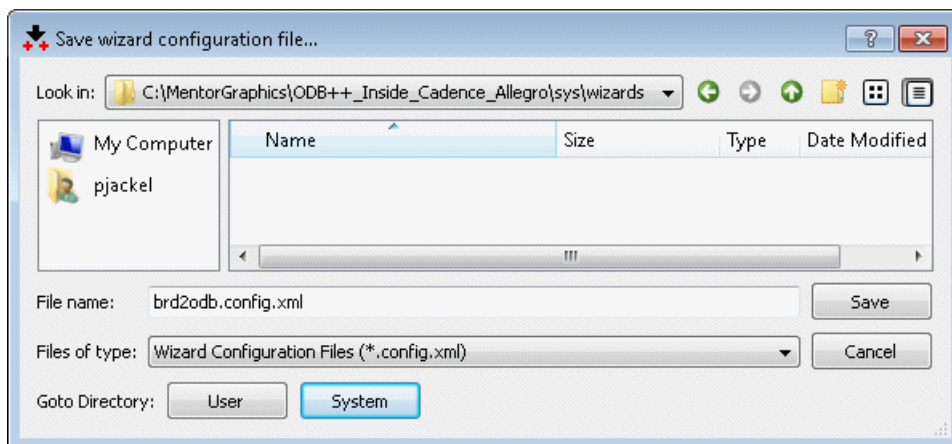
If you will be using the same parameters for several translations, you can save the configuration to a file.

You can save the configuration to the standard system location, to the standard user location, or to another location. The user-level configuration, if it exists, is loaded when ODB++ Inside starts. Otherwise, the system-level configuration is used.



Procedure

1. Run the ODB++ Inside wizard as described in “Using the Translator” on page 7.
2. Enter parameters as described in “Specifying File Options and Output Options” on page 8.
3. Enter parameters as described in “Specifying Parameters Specific to Cadence Allegro” on page 10.
4. To save the parameters you have entered, select **Settings > Save Config**.

The Save wizard configuration file dialog box opens.



5. Save the file in the appropriate location:

File Location	File Name	Result
System location accessed using the System button 	brd2odb.config.xml	This configuration is loaded when ODB++ Inside opens, if there is no configuration file in the user location.
User location accessed using the User button 	brd2odb.config.xml	If there is a file saved at the user location, it is loaded when ODB++ Inside opens.

File Location	File Name	Result
other	any	You can save the configuration file in another location. To use the parameter settings, copy the file to the standard name, in either the user location or the system location, before opening ODB++ Inside.

Editing the Matrix File

Layers are translated according to the data taken from the files **layers_<product_model>.out** and **films_<product_model>.out**. It is not unusual to find data for copper layers mixed with document layers. The translator designates the top/bottom layers according to the pairs of class | sub-class **ETCH|<layer_name>**. If several layers contain these pairs, the first one found is used. To avoid the mixing and duplication of layer data, it is necessary to edit the matrix file before translation.

The first time a design is translated, it does not usually contain a matrix file.

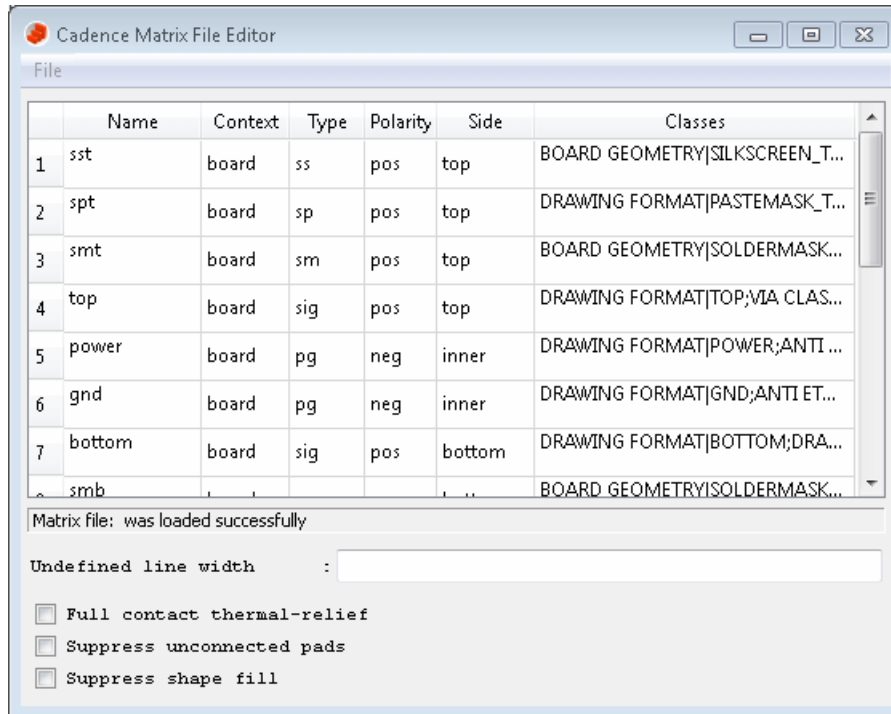
If **Matrix File** is left empty, the translator will create a matrix file from *out* files.

You can view and set options for the width of undefined lines, thermal relief, unconnected pads and shape fill for each layer. By default, settings are read from the films file. The options of the matrix file editor function like the options on the **Artwork Control Form** dialog box of Cadence Allegro:

- **Full contact thermal-relief**—controls the creation of thermal symbols. **valor_ex.il** creates ASCII files named **<thermal symbol name>.outdra** if there are DRA files with the design. These files are used to create thermal symbols. Each file defines one thermal symbol. Only the thermals for which there are **outdra** files are replaced.
- **Suppress unconnected pads**—suppresses unconnected pads for the selected layer.
- **Suppress shape fill**— suppresses the creation of the laminate area during translation.

Procedure

1. Click **Open Matrix file Editor**.
 - If you have specified a file in **Matrix file**, that file is opened.
 - If no **Matrix file** is specified, the translator reads layer data in **Input path** and builds a matrix as it would appear after translation.



2. Use the drop-down lists to edit parameters so that each layer is correctly defined. You can edit these parameters: **Context**, **Type**, **Polarity**, and **Side**.
 - If you change a top/bottom layer to a document layer, its name is changed to what it was originally.
 - If you change a document layer to a signal layer, its name is assigned according to the **ETCH** sub-class found in it.

Make sure that changes to layers remain synchronized. For example, signal must be assigned **side=top/bottom** and P&G layers must be **side=inner**. They cannot be of

context **misc**. Document layers must be assigned **side=auto** and **polarity=pos**.
Unsynchronized data causes incorrect translation.

Option	Explanation
Full contact thermal-relief	Controls the creation of thermal symbols on a specific layer. <ul style="list-style-type: none">• Selected—Suppresses the creation of thermal symbols.• Cleared—Creates thermal symbols, if they are defined, according to the following:<ul style="list-style-type: none">• If there are <thermal symbol name>.outdra files, thermal symbols are added as defined in these files.• If there are no outdra files, and Thermals = Use file was specified in EDA Interface Popup, the thermal model specified in Model is searched. If there are thermal symbols defined there, they are added.
Suppress unconnected pads	Controls whether unconnected pads are suppressed for the selected layer.
Suppress shape fill	Controls the creation of the laminate area for the selected layer during translation. <ul style="list-style-type: none">• Selected—Creation of the laminate area is suppressed. The design must have filled areas replaced with separation lines in Power & Ground layers.• Cleared—By default, text on P&G layers is translated with negative polarity. This reads product models in the same way the -s switch is used in the Allegro Artwork command. The laminate area is created for all negative layers by creating a single surface consisting of the board outline (filled) with all split plane areas subtracted from it. Creation of the laminate area in ODB++ is equivalent to the “shapefill” algorithm in Allegro (the -s switch is used to suppress the shapefill algorithm).

3. Select **File > Save** to save the corrections. You can specify the edited matrix file in **Matrix file** so that the translation creates layers according to the file.

