

Allegro® X ECAD-MCAD Library Creator - Frequently Asked Questions

Product Version 23.1

September 2023

© 2023 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Allegro PCB Editor contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522. Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission. All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

<u>Frequently Asked Questions</u>	5
<u>What is a package in Library Creator?</u>	5
<u>Can I import an existing Allegro PCB Editor library of footprints to Library Creator?</u> ..	5
<u>Can I generate 3D STEP models using existing Allegro X PCB Editor library footprints?</u>	6
<u>What is the difference between a package and a footprint in Library Creator?</u>	6
<u>Does Library Creator work with older versions of Allegro X PCB Editor or just 23.1?</u> ..	6
<u>Can I customize Library Creator to our company standards for symbol creation?</u>	6
<u>What IPC standards does Library Creator conform to?</u>	6
<u>Is concurrent creation of IPC-735-compliant IPC-A, IPC-B and IPC-C footprints possible?</u>	7
<u>Does Library Creator support THT and SMT components?</u>	7
<u>Which STEP model formats can I import?</u>	7
<u>Which STEP model formats can I export?</u>	7
<u>Can I add Library Creator into our company MCAD tool?</u>	7
<u>Does Library Creator work with PLM systems?</u>	7
<u>What units of measurement does Library Creator support?</u>	8
<u>Can I assign different colors to STEP models?</u>	8
<u>Can I work with Library Creator when not connected to the company network?</u>	8

Allegro X ECAD-MCAD Library Creator - Frequently Asked Questions

Frequently Asked Questions

This document lists answers to frequently asked questions about Allegro X ECAD-MCAD Library Creator. This document will be periodically updated to resolve the issues our customers face while working with Allegro Library Creator.

What is a package in Library Creator?

A package in Library Creator is a model of a physical component with sufficient detail and accuracy to support both Printed Circuit Board (PCB) layout and MCAD design requirements. In addition to 3D geometry, a package contains body and terminal features, 2D contact regions, pin numbers, lead form attributes, and tolerances.

Can I import an existing Allegro PCB Editor library of footprints to Library Creator?

Existing footprints from Allegro PCB Editor can be imported into Library Creator. You can easily automate footprint updates such as changing solder mask, solder paste pullback, and adding keepouts between or around pins within Library Creator by applying rules.

Note the following before importing an existing Allegro X PCB Editor footprint:

- Automated footprint updates might not be possible for existing PCB Editor footprints with no data. For example, the standard IPC-7351 padstack rules require details of the terminal contact areas and tolerances. Because these details are not typically available in an existing Allegro PCB Editor footprint, it is not possible to automatically generate a new IPC-7351 padstack for this Allegro PCB Editor footprint without manually adding the required information.
- Library Creator supports reading and writing of the Lead Form and Contact Geometry from Allegro Lead Editor. For Allegro footprints with valid lead data, it might be possible to generate new IPC-7351 padstacks in Library Creator.

Can I generate 3D STEP models using existing Allegro X PCB Editor library footprints?

It is not possible to automatically generate 3D STEP models from existing Allegro footprints. However, you can search for and use common electronic packages that can be quickly associated with existing Allegro PCB Editor footprints from the thousands of 3D package models available in the Allegro ECAD-MCAD Library Creator repository.

Existing 3D packages can be modified, and new 3D models can be created from existing templates or import into Library Creator to address non-standard components.

What is the difference between a package and a footprint in Library Creator?

A footprint contains the design elements required to fabricate and assemble a PCB that includes a particular package. The details of a footprint are dependent on design rules and manufacturing technology. In Library Creator, rules can be applied to a package to generate a footprint that satisfies specific design and manufacturing requirements.

Does Library Creator work with older versions of Allegro X PCB Editor or just 23.1?

Library Creator is available with all the versions of Allegro XPCB Editor that Cadence supports.

Can I customize Library Creator to our company standards for symbol creation?

Yes, the footprint generation rules in Library Creator can be extensively customized to support internal standards.

What IPC standards does Library Creator conform to?

Library Creator is compliant with IPC-7351B, which is the current published revision for Surface Mounted Technology (SMT) land pattern generation. Through Hole Technology (THT) padstack rules based on an unpublished draft of the IPC-7251 standard are also included in Library Creator.

Is concurrent creation of IPC-735-compliant IPC-A, IPC-B and IPC-C footprints possible?

Yes. You can generate multiple footprints for a single package by applying the applicable footprint generation rules. This process can be automated for multiple packages and multiple rules using the [Allegro Bulk Export](#) utility in Library Creator.

Does Library Creator support THT and SMT components?

Yes. footprint generation rules in Library Creator can be applied to both SMT and THT packages as well as packages containing both SMT and THT pins.

Which STEP model formats can I import?

Library Creator can import industry standard AP203 and AP214 MCAD models. Library Creator can also import AP210 which is a specialized STEP extension for electronics that includes the package structure and footprint elements of a Library Creator package.

Which STEP model formats can I export?

Library Creator can export AP203 which can be used in Allegro X PCB Editor and MCAD tools not supported by a direct plug-in. Library Creator can also export AP210 which is a specialized STEP extension for electronics, which includes the package structure and footprint elements of a Library Creator package.

Can I add Library Creator into our company MCAD tool?

Yes. You can use the Library Creator MCAD native plug-ins to push 3D models to PTC Creo and Siemens NX. Other MCAD tools can be supported through STEP data exchange.

Does Library Creator work with PLM systems?

MCAD models generated by Library Creator can be automatically stored in PTC Windchill and Siemen's Teamcenter when used in the context of the respective MCAD plug-ins. Library Creator models can be used in combination with other Product Lifecycle Management (PLM) systems through a manual check-in process.

What units of measurement does Library Creator support?

Library Creator supports Mils, Inches, Microns, Millimeters, and Centimeters. Library Creator footprints can be imported into and exported from Allegro PCB Editor in same units supported by Allegro PCB Editor.

Can I assign different colors to STEP models?

Yes, you can specify top-level colors or colorize elements individually. You can also use Source Colors (original colors inherited from the model author) or Feature Colors (colors set by the user based on different criteria, such as body type, and so on).

Can I work with Library Creator when not connected to the company network?

No. You must be connected to the company network. The Library Creator server is a centralized database and repository for all rules, templates, and packages used by the application. Because Library Creator must be connected to this server at all times, you must be connected to the company network.