



Allegro X ECAD-MCAD Library Creator Tutorial

Product Version Product Version 23.1

September 2023

© 2023 Cadence Design Systems, Inc.
Printed in the United States of America.

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

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.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. (Cadence) 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.

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 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

1		8
About the Tutorial		8
Audience		8
Prerequisites		8
Setup the Tutorial Database		8
Login to the Library Creator Repository		9
2		11
Introduction to Library Creator		11
Understanding the Basics of Library Creator		11
Repository		11
Footprint		11
Workspace		12
Package		12
Launching Library Creator		12
Adding Rules to Apply Menu		12
Deleting Rules from Apply Menu		13
3		14
Creating a Footprint from Packages		14
What You Will Learn		14
Loading Package		14
Applying Rule to the Package		18
Exporting the Footprint to Allegro PCB Editor		22
Summary		24
4		25
Creating a Footprint from Templates		25
What You Will Learn		25
Accessing the Package Templates		25
Creating a New Instance of Template		26
Applying Rules to Template		29
Saving the Footprint		30
Exporting Footprint to Allegro PCB Editor		32

5		33
Creating a Footprint from Package Templates		33
What You Will Learn		33
Loading a Package		33
Creating a New Instance of Package		35
Saving the Footprint		38
Applying Rules to Package		38
Exporting Footprint to Allegro PCB Editor		39
Summary		40
6		41
Creating Custom Footprints		41
What You Will Learn		41
Loading the Template		41
Creating a New Instance of Template		42
Editing Thermal Pads		43
Changing Thermal Pad Shape		49
Updating Thermal Contacts		59
Saving the Footprint		61
Applying Rules to Package		61
Exporting Footprint to Allegro PCB Editor		62
Summary		63
7		64
Creating Footprints from a STEP Model		64
What You Will Learn		64
Importing STEP Models		64
Editing Seating Pane		65
Adding Contact Features		66
Assigning Terminal Types		70
Assigning Model Height		70
Assigning Pin Numbers		70
Editing the Origin		71
Applying Rules		72
Saving the Footprint		73
Reusing Existing Padstacks		73
Importing External Padstack		74

Replacing Pads with Imported Padstack	75
8	77
Creating Footprint of a Mechanical Part	77
What You Will Learn	77
Importing a Step Model	77
Editing Seating Pane	79
Renaming Features	80
Editing the Origin	81
Measuring Thickness	82
Applying Rules	83
Creating 3D Placebound Shapes	84
Adding Section Offsets	85
Saving the Footprint	88
Exporting Footprint to Allegro PCB Editor	88
9	89
Adding STEP Model to Existing Allegro PCB Editor Footprint	89
What You Will Learn	89
Importing Footprints	89
Checking STEP Model	90
Searching Packages	91
Importing STEP Models	91
Adding Contact Features	92
Assigning Terminal Types	96
Assigning Model Height	96
Editing the Origin	98
Assigning Pin Numbers	98
Saving the Footprint	101
Merging Package with Footprint	101
10	104
Synchronizing Libraries	104
What You Will Learn	104
Synchronizing Existing Footprints with Packages	104
Importing Footprints	104
Searching Packages	106
Comparing Packages and Footprint	107

Merging Package with Footprint	108
Checking Consistency of Footprints with Rules	111
Adding Packages to Repository	115
Importing the STEP Model	115
Editing Seating Pane	116
Adding Contact Features	116
Saving the Footprint	122
Synchronizing Non-Standard Footprints	122
Summary	124
Exercise	125
11	126
Creating Multi-Grid BGA Packages	126
What You Will Learn	126
Accessing the Package Templates	126
Creating a New Instance of the BGA Template	126
Adding Pins	127
Saving the Footprint	130
12	131
Creating Non-Standard Packages	131
What You Will Learn	131
Creating Packages	131
Adding Pins to Package	131
Adding Side Contacts	135
Assigning Pin Numbers	138
Saving the Footprint	138
Applying Rules	138
Adding Orientation Mark	139
Exporting Footprint to Allegro PCB Editor	141
13	142
Creating Footprints from Scratch	142
What You Will Learn	142
Importing a Step Model	142
Editing Seating Pane	142
Creating Holes	143
Saving the Footprint	152

Applying Rules to Package	153
Exporting Footprint to Allegro PCB Editor	154
Summary	155
14	156
Changing Package Colors	156
What You Will Learn	156
Loading Packages	156
Changing Primary Pin Color	156
Changing Body Color	160
15	166
Creating a New Library Using Bulk Export	166
What You Will Learn	166
Summary	170
16	171
Editing Existing Rules	171
What You Will Learn	171
Creating Copy of Rules	171
Editing Variable Rules	172
Editing Geometry Procedures	173
Testing Edited Rules	175
Publishing Rules	176
Summary	177

About the Tutorial

This tutorial familiarizes you with Allegro® X ECAD-MCAD Library Creator. You can go through the steps mentioned in this tutorial to perform the basic tasks that are involved in creating a footprint, in sequence.