Chapter 3

Viewing the Product Model in ODB++ Viewer

You can use ODB++ Viewer to view the ODB++ product model. You can view the graphic of a step and information about each layer of the step.

Opening ODB++ Viewer	21
Viewing Component or Feature Properties	27
Setting Display Colors	30

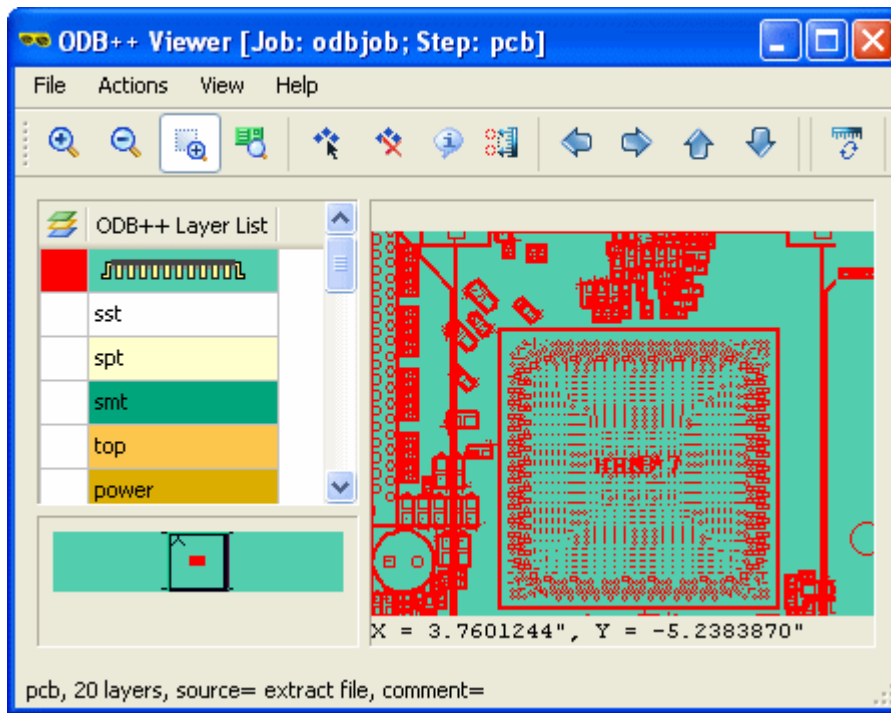
Opening ODB++ Viewer


When you run the translator, you can specify that the ODB++ Viewer be opened when the translation completes.

Procedure

1. Start the ODB++ translator.
2. In the **Specifying File Options and Output Options** page of the wizard, set **Open ODB++ viewer = Yes**.
3. Perform the translation.

The ODB++ Viewer opens, displaying the resulting ODB++ data.



4. Click the **Toggle Units** tool on the toolbar  to toggle the measurement units displayed in the X and Y coordinates under the board viewer, and for the **Measure** tool, between inches and mm.
5. Select one or more layers in the **ODB++ Layer List** to display them in the board viewer.
6. Use the menu options and shortcuts to view the step:
 - [“File Menu Options”](#) on page 23
 - [“Actions Menu Options”](#) on page 24
 - [“View Menu Options”](#) on page 24
 - [“Help Menu Options”](#) on page 25
 - [“Board Viewer Right-Click Menu Options”](#) on page 25
 - [“Board Viewer Shortcuts”](#) on page 26

Related topics

[Viewing Component or Feature Properties](#)

[Setting Display Colors](#)





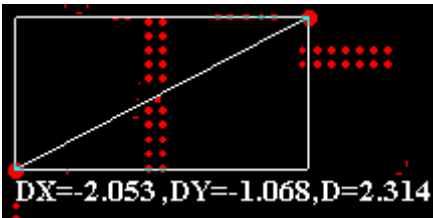
File Menu Options

Use these **File** menu options to open a step to view.

Tool	File Menu Option	Explanation
-	Open Job	(not supported in this version)
-	Open Step	Lists the steps of the current product model, so you can open the step you want to view. This option is disabled if the product model has only one step.
-	Exit	Exit the ODB++ Viewer.



Actions Menu Options



Use these **Actions** menu options or toolbar tools to rotate or mirror the board view, to highlight features, or to measure between points on the board.

Tool	Actions Menu Option	Explanation
	Highlight	<p>Select the tool and click a feature to be highlighted in the colors set for Highlighted.</p> <p>A symbol in the Color of features for each layer column of the ODB++ Layer List indicates whether a feature is highlighted on the layer.</p>  <p>See “Setting Display Colors” on page 30.</p>
	Clear Highlighted	Clear highlighting applied using Highlight .
	Measure	<p>To measure the distance between two points, select the tool, and click the first point and the second point in the board viewer. The distances in the X and Y direction, and the distance between the points are displayed.</p> 

View Menu Options

Use these **View** menu options or toolbar tools to customize the view of the board, and display additional information about the product model.

Tool	View Menu Option	Explanation
	Zoom In, Zoom Out Zoom Area Zoom Home	<p>Zoom in or zoom out.</p> <p>Invoke the zoom area tool for drawing a zoom rectangle in the board viewer or overview pane.</p> <p>Zoom to a view that fits the board in the graphic area.</p>
	Pan Left, Pan Right, Pan Up, Pan Down	<p>Pan left, right, up, or down.</p> <p>(shortcuts: keyboard arrow keys)</p>

Tool	View Menu Option	Explanation
	View Properties	Opens the Component Information or Feature Information dialog box. See “ Viewing Component or Feature Properties ” on page 27.
	Color Settings	Set colors as described in “ Setting Display Colors ” on page 30.





Help Menu Options

Use these **Help** menu options to view information about ODB++ Viewer.

Tool	View Menu Option	Explanation
-	View Help	Opens documentation for the ODB++ Viewer. (shortcut: F1)
-	About	Displays ODB++ Viewer version and system information.









Board Viewer Right-Click Menu Options






Right click on the board viewer pane to access these options:

Option	Explanation
Zoom area	Invokes the zoom area tool so you can drag a zoom rectangle in the board viewer pane or overview pane. Equivalent to clicking the Zoom Area  tool.
Measure	Invokes the measure tool so you can measure the distance between two points. Equivalent to clicking the Measure  tool.
Highlight	Invokes the highlight tool so you can highlight a feature or component. Equivalent to clicking the Highlight a feature or a component  tool.
Clear highlighted	Clears all highlighted features and components. Equivalent to clicking the Clear Highlighted  tool.

Board Viewer Shortcuts

Use these shortcuts if necessary to customize the view in the board viewer pane. The mouse pointer must be located in the board viewer pane.

Shortcut	Explanation
Ctrl + a	Invokes the Zoom area tool for one actuation. After you have dragged a rectangle in the board viewer pane or in the overview pane, the tool that was previously in effect is in effect.
Ctrl + d down arrow	Pan down with 90% overlap.
Ctrl + e	Click on the board graphic and press ctrl + e to pan the board viewer pane to center that point in the pane.
Ctrl + h Ctrl + Home	Zooms to a view that fits the entire board in the graphic area.
Ctrl + i Page Up	Zooms in. Equivalent to clicking the Zoom In tool.    
Ctrl + l left arrow	Pan left with 90% overlap.
Ctrl + m	Toggles between a four layer display and more than four layers.
Ctrl + o Page Down	Zooms out. Equivalent to clicking the Zoom Out tool.    
Ctrl + r right arrow	Pan right with 90% overlap.
Ctrl + Shift + d Shift + down arrow	Pan down with 10% overlap.
Ctrl + Shift + l Shift + left arrow	Pan left with 10% overlap.
Ctrl + Shift + r Shift + right arrow	Pan right with 10% overlap.
Ctrl + Shift + Space	Returns to the zoomed display in memory after zooming other areas.
Ctrl + Shift + u Shift + up arrow	Pan up with 10% overlap.
Ctrl + Space	Copies the current zoomed display to memory.
Ctrl + u up arrow	Pan up with 90% overlap.