Audience

This document is intended for the first time users of Allegro X ECAD-MCAD Library Creator. If you are a PCB librarian, an ECAD or MCAD designer, you might want to generate advanced footprints. Library Creator allows you to accurately create and maintain accurate and efficient footprints.

Prerequisites

Before you start to run the tutorial, ensure that you do the following:

1. [Setup the Tutorial Database](#)
2. [Login to the Library Creator Repository](#)

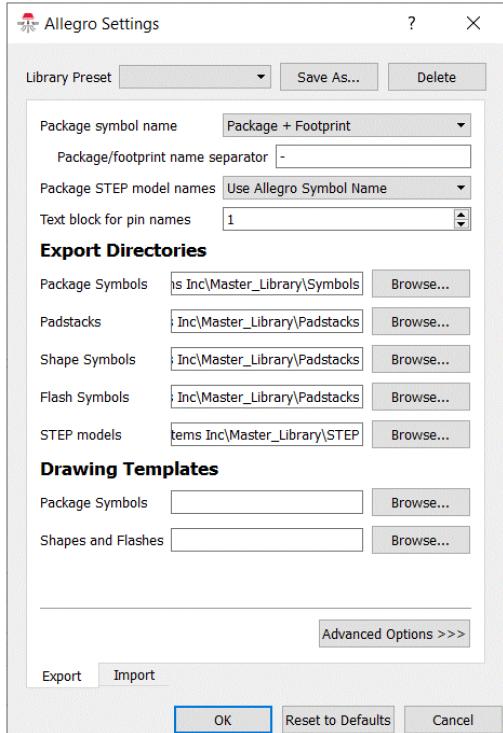
Setup the Tutorial Database

For this tutorial to work correctly, set the paths for the Package Symbols, Padstacks, Shape and Flash Symbols, and STEP Models. To do this:

1. Extract the file to a location where you have write permissions. You can copy this file from the following location:
`<your_install_dir>/doc/lc_tut/examples`
2. In Library Creator, choose *Tools - Allegro Settings*.
The *Allegro Settings* dialog is displayed.
3. Set the path in the *Package Symbols* field under the *Export Directories* section to the *Symbols* folder in the following location:

<your_install_dir>/doc/lc_tut/examples/Master_Library

4. Similarly, set the path in the Padstacks, Shape Symbols, Flash Symbols, and STEP models fields to their respective folders.



5. Set the paths of the *Padstacks* and *Symbols* folders in the *Setup – User Preferences – Paths – Library – padpath* and *psmpath* of *Allegro PCB Editor*.

Login to the Library Creator Repository

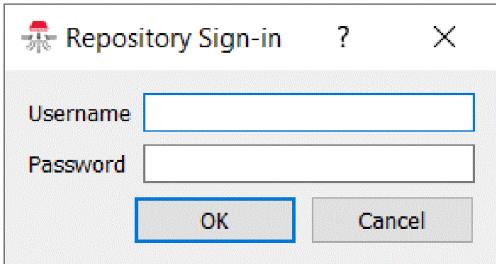
To access the Library Creator repository, you must be logged-in to the repository account. You can login and logout of the repository, change password, and view status from the *Repository* menu. Also, some of the exercises in this tutorial require additional user permissions to the repository. For example, to create, view, and edit a package, you would need permissions to create, view, and edit in the Library Creator repository. If your company is using:

- a Cadence Hosted Repository server, contact *lc_admin@cadence.com* to get the login details or change the repository permissions.
- a locally installed repository server, contact the administrator of the repository to get the login details or change the repository permissions.

To check which permissions you have, choose *Repository - Status*.

To login to Library Creator repository:

1. Choose *Repository – Sign In/Out* to login to the Library Creator repository.
Repository Sign-in dialog is displayed.



2. Enter the login details in the *Username* and *Password* fields.
3. Click *OK*.

A success message is displayed.

For more information about the Library Creator repository, refer to the [*Repository*](#) section of the [*Library Creator User Guide*](#).

Introduction to Library Creator

Allegro ECAD-MCAD Library Creator combines a local or on-line repository containing thousands of detailed package models with unique capabilities to leverage existing 3D models and create new packages from over one hundred parametric templates. Allegro ECAD-MCAD Library Creator enables PCB librarians, ECAD, and MCAD designers to improve the accuracy and efficiency of creating and maintaining high-fidelity component models to serve PCB layout and MCAD design requirements.

Understanding the Basics of Library Creator

In this section, you will learn about the basic terms used in Library Creator to help you understand the Library Creator environment.

Repository

The Library Creator repository enables centralized configuration-controlled storage and management of package models. Package models may be instances of parametric templates or packages created from a featurized external model. Multiple users within a single organization share access to a common set of packages through an account. Additionally, users have access to templates and package instances provided by the system. In order to access the repository, a user must be signed in as an authorized user of a valid repository account.

Footprint

A footprint in Library Creator consists of an origin, padstacks, padstack placements, and shape, elements containing both geometry and text. Footprint details can be inspected or edited through the *Explorer* panel and the *Footprint (2D)* tab. The visibility of various footprint elements and layers is controlled through the *Layer Control* panel.

Workspace

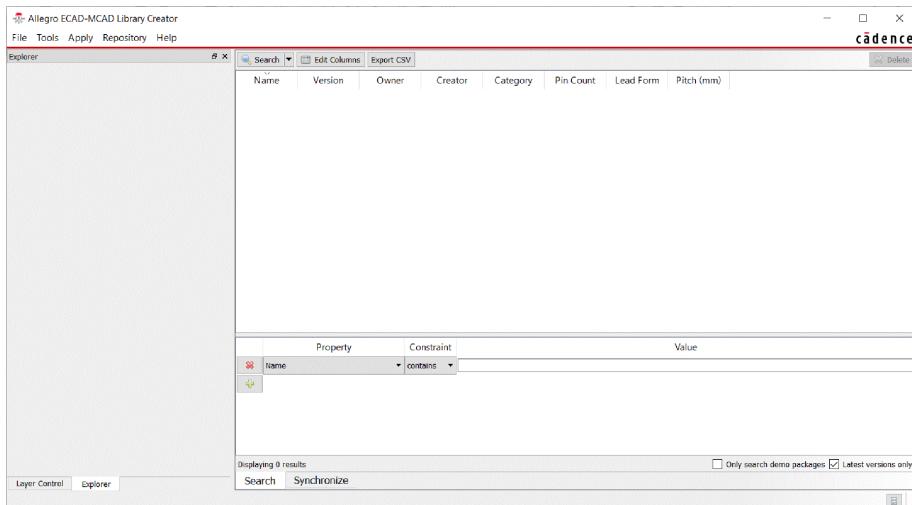
The Workspace contains a local copy of all the changes that are made prior to publishing to the repository. You can also create new configuration objects from Workspace.

Package

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

Launching Library Creator

1. Choose *Start – Cadence PCB 2023 – Allegro X Library Creator 2023*.
Allegro ECAD-MCAD Library Creator is displayed.



Adding Rules to Apply Menu

To add rules to the *Apply* menu:

1. Choose *Apply - Edit Quick Launch*.
Edit Quick Launch dialog appears.
2. Click the *Add* button.
A new rule is added to the bottom of the quick launch list.

3. Double-click the newly added rule and select the rule you want to add from the drop-down list.
 4. Click *Save*.
- The new rule is now available in the *Apply* menu.

Deleting Rules from Apply Menu

To delete a rule:

1. Choose *Apply - Edit Quick Launch*.
Edit Quick Launch dialog appears.
 2. Select a rule from the list.
 3. Click *Delete*.
 4. Click *Save*.
- The new rule is deleted from the *Apply* menu.

Creating a Footprint from Packages

What You Will Learn

In this module, you will learn how to search for and load an existing package from the repository, apply a set of supplied rules to the package to generate a footprint. You will also learn how to export the padstacks, footprint and a detailed 3D STEP model to Allegro PCB Editor.

- [Loading Package](#)
- [Applying Rule to the Package](#)
- [Exporting the Footprint to Allegro PCB Editor](#)

Loading Package

The first step in creating a footprint from a package is searching and loading a package from the repository.

To load a package:

1. Select *Pin Count* from the *Property* drop-down list and *equals* from the *Constraint* drop-down list in the *Search* tab.
2. Type *8* in *Value* and press *Enter*.