Shortcut	Explanation
Ctrl + w	Toggles the view of features among these modes:  full  , outline  , or width off 
middle-click	The middle button can be used for these pan and zoom functions: <ul style="list-style-type: none"> • Center in Graphic Area—Middle-click a point in the graphic area. The click point is panned to the center of the graphic area. • Zoom Out—With the middle button, drag a diagonal up and to the right, ending at the point you want in the center of the graphic area. • Zoom In—With the middle button, drag a diagonal down and to the left, ending at the point you want in the center of the graphic area. • Zoom to Area—With the middle button, drag a diagonal down and to the right, so that the start and end of the line you drag define a rectangular area to which you want to zoom. • Zoom Home—With the middle button, drag a diagonal up and to the left. The view zooms so that the whole board is displayed centered in the graphic area.
Shift + Home	Zooms to a view that fits the highlighted component in the board area. Equivalent to clicking the Zoom Home tool. 

Viewing Component or Feature Properties

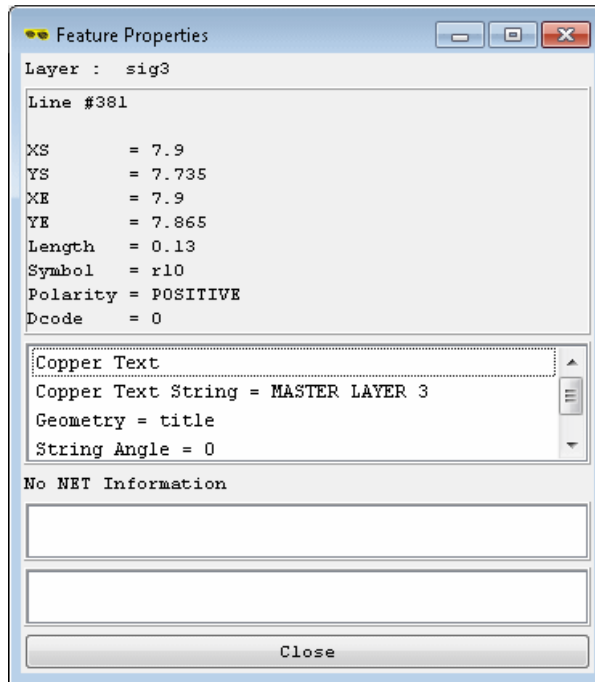
You can view properties of a component or feature that is highlighted in the graphic.

Procedure

1. Open ODB++ Viewer as described in “Opening ODB++ Viewer” on page 21.
2. In the ODB++ Layers List, display the layer containing the components or features you want to view.
3. To view properties of a feature, highlight a feature in the graphic and click **View**

Properties .

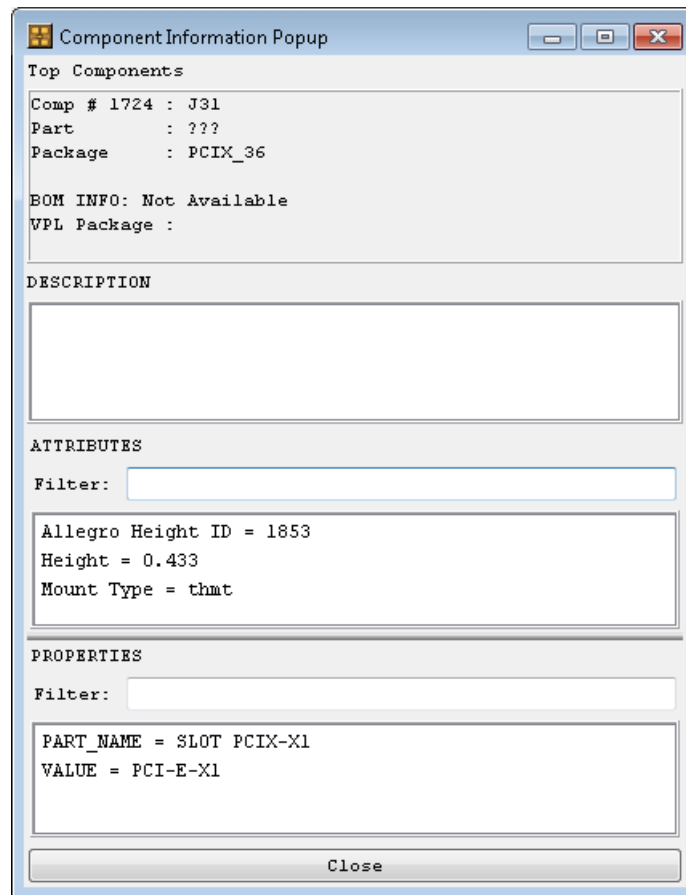
The Feature Properties dialog box opens.



While the Feature Properties dialog box is open, each time you click on a feature in the board viewer, the properties of that feature are displayed in the Feature Properties dialog box.

4. To view properties of a component, highlight a component in the graphic and click **View Properties** .

The Component Properties dialog box opens.



While the Component Properties dialog box is open, each time you click on a component in the board viewer, the properties of that component are displayed in the Component Properties dialog box.

Related topics

[Opening ODB++ Viewer](#)

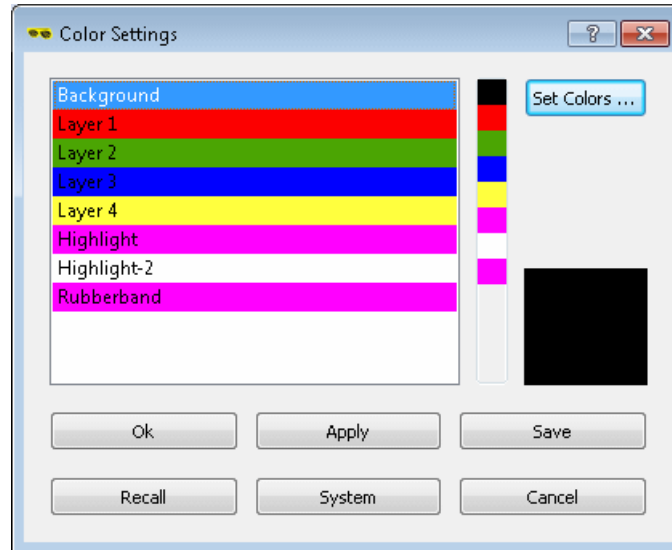
[Setting Display Colors](#)

Setting Display Colors

You can set the colors that uses to display items in the board viewer. You can save color specifications in a file. Where colors overlap, they are mixed.

Procedure

1. Open ODB++ Viewer as described in “Opening ODB++ Viewer” on page 21.
2. Select **View > Color Settings**.

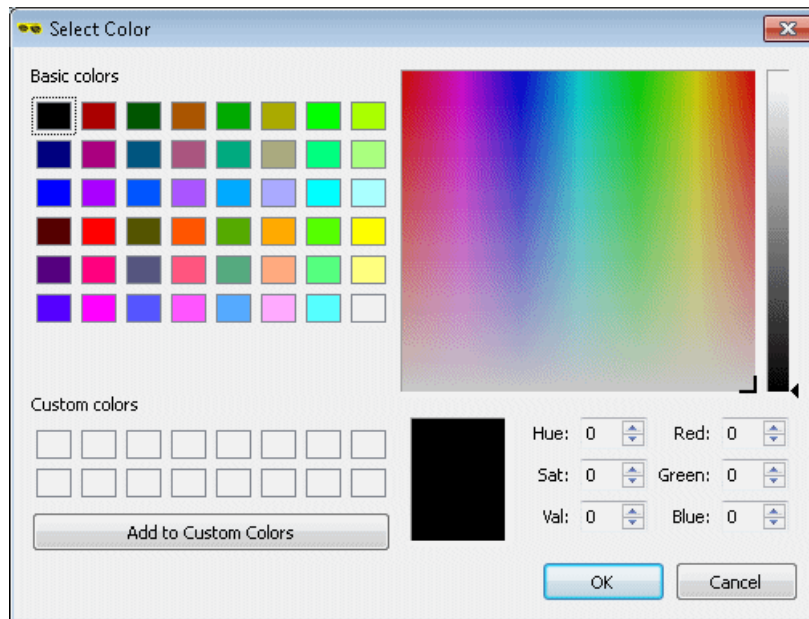


In the Colors Settings dialog box, select the item for which you want to set the color.