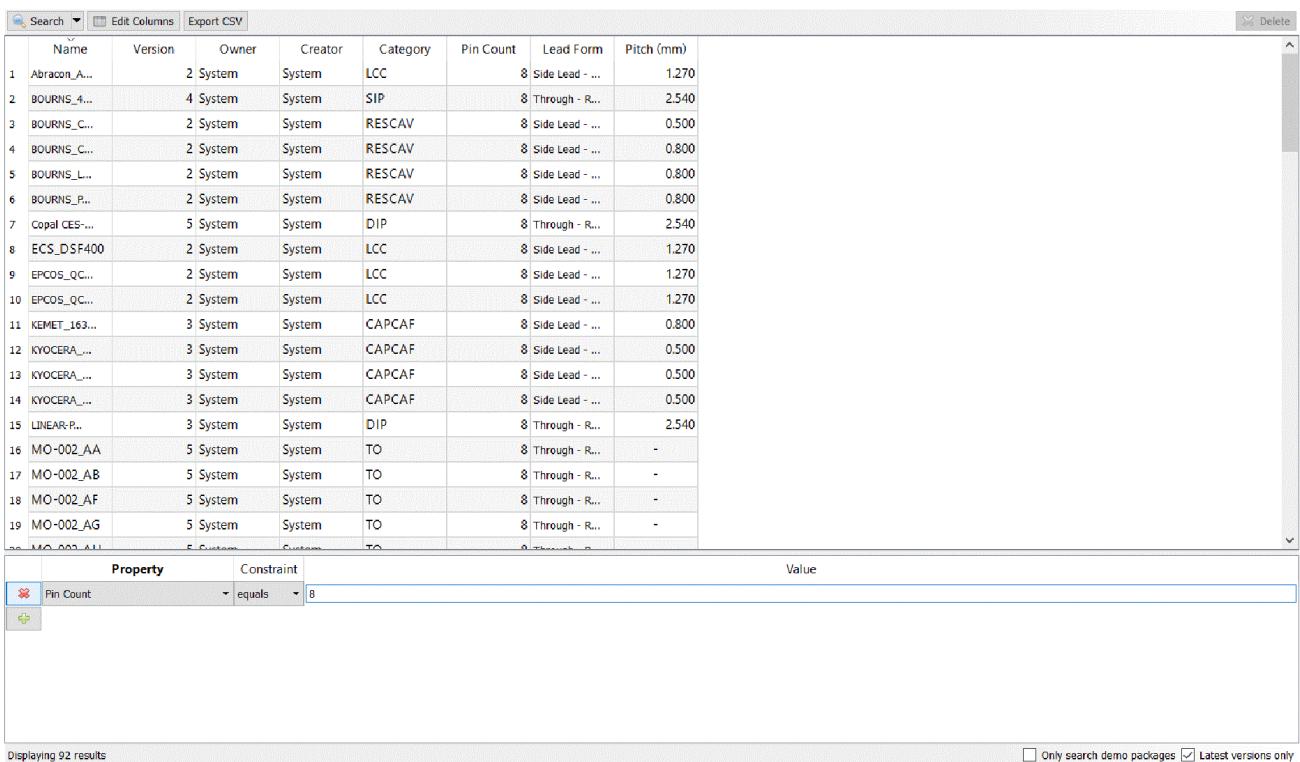
Ensure that the *Latest versions only* check box is selected.

The screenshot shows the search interface with the following details:

- Property:** Pin Count
- Constraint:** equals
- Value:** 8
- Search Options:** Only search Noida_Training packages (unchecked), Latest versions only (checked)
- Results:** Displaying 93 results
- Buttons:** Search, Synchronize

Search results are displayed.

Allegro X ECAD-MCAD Library Creator Tutorial
Creating a Footprint from Packages--What You Will Learn

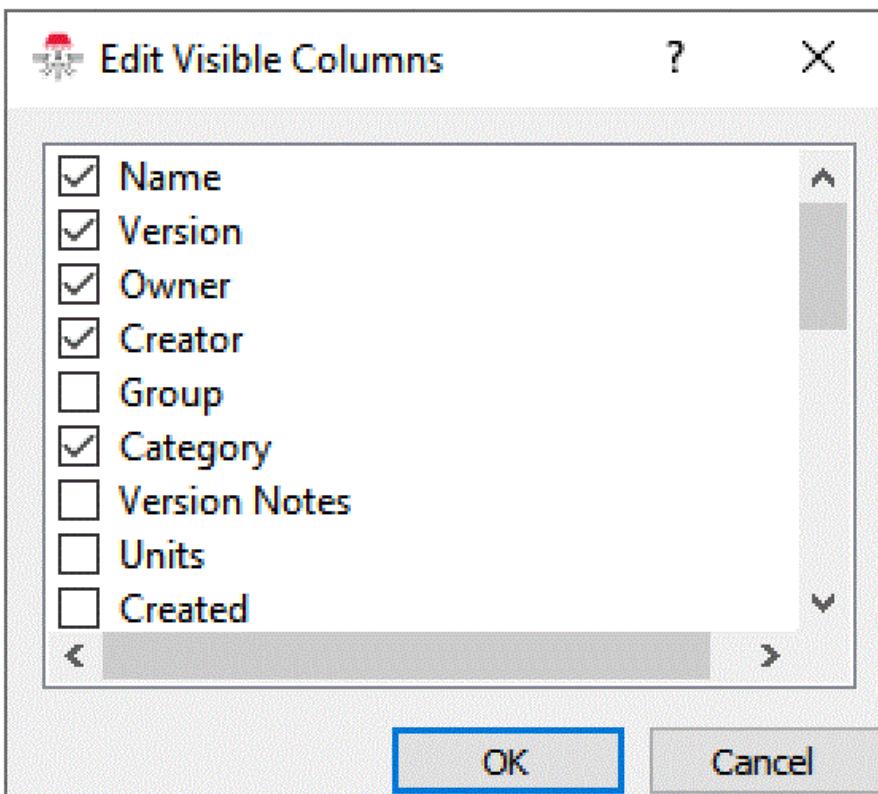


The screenshot shows the search results window in the Allegro X ECAD-MCAD Library Creator. The main area displays a table of components with columns: Name, Version, Owner, Creator, Category, Pin Count, Lead Form, and Pitch (mm). Below the table is a constraint editor with a header 'Property' and 'Constraint'. A dropdown menu under 'Property' shows 'Pin Count' selected, and the 'Constraint' dropdown shows 'equals' selected with the value '8'. At the bottom left, it says 'Displaying 92 results'. At the bottom right, there are checkboxes for 'Only search demo packages' and 'Latest versions only'.

	Name	Version	Owner	Creator	Category	Pin Count	Lead Form	Pitch (mm)
1	Abraccon_A...	2	System	System	LCC	8	Side Lead - ...	1.270
2	BOURNS_4...	4	System	System	SIP	8	Through - R...	2.540
3	BOURNS_C...	2	System	System	RESCAV	8	Side Lead - ...	0.500
4	BOURNS_C...	2	System	System	RESCAV	8	Side Lead - ...	0.800
5	BOURNS_L...	2	System	System	RESCAV	8	Side Lead - ...	0.800
6	BOURNS_P...	2	System	System	RESCAV	8	Side Lead - ...	0.800
7	Copal CES...	5	System	System	DIP	8	Through - R...	2.540
8	ECS_DSF400	2	System	System	LCC	8	Side Lead - ...	1.270
9	EPCOS_QC...	2	System	System	LCC	8	Side Lead - ...	1.270
10	EPCOS_QC...	2	System	System	LCC	8	Side Lead - ...	1.270
11	KEMET_163...	3	System	System	CAPCAF	8	Side Lead - ...	0.800
12	KYOCERA_...	3	System	System	CAPCAF	8	Side Lead - ...	0.500
13	KYOCERA_...	3	System	System	CAPCAF	8	Side Lead - ...	0.500
14	KYOCERA_...	3	System	System	CAPCAF	8	Side Lead - ...	0.500
15	LINEAR-R...	3	System	System	DIP	8	Through - R...	2.540
16	MO-002_AA	5	System	System	TO	8	Through - R...	-
17	MO-002_AB	5	System	System	TO	8	Through - R...	-
18	MO-002_AF	5	System	System	TO	8	Through - R...	-
19	MO-002_AG	5	System	System	TO	8	Through - R...	-
...	MO-002_AI	5	Custom	Custom	TO	8	Through - R...	-

Property	Constraint	Value
	Pin Count	equals 8

The columns in the search result window can be added or removed by clicking the *Edit Columns* button and then selecting the column name in the *Edit Visible Columns* dialog.