Board Viewer Item	Explanation	Suggestion
Background	Board viewer background	dark color
Layer 1, Layer 2, Layer 3, Layer 4	Features or components in the first, second, third and forth layer selected for display	bright color
Highlight	Highlighted features and components	light color
Rubberband	The rectangle drawn in the board viewer pane or the overview pane to zoom to an area of the board.	light color

The current color for that item is displayed in the square on the right.

3. Click **Set Colors**.



In the Select Color dialog box, select a color and click **OK**.

4. Use the buttons on the Color Settings dialog box to perform these actions:

Button	Action
OK	Apply the selected color to the items and close the dialog box.
Apply	Apply the selected color to the items.
Save	Save the current color settings.
Recall	Recall the saved color settings.
System	Revert to the system default colors.
Cancel	Close the dialog box without changing color settings.

Related topics

[Opening ODB++ Viewer](#)

[Viewing Component or Feature Properties](#)

Chapter 4

System Administrator Notes

Running the Translator from Design Workbench	33
Setting Configuration Parameters	34
Setting Environment Variables	39
Configuring a Thermal Model	39
Generating the Extraction Files	42
Using Command Line Parameters	46
Supported Features	49
Release Notes	53

Running the Translator from Design Workbench

When Allegro is to be launched from the Allegro Design Workbench, environment variable **PCBDW_USER_PATH** must be set when ODB++ Inside is installed.

Procedure

1. Locate the Allegro Design Workbench launch wrapper file **adwstart.bat**. This file is typically located under the install tree.
2. Edit **adwstart.bat** to include this line:

```
set PCBDW_USER_PATH=<path to ODB++ Inside>\nv\bin
```

where <path to ODB++ Inside> is the path to the ODB++ Inside module, typically C:\MentorGraphics\Allegro Export ODB++.

Setting Configuration Parameters

This topic discusses the config file and the relevant configuration parameters:

- [Using the config file to set Configuration Parameters](#)
- [Configuration Parameters](#)

Using the config file to set Configuration Parameters

Configuration parameters are defined in the file **config** that is installed with Allegro Export ODB++, in `$ALLEGRO_BRD2ODB`—the same directory as the program executables.

If you run ODB++ Inside for Cadence Allegro from a line mode command, you can specify a different location for the config file, in parameter **-cfg**. See [“Using Command Line Parameters”](#) on page 46.

The config file can contain parameters used by several translators.

Edit the file to set the appropriate values for the configuration parameters, before opening the translator. If you edit the **config** file after opening the translator, the new values are not used.

Lines of the **config** file have format `<parameter_name>=<parameter value>`.

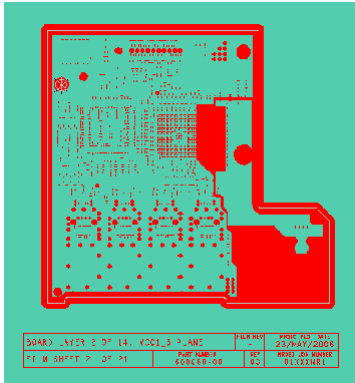
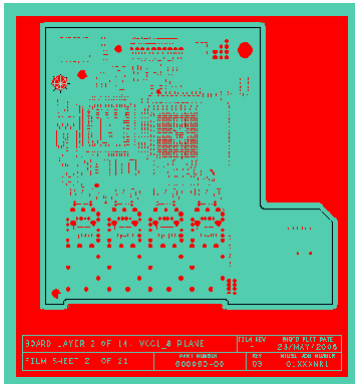
Configuration Parameters

These configuration parameters are relevant to ODB++ Inside for Cadence Allegro:

Configuration Parameter	Type	Default	Explanation
drc_comp_height	Text	drc_comp	Document layer containing component height restriction areas (used to check for components above or below height limits in height restricted areas).
attr_value_correct			Controls how the translator handles illegal attribute values (out of range, etc.) that are input with a design. <ul style="list-style-type: none">• Yes - The attribute is reset to its default value, and a message is written to the log.• No - Translation is halted.
drc_comp_keepin	Text	drc_comp	Document layer containing component keepin areas (to check for components outside keepin areas).

Configuration Parameter	Type	Default	Explanation
drc_comp_keepout	Text	drc_comp	Document layer containing component keepout areas (to check for components inside keepout areas).
drc_pad_keepout	Text	drc_route	Document layer containing pad keepout areas (to check for pads inside keepout areas).
drc_plane_keepout	Text	drc_route	Document layer containing plane keepout areas (to check for planes inside keepout areas).
drc_route_keepin	Text	drc_route	Document layer containing rout keepin areas (to check for traces, planes, pads, vias outside the keepin areas).
drc_route_keepout	Text	drc_route	Document layer containing rout keepout areas (to check for traces, planes, pads, trace-bends inside keepout areas).
drc_tp_keepin	Text	drc_tp	Document layer containing testpoint keepin areas (to check for testpoints outside keepin areas).
drc_tp_keepout	Text	drc_tp	Document layer containing testpoint keepout areas (to check for testpoints inside keepout areas).
drc_trace_keepout	Text	drc_route	Document layer containing trace keepout areas (to check for traces inside keepout areas).
drc_via_keepout	Text	drc_route	Document layer containing via keepout areas (to check for vias inside keepout areas).
eda_cadence_apd_bot_name	Text	base	The name used to indicate the bottom layer in APD files. When Cadence APD files are translated, the default name for the bottom layer is base . If you have APD files that use a layer name other than base for the bottom layer, you can specify an alternate name.
eda_cadence_apd_top_name	Text	surface	The name used to indicate the top layer in APD files. When Cadence APD files are translated, the default name for the top layer is surface . If you have APD files that use a layer name other than surface for the top layer, you can specify an alternate name.

Configuration Parameter	Type	Default	Explanation
eda_cadence_check_package_shape	Boolean	No	<p>If a design is likely to have components with identical package names but different geometries, you can have the translator check the geometry of a component if its package name is identical to that of another component.</p> <ul style="list-style-type: none"> • Yes - If a component has the same package name as another component, the package shapes in the file comps_<product_model>.out are compared. If they are not the same, a new package is created for the second component. The new name is created by adding a suffix consisting of a plus sign (+) and an index number. • No (default) - If a component has the same package name as another component, they are assumed to have the same geometry. No checking is performed.
eda_cadence_copper_layers_from_film	Boolean	No	<p>Controls how to handle a layer in the layer file that is not in the film file.</p> <ul style="list-style-type: none"> • Yes—The following is done: The layer is added as a copper layer Polarity for the layer is taken from the layer file suppress_unconnected_pads and suppress_shape_fill are set to false the layer is assigned the classes ETCH, PIN, and VIA. • No—If the product model has no copper layer, an error is reported.
eda_cadence_delete_sort_pins_file	Boolean	no	<p>Yes - Always delete temporary sort pins file. When a translation has warnings about the pins file, it does not delete the temporary pins file, so that the user can view the warnings. When running from a script, the temporary files accumulate. This parameter lets you specify that the files be deleted even if there are warnings.</p>
eda_cadence_font_file_name	Text	ansi (the supplied font file)	<p>The name of the font file to be used. The file must reside in \$GENESIS_EDIR/all/eda/cadence/fonts. You can provide an alternate font file so that fonts used in ODB++ match the fonts used in Cadence Allegro.</p>

Configuration Parameter	Type	Default	Explanation
eda_cadence_layer_polarity_source	Text	f (film)	<p>Informs the translator to use the suppress shape fill information from the films_XXX.out file or from the layers_XXX.out file.</p> <ul style="list-style-type: none"> • f - films • l - layers (Cadence Allegro only)
eda_cadence_pos_anti_etch	Boolean	no	<p>Controls whether ANTI ETCH surfaces will be negative or positive.</p> <ul style="list-style-type: none"> • Yes - ANTI ETCH surfaces will always be positive. (If the product model has a photoplot outline - positive surface - that covers legend text, the text will not be visible.)  <ul style="list-style-type: none"> • No (default) - ANTI ETCH surfaces will be negative. 
eda_cadence_profile_sym_type	Text	r (round)	<p>Defines symbol type for arcs and lines which are part of step profile polygon.</p> <ul style="list-style-type: none"> • r - round symbol • s - square symbol

Configuration Parameter	Type	Default	Explanation
eda_cadence_silk_fill	Boolean	No	Fills surfaces on Cadence Allegro silkscreen layers. <ul style="list-style-type: none"> • Yes - draws surfaces. • No - draws surfaces as outlines.
eda_cadence_sort_pins_file	Boolean	Yes	Controls whether to sort the pins file before reading it. <ul style="list-style-type: none"> • Yes - sorts the pins file alphabetically before it is read. • No - does not sort the pins file.
eda_cadence_sort_pins_numeric	Text	No	Controls how to sort pins. <ul style="list-style-type: none"> • Yes - sorts the pins file numerically. • No - sorts the pins file textually. • yes_num_last (or any string other than yes or no) sorts the pins file numerically but with numeric pins after pins beginning with a letter.
eda_cadence_sqa_area_layer_name	String	SQA_areas	Defines the layer where an sqa area is saved during translation from Cadence. If not defined, default value sqa_areas are saved.
eda_cadence_support_exceptional_pins	Boolean	Yes	To control whether to add to the comps_XXX.out file pins lacking names and component designation, or whose components do not appear in the file. <ul style="list-style-type: none"> • Yes - (default) adds a component named no_refdes+XX. • No - ignore the pins.
eda_cadence_suppress	Boolean	No	Default value for option Suppress Unconnected Pads on the Additional Parameters dialog box. <ul style="list-style-type: none"> • Yes - perform unconnected pad suppression. • No - do not suppress unconnected pads.
eda_cadence_suppress_shape_fill_setting (obsolete)	Text	f	Obsolete - replaced by the Suppress shape fill option of the Cadence Matrix File Editor .
eda_cadence_therm_err	Boolean	No	<ul style="list-style-type: none"> • Yes - the translation process will abort with a message listing the padstack / thermal names that did not have a match in the models file. (Thermals Mode in Input Parameters must be set to Use File for this configuration parameter to work). • No - does not flag missing thermals.

Configuration Parameter	Type	Default	Explanation
eda_cadence_v14_popup (obsolete)	Boolean	Yes	Obsolete.
gns_pdf_viewing_prog	Text	/sw/bin/acroread - useFront EndProgram	Default program path and arguments to open a PDF file. - used for standalone translators only.

Setting Environment Variables

These environment variables are used to define values used by the translator.

Environment Variable	Default	Mandatory	Explanation
BRD2ODB_TMP	None	Yes — for Windows XP No — otherwise	Location for storing temporary files. If not defined, the translator stores files in the location defined by environment variable APPDATA\translator_logs .

Configuring a Thermal Model

Cadence Allegro Designer (Version 13) does not explicitly define the shape of the thermal pads or the Padflash definitions. Typically, these definitions are deferred until the Gerber wheel apertures are defined. However, to generate accurate board data, ODB++ Inside for Cadence Allegro requires the use of a Thermal Model to explicitly define these shapes.

This chapter discusses the following:

- [Structure of a Thermal Model File](#)
- [BNF \(Backus-Naur Form\) of Rules](#)
- [Sample Thermal Model File](#)

Structure of a Thermal Model File

The Thermal Model File includes a units definition followed by any number of models. Lines preceded by a number sign (#) are treated as comments and ignored.

Line	Explanation
<code>.units < inch mm ></code>	The units definition specifies the measurement units that will be used for the models. Legal values are inch and mm.
<code>.model <name></code>	Each model begins with the model directive. The name is limited to 64 characters that can include letters, digits, and the following symbols: dash (-), underscore (_), period (.), plus (+).

See “[Sample Thermal Model File](#)” on page 42.

BNF (Backus-Naur Form) of Rules

The model directive is followed by a sequence of rules.

```
<rule> ::= <condition> : <derivation>
```

If the condition is met, then the thermal shape definition described in the derivation is used for the pad. The first match is used.

```
<condition> ::= <type> {<equation>}
```

```
<type> ::= PIN | VIA | geometry_name | 'geometry_name'
```

```
<equation> ::= [D|C] ['<' | '<=' | '=' | '>' | '>='] <value>
```

The padstack can be compared by its type (PIN or VIA) or by its padstack name. The padstack name may be enclosed in quotes to accommodate padstacks named PIN or VIA.

The padstack can also be compared based on drill size (represented by the ‘D’ in the equation), or by clearance size (represented by the ‘C’ in the equation).

```
<derivation> ::= NULL | '<sym_name>' | <set_values>
```

```
<sym_name> ::= any ODB++ standard or semi-standard name
```

The shape can be defined directly by providing the symbol name; or the parameters for the symbol can be derived based on the drill and clearance sizes (represented by ‘D’ and ‘C’, respectively).

The ‘NULL’ shape can be used to produce direct connects.

```
<set_values> ::= <id> <id> <tie> <num_ties> <angle> <oshape>  
<ishape> <style>
```


`<id> ::= diam_value`

`<od> ::= <diam_value>`

`<diam_value> ::= [C | D] ["+" | "-"] [<value>]`

The inner diameter is represented by **<id>** and the outer diameter is represented by **<od>**.

`<tie> ::= <value>`

`<num_ties> ::= <value>`

<tie> is the size of the tie and **<num_ties>** is the number of ties.

`<angle> ::= <value>`

<angle> represents the start angle for the first tie in degrees.

`<oshape> ::= R | S | C`

`<ishape> ::= R | S | C`

`<style> ::= R | S`

- **<oshape>** represents the outer shape. It can be either Round ('R'), Square ('S') or the same shape as the clearance ('C').
- **<ishape>** represents the inner shape. It can be either Round ('R'), Square ('S') or the same shape as the clearance ('C').
- **<style>** can be Rounded ('R') or Squared ('S').

Sample Thermal Model File

This is a sample thermal file:

```
.units inch
.model allegro_model
# Direct replacement of symbols

# Replace the padflash named "TH05" with a round
# clearance of 5 mils.
TH05: 'r5'

# Replace the padflash named "T165X145X20X45" with
# a square thermal with an outer diameter of 165 mils, inner
# diameter of 145 mils with four ties each of 20 mils, first
# starting at 45 degrees.
T165X145X20X45: 'ths165x145x45x4x20'

# Replace the padflash named "5MIL" with a direct
# connection
5MIL: 'null'

# calculated values

# Place a direct connect for all VIA pads
VIA: NULL

# For pins with a clearance less than or equal to 45 mils,
# place a rounded thermal with outer diameter the size of the
# clearance inner diameter 20 mils smaller, 4 ties of 20 mil
# starting at 45 degrees outer and inner diameters shaped as
# the clearance
PIN C<=45 : C C-20 15 4 45 C C R
```

Generating the Extraction Files

To perform the translation, ODB++ Inside for Cadence Allegro makes use of the following extract files. The extraction files are generated using `$ALLEGRO_BRD2ODB/valor_ext.il` skill code.

<product_model> represents the name of the current product model.

For CAD layers to be present in the generated ODB++, you must create the films for those layers.

- `comps_<product_model>.out`
- `films_<product_model>.out`
- `geoms_<product_model>.out`
- `layers_<product_model>.out`
- `nets_<product_model>.out`

- [pads_<product_model>.out](#)
- [pins_<product_model>.out](#)
- [props_<product_model>.out](#)
- [tech_<product_model>.out](#)

[comps_<product_model>.out](#)

Contains the outline shape of the components. In ODB++, components must each be defined as a single closed shape, therefore the `PLACE_BOUND` outlines are typically used.

Pins lacking names and component designations, or whose components are not found in the `comp_<product_model>.out` file, might be either mechanical pins or testpoints, and should not be ignored. The `eda_cadence_support_exceptional_pins` configuration parameter handles such pins.