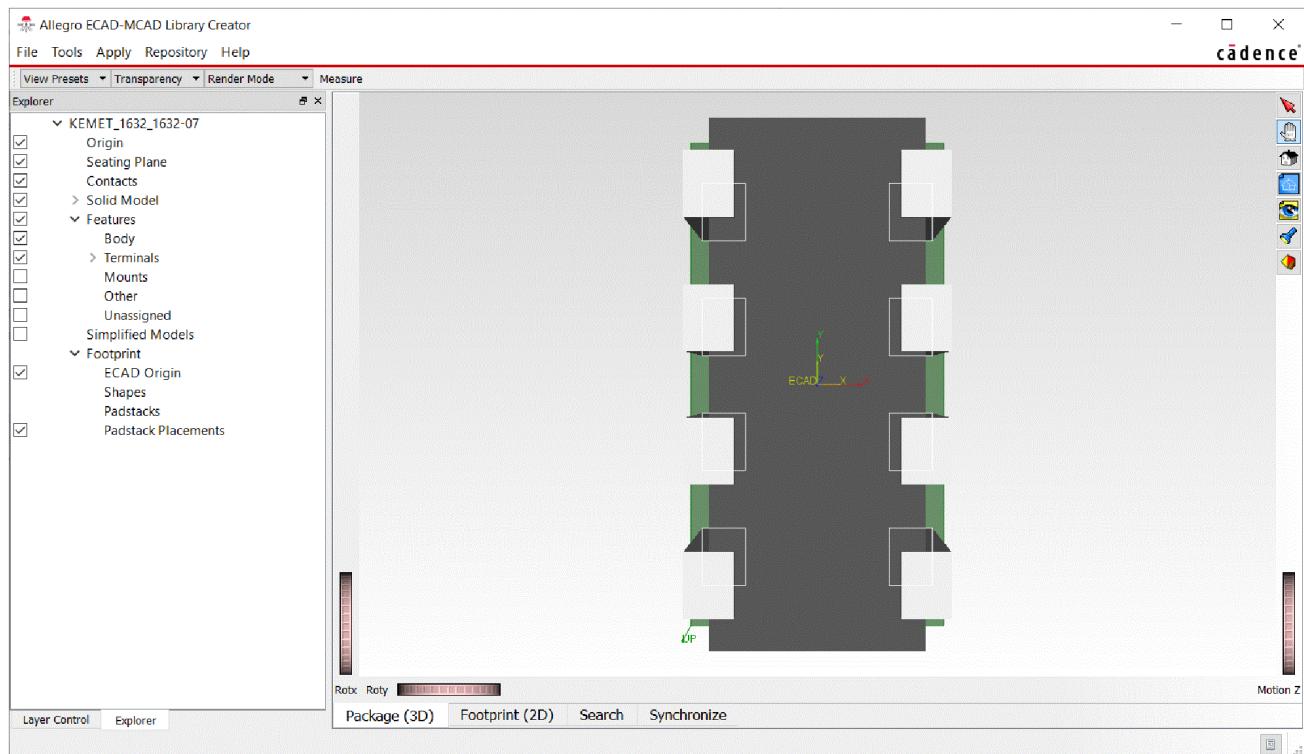


You can also sort all the columns in ascending or descending order by just clicking the column name.

3. Right-click the package *KEMET_1632_1632-07* in the *Name* column and select *Load*. Alternatively, you can also load a package by double-clicking the package name. The package *KEMET_1632_1632-07* loaded and displayed on the grid as well as *Explorer*.

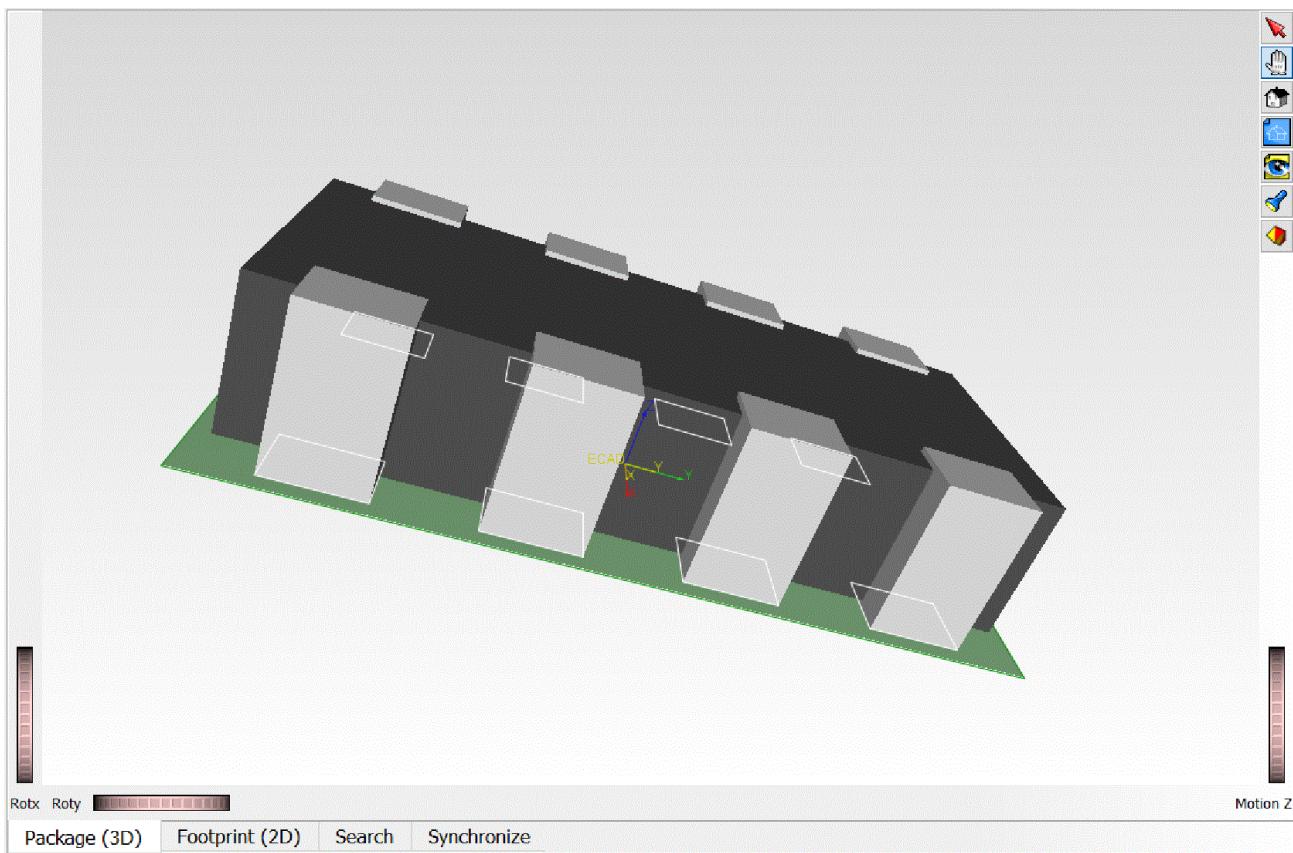
Allegro X ECAD-MCAD Library Creator Tutorial