If a design is likely to have components with identical package names but different geometries, you can have the translator check the geometry of a component if its package name is identical to that of another component. If configuration parameter `eda_cadence_check_package_shape` is set to **yes**, the package shapes in the file `comps_<product_model>.out` are compared for components having identical package names. If a component is found to have a different geometry from a component with the same package name, a new package name is created for the component. The name is generated from the existing package name by adding a suffix consisting of a plus sign (+) and a number. See [“Setting Configuration Parameters”](#) on page 34.

[films_<product_model>.out](#)

Contains the artwork information from Allegro.

The following rules are used to determine which film to apply to a layer. If these rules do not produce the desired results, you can create a matrix file to define the layers. See [“Editing the Matrix File”](#) on page 18.

- If there is only one film in the file with class = ETCH and subclass = <layer name>, this film is used.
- Otherwise, if there are multiple films with ETCH!<layer name>, and one of the films has the same name as the layer, this film is used. (A warning is issued if there are multiple films with ETCH!<layer name>.)
- Otherwise, if only one of the films has only one ETCH!<layer name> class, this film is used.
- Otherwise, the following films are rejected, and the first remaining film in the file having ETCH!<layer name> is used:

- a film with more than ten classes
- a film with keepin / keepout classes
- a film with silkscreen / soldermask / solderpaste classes

See “[layers_<product_model>.out](#)” on page 44.

If there is a layer in the layer file that is not listed in the film file, you can use configuration parameter **eda_cadence_copper_layers_from_films** to translate the layer. If **eda_cadence_copper_layers_from_films = yes**, and there is a layer in the layer file but not in the film file, the following is done:

- The layer is added as a copper layer
- Polarity for the layer is taken from the layer file
- **suppress_unconnected_pads** and **suppress_shape_fill** are set to false
- the layer is assigned the classes ETCH, PIN, and VIA.

Because an empty films file causes the translation to fail, the **valor_ext.il** import script checks whether **films_<product_model>.out** is empty and prompts the user to check the **artworks** file and extract again.

geoms_<product_model>.out

The graphical data describing feature placement.

The extraction program **valor_ext.il** adds the field FILLET in the geoms file. The attribute **.tear_drop** is added to components having **CLASS = ETCH** and **FILLET = yes**.

layers_<product_model>.out

Provides information on the order of the physical layers.

For an explanation of how films listed in the film file are added to the appropriate layers, see “[films_<product_model>.out](#)” on page 43.

nets_<product_model>.out

This file is optional and is not required for translation. It contains two types of information on nets: classes and properties. Allegro declares three types of class: spacing, physical, and electrical. Every net may connect/have any combination of triplet of spacing, physical, and electrical classes, if any. The classes are defined in the technology file. It also contains some net properties, such as impedance.

The property **NO_TEST** was added to the **nets_<product_model>.out** file. A net with a value of **Yes** for this property will have the attribute **.testpoint_count** set to **0**.

pads_<product_model>.out

Contains information on the padstacks in the product model.

The Allegro BRD file currently does not contain information on the Padshapes. Therefore, the names that appear in the Padflash field, must be defined with the Thermal Model. (See [“Configuring a Thermal Model”](#) on page 39.)

pins_<product_model>.out

Contains information on pins and vias.

The translator reads the pins in order of appearance. Therefore, the extract pins file **must** be sorted in ascending order according to pin number. This is performed automatically when invoked from Cadence Allegro.

Two fields added to **pins_<product_model>.out** during translation support the V15.2 drill tolerances of Cadence Allegro. They are **DRILL_HOLE_POSTOL** and **DRILL_HOLE_NEGTOL**. They enable reading maximum and minimum drill tolerance values.

props_<product_model>.out

Contains additional component property information. This file is optional. This was originally added so that users could read additional component properties directly into ODB++. Users requiring the extraction of additional properties can manually add them to the view file.

tech_<product_model>.out

This file is an optional ASCII file that contains Allegro/APD parameter and constraint data. By using the tech file, a user can apply a uniform set of design rules and constraints to designs that share the same set of design rules and constraints.

- User Units
- Drawing Parameters
- Layout Cross Section Parameters
- Spacing Constraints (Part of the spacing constraints are the clearances rules.)
- Physical Constraints
- Electrical Constraints

- User Property Definitions

From Cadence Allegro version 16.0, tech files are generated in XML format. ODB++ Inside for Cadence Allegro can identify whether the technology file is of the new XML-based format or of the old format, and can read the file in either format.

Using Command Line Parameters

You can run ODB++ Inside for Cadence Allegro from the command line.

Usage: **brd2odb** [parameters] <full path to input brd file>

Switches are preceded by a dash. Some switches accept parameters. Parameters must be separated by spaces. An unrecognized switch is ignored. If you are working in console mode (the **-gui** switch as not been set) missing or incorrect parameters cause the program to terminate.

Parameter	Description
-a2l -append2log	Appends log messages to existing log file log_brd2odb . By default, the new log file overwrites any existing log file for each translation.
-bb	When Suppress Unconnected Pads is checked, the suppression does not suppress pads on top/bottom edge of blind/buried drills. Equivalent to Don't suppress pads on top/bottom in the GUI.
-c <outline> -component <outline>	Where <outline> can be either p[lacebound] or a[ssembly] . Determines whether PlaceBound or Assembly geometries are used for the component outline. PlaceBound is recommended. Because a component must be a single closed polygon, in cases of ill-defined outlines (commonly seen in the assembly geometries), the limits (bounding box) of the defining features are used for the outline. Default is -c placebound (the bounding box).
-cfg [<config file>]	Read configuration file. If <config file> is not specified, the default name is \$ALLEGRO_BRD2ODB/config . The configuration file used by the translator can only be changed by using a line mode command. The configuration file must be in the same directory as the program executables.
-d -delete	By default, all intermediary files are saved. With this option set, the source extract files are deleted. If they are not deleted, and the -gz (zip) parameter is used, the extract files are automatically compressed.

Parameter	Description
-fi	When Suppress Unconnected Pads is set, the suppression suppresses only fully isolated pads. Without this switch, pads are considered to be unconnected in these cases: <ul style="list-style-type: none"> • - a single totally isolated pad • - two pads touching or intersecting • - a pad transversed by a trace not through its center • - a pad touching a surface where its center is not inside the surface Equivalent to setting Fully isolated pads in the GUI.
-gui	Starts the GUI version of the BRD2ODB translator. The default is without a GUI.
-gz -gzip	Zip ODB++. An ODB++ product model is made up of a hierarchy of subdirectories containing files describing the product model. For convenience, the product model can be tared and gzipped into a single <i>tgz</i> file.
-help	Lists the help switches in the console window.
-hg -help_gui	Displays online help.
-iff	When Suppress Unconnected Pads = Yes, the suppression ignores the Allegro FIXFLAG setting. Equivalent to setting Ignore FIXFLAG in the GUI.
-ijp <path>	Input (Allegro) design path. If none specified, uses the default (current working directory).
-jn <product_model>	Output ODB++ product model name. If none specified, uses default name odbjob .
-jp <product_model_path>	Output path for the ODB++ product model. If none specified, uses the default (current working directory).
-lp <log_path>	Log file path. Default is output product model path.
-m <tolerance> -match <tolerance>	Where <tolerance> is the number of mils for symbol tolerance. As new symbols are encountered, they are compared to symbols already generated during the same session. If the shape is within the specified tolerance, the previous shape is reused. The default value is 0.2 inches (200 mils).
-matrix_file <matrix_file_full_path>	Designates the full path to the matrix file. Can only be edited from the GUI.
-net_none_flag	Does not assign \$NONE\$ net to features with no net.
-nn -neut_nets	Neutralizes Nets names (generates random net names).
-no_view	Runs the translator without opening the ODB++ viewer.