Creating a Footprint from Packages--What You Will Learn



You can also see all the sides of the loaded package from the *Package (3D)* tab.

4. Rotate the package using the left mouse button to see all sides of it. You can also use the mouse wheel to zoom in or zoom out the package.



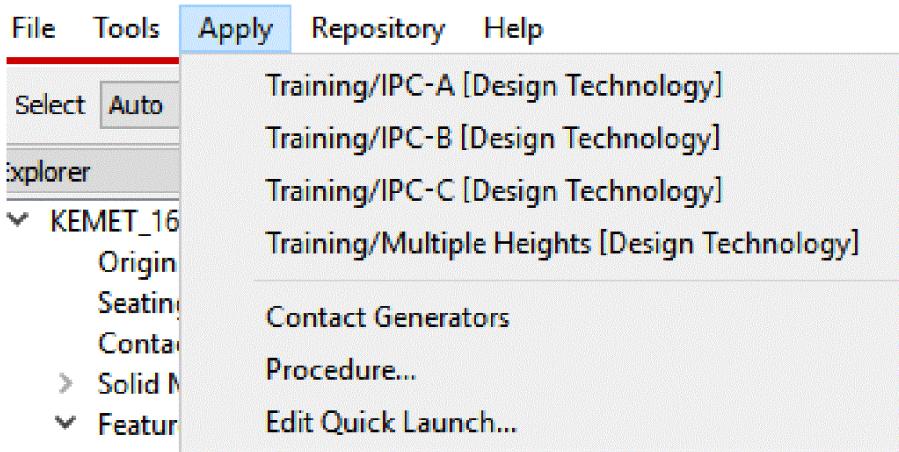
Applying Rule to the Package

You can apply rules to packages from the *Apply* menu. Depending on the context, different types of rule objects can be applied. In this section, you will apply *Training/IPC-A [Design Technology]* rule to the package.

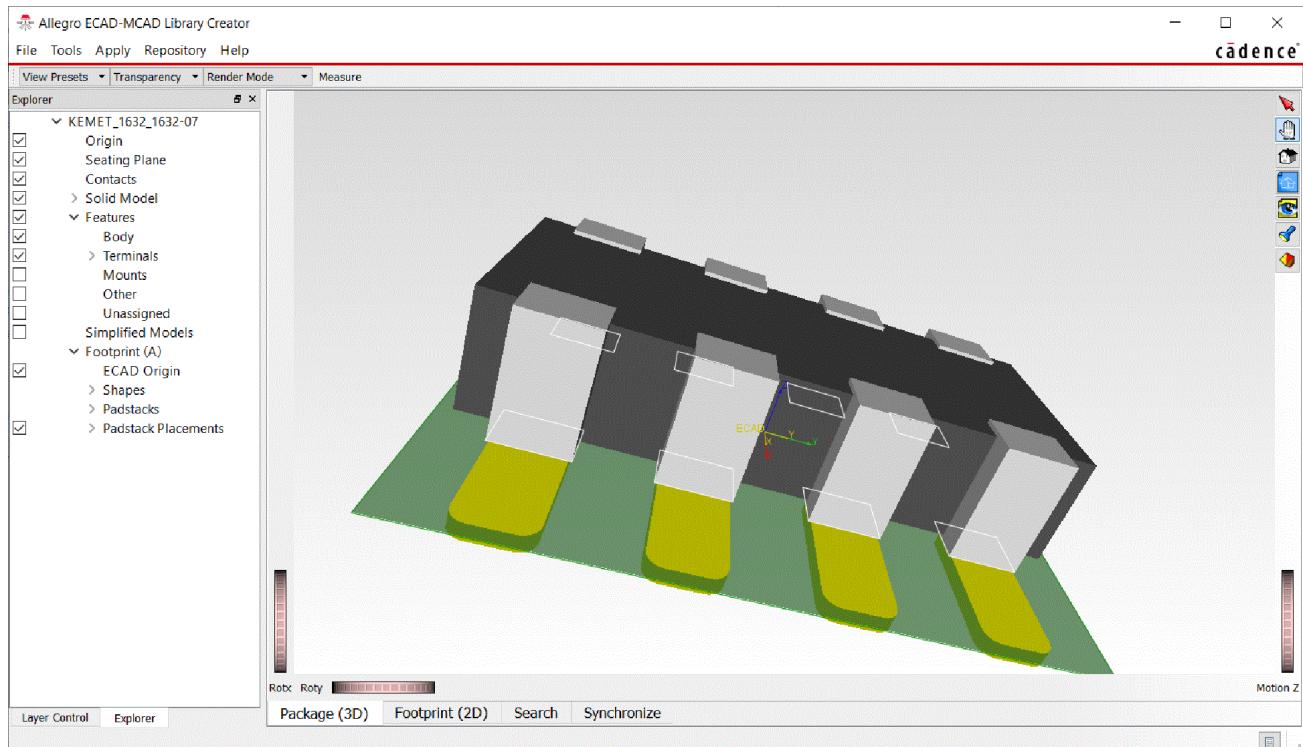
For more information about adding a specific rule to the *Quick Launch* list, refer [Adding Rules to Apply Menu](#).