Parameter	Description
-o <dist> -outline <dist>	Where <dist> is the number of mils to extend the outline on negative planes. This parameter corresponds to the "-o" option of the Cadence Allegro artwork program. The default value is 0.1 inches (100.0 mils)
-p <mode> -padflash <mode>	Where <mode> is either s[ubstitute] or i[gnore]. Determines whether to substitute Padflash definitions using the thermal modeling file. The default is -p ignore . Default is ignore.
-pst <option> -profile_symbol_type <option>	Defines the symbol type of lines and arcs describing step profile: r(ound)/s(quare). Set eda_cadence_profile_sym_type with <option>.
-p_fab	Specifies that only fabrication data is to be written to ODB++ output. Equivalent to selecting Export Option = FAB in the user interface.
-p_assem	Specifies that only assembly data is to be written to ODB++ output. Equivalent to selecting Export Option = ASSY in the user interface.
-rc -round_corners	Corners of polygons (contours) will be rounded.
-read_sqa <option>	Determines whether to create the signal quality layer. <option> = [yes/no] Default = No .
-re -remove_eda	Removes EDA data (components & packages).
-r -read_drc	Read DRC layers.
-sf	Turn eda_cadence_silk_fill on.
-skip_refdes <option>	Skip components with RefDes containing asterisk characters (*). Options are yes/no/part .
-sp	Turn eda_cadence_suppress on.
-te	Turn eda_cadence_therm_err on.
-tf <thermal_file>	Where <thermal_file> is the full path name for the thermal model file. If the thermal_file is specified, the thermal_model (see parameter -tm) must also be defined. If no thermal file is specified, a default model is used, which uses direct connect and no thermals.
-tm <thermal_model>	Where <thermal_model> is the name of the model to be used. The available model names are defined in the thermal model file. If no thermal_file is specified, the thermal_model is ignored.
-tr_sym	Controls the translation of symbols. yes - Translate symbols as components. If there are multiple shapes with the same name, each will be translated as a separate component. no (default) - Do not translate symbols.

Parameter	Description
-ups	Disables the options Ignore FIXFLAG , Don't suppress pads on top/bottom , and Fully isolated pads . If you access ODB++ Inside for Cadence Allegro from Allegro, and you have used the Allegro V16.2 options for suppressing unconnected pads, this switch is set automatically. If you access ODB++ Inside for Cadence Allegro from the command line, you do not need this switch.
-verify	Requests verification from the user before performing various actions such as Save and Translate.
-v -ver	Displays version information about the translator. When this option is used, all other parameters are ignored.

Supported Features

The following features are included in the translator.

- [Skipping Extraction of Net Impedance Average](#)
- [Placing Components Underneath Raised Components](#)
- [Using Height Parameters for Keepout Areas](#)
- [Back-Drill Information is Translated](#)
- [Support for Mirrored Padstacks](#)
- [Support for the COMPONENT KEEPOUT Class](#)

Skipping Extraction of Net Impedance Average

During export from Allegro, the attribute **NET_IMPEDANCE_AVERAGE** is calculated for each net even though the attribute is not used in any vSure analysis. This slows down extraction considerably.

The **valor_ext.il** import script prompts for permission to skip this calculation. As a result, extraction time is reduced.

Placing Components Underneath Raised Components

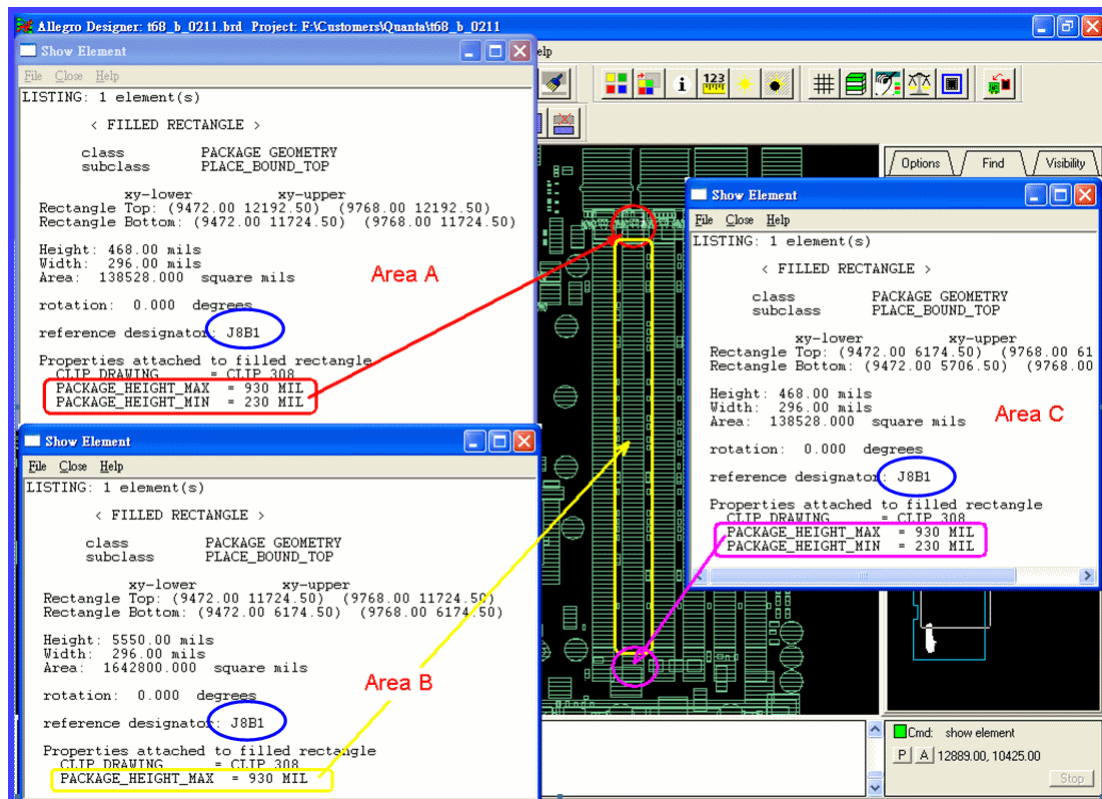
If a component is defined in Allegro as having a value for property **package_height_min** (the amount of space under the component) or if there are areas of the component with values for **package_height_min**, this information is stored in ODB++ output and can be used in ODB++ Inside for Cadence Allegro during component analysis.

The height of the component, as defined in the Allegro property **package_height_max**, is stored in the ODB++ component attribute **.comp_height**.

A layer (**height_top**) is created in ODB++ to store height information for areas where there is space underneath components. In this layer, the maximum height of components that can be placed in a particular area is defined in the ODB++ feature attribute **.drc_max_height**. This attribute is set to the value of **package_height_min** for components (or areas of components) where **package_height_min** is specified.

In the example, RefDes J8B1 is a component with three areas defined:

- The main part of the component (area B in the example) is resting on the board, so it has no value for **package_height_min**.
- At the two ends of the component (areas A and C in the example), there is a space of height 230 MIL underneath. A component that is placed under an end area of this component is not reported as an error if its height is less than 230 MIL.



Using Height Parameters for Keepout Areas

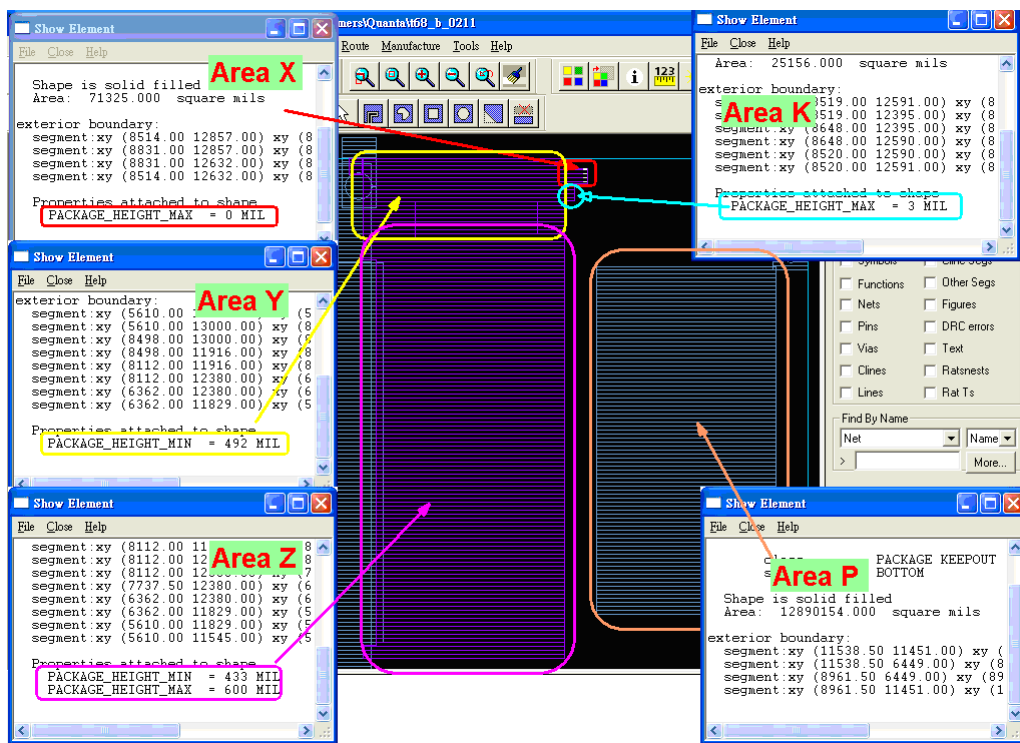
The parameters **package_height_min** and **package_height_max** are interpreted to match their meaning in Cadence Allegro.

package_height_min—the lower limit of the height of the keepout area. If this value is specified for a keepout area, only components with height less than this value can be placed in this area.

package_height_max—**package_height_max** is not considered when **package_height_min** is specified. If **package_height_min** is not specified, this parameter is used to indicate whether all components or no components can be placed in the area, regardless of their height:

- **package_height_max = 0** means any components can be placed in this area
- **package_height_max > 0** means no components can be placed in this area

To determine the maximum height of components that can be placed in an area, the ODB++ attribute **.drc_max_height** is set to **package_height_min** if **package_height_min** is specified.



The example shows the following areas:

Area	package_height_min	package_height_max	components can be placed	reason
X	not specified	0	all	package_height_max = 0
Y	492 mil	not specified	height < 492 mil	
Z	433 mil	600 mil	height < 433 mil	package_height_max is not considered when package_height_min is specified
K	not specified	3 mil	none	package_height_max > 0
P	not specified	not specified	none	neither property is specified

Support for CLASS_CONSTRAINT_REGION

ODB++ Inside for Cadence Allegro supports class type CLASS_CONSTRAINT_REGION that was added to Cadence Allegro version 16.

Back-Drill Information is Translated

If the Cadence Allegro design contains back-drill information, new drill layers are created, for each drill span, to include this information.

Cadence Allegro version 15.7 implements back-drilling via the net property **BACKDRILL_MAX_PTH_STUB**, with the value denoting the maximum depth of the back-drill.

During translation, backdrills are added for pins/via holes and to existing drills. The span cannot be from top to bottom but must start or end with the top/bottom. A new layer is added for each backdrill span.

Support for Mirrored Padstacks

If a via is mirrored, or if a pin is used for a component on the bottom of the board, padstack information is taken from the mirrored layer.

Support for the COMPONENT KEEPOUT Class

The COMPONENT KEEPOUT class works like the PACKAGE KEEPOUT class.

Release Notes

Features and resolved issues are listed for these versions:

- [Version 9.0 Features and Resolved Issues](#)
- [Version 9.1 Features and Resolved Issues](#)
- [Version 9.2 Features and Resolved Issues](#)
- [Version 9.3 Features and Resolved Issues](#)

Version 9.0 Features and Resolved Issues

These issues were resolved in this version.

ID	Issue
9419	There was no way to specify that component outlines should be taken from user-defined classes. See the Component Outline parameter in “Specifying Parameters Specific to Cadence Allegro” on page 10.
14875	Translation failed if the net properties file was very large.
16797	The COMPONENT KEEPOUT class was not recognized as defining a keepout area.
16855	The dimensions of the profile were incorrect.
16878	FILLETS from Cadence Allegro V16.2 were not assigned the .tear_drop attribute during translation.
16342 SF 8413, SF 8665	Features were missing when they were defined in a mirrored padstack. See “Support for Mirrored Padstacks” on page 52.
16900 - SF 8715	Translation failed during profile creation.
SF 11342	Application crashed during translation.
SF 11541	If there are components with both ASSEMBLY_TOP and ASSEMBLY_BOTTOM, or PLACE_BOUND_TOP and PLACE_BOUND_BOTTOM, how does the translator decide on the placement side?
SF 8032	Slot figures were missing from the translation.
SF 8664	If a drill is OBLONG_Y and its size in the pins file does not match its size in the padstack, the drill slot was sometimes rotated incorrectly relative to the copper layer pads.
SF 8758	ODB++ Inside for Cadence Allegro could not be run on Linux RH5.

Version 9.1 Features and Resolved Issues

Designs from Cadence Allegro V16.3 can be translated to ODB++.

The **Allegro Export ODB++** dialog box has been enhanced.

You can display the resulting ODB++ file in the ODB++ Viewer.

Different AIF formats are supported. (ID = 705745)

A new cavity class is supported (ID = 744654)

These issues were resolved in this version:

ID	Title
705391	A problem with Minimum Comp Height DFM category.
705816	Cadence Wirebonding Translation issue with Config parameter eda_cadence_copper_layers_from_films
705819	Line is missing in log after second translation.
705935	A problem with Cadence Allegro EDA Translation.
706074	ODB is inconsistent with source file (Cadence APD)
706181	A problem with Cadence Allegro EDA input.
706207	netpoint is on cadnet.top and bot instead of tru
706251	ODB++ Inside parameter problem when run in command line
707590	ODB++Inside and problems with backdrilling
712910	Area rules not rotating with part
717567	A problem with Cadence Allegro EDA input.
722970	brd2odb v8.2 does not work on linux 64 bit
725459	Only 8 decimal places accuracy when downgrading to odb++v6.0
734546	Problems with partial ODB++
741482	vSure help about window
746075	Chamfered Pads will be shown as rectangle Pad in ODB++; Export ODB++ and Import to VUV8.2 shows wrong results.
749144	ODB++ is not generating the Drill Table correctly.
756289	dofile issue - CLASS COMPONENT GEOMETRY and SUBSTRATE GEOMETRY are not displayed
757114	Generated ODB++ output file contains incorrect Thermals.

ID	Title
757748	Features are wrong in the translated board
759390	Problem when installing ODB++ Inside for Allegro 9.1
759615	SAT - BRD2ODB ODB++ Inside 9.1 ODB++ viewer shows incorrect image of top copper layer
760727	"Fatal error occurred while exporting design" when exporting ODB++ with decal custom thermals set to 8 spokes
762522	A problem with Pad Suppression in Allegro ODBI 9.1

Version 9.2 Features and Resolved Issues

ODB++ Viewer can read TGZ files (ID = dts100796685)

These issues were resolved in this version:

ID	Title
dts0100747607	APD 15.5.1 cannot extract the data using valor_ext.il (for VALOR_NPI)

Version 9.3 Features and Resolved Issues

The Graphic User Interface has been enhanced.

ODB++ Viewer can read TGZ files (ID = dts100796685)

ODB++ Viewer can display Step and Repeat tables (ID - dts100805216)

These issues were resolved in this version:

ID	Title
dts0100704940	Fixed fill surface issue.
dts0100748565	Pads with chamfered corners are included in the resulting ODB++ file.
dts0100754139	Fixed issue.
dts0100776652	Fixed error when using the translator.
dts0100791118	Components from top side do not now fall to the bottom side.
dts0100802150	Fixed component keepout issue.
dts0100829891	Suppress unconnected pads correctly handles connected pads.
dts0100857527	Extracted files from Allegro 15.5.1 can now be translated

ID	Title
dts0100873791	Fixed zero Mil rout width issue.
dts0100880260	Fixed issue.
dts0100898527	Added support for mirror padstacks.
dts0100902941	Fixed pad offsets.
dts0100902951	Fixed arc handling.

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 FLEXIm 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