To apply rules to a package:

1. Choose *Apply - Training/IPC-A [Design Technology]*.



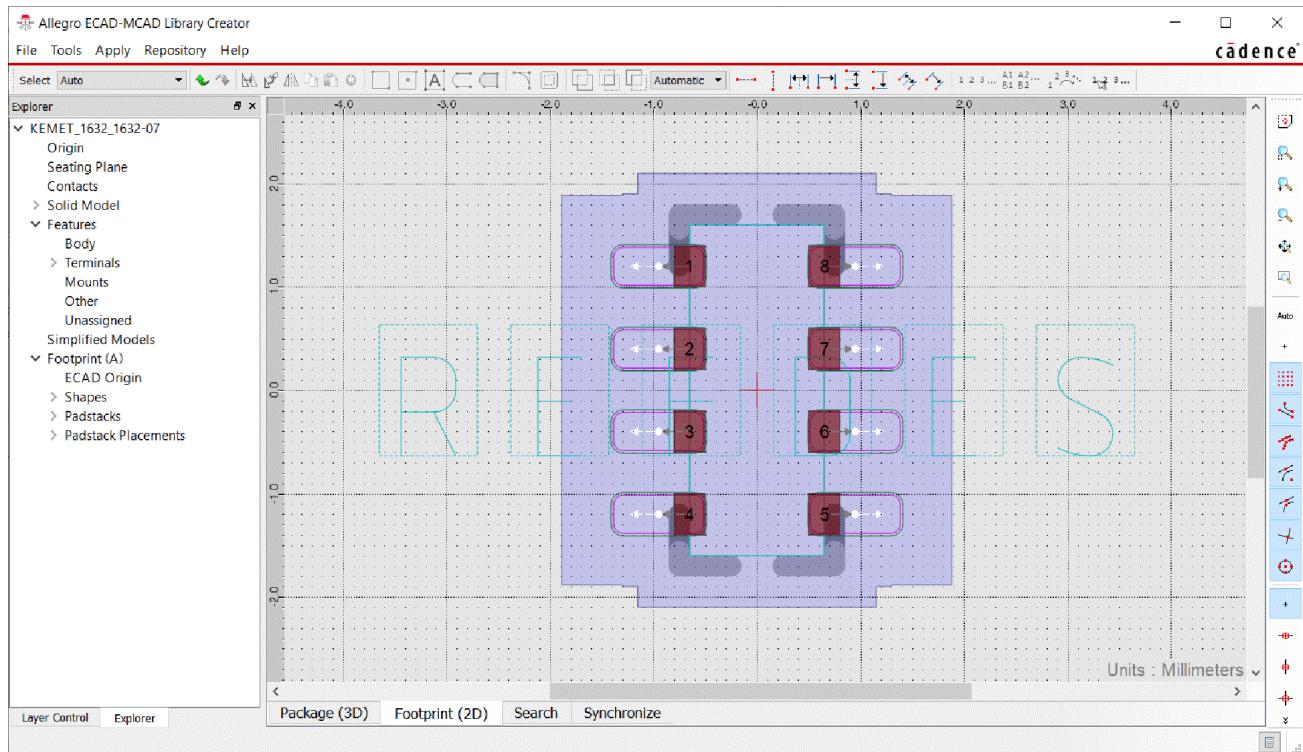
A footprint is created that matches the rules mentioned in the *Training/IPC-A [Design Technology]*.



2. Click the *Footprint (2D)* tab to view the footprint in 2D view.

Allegro X ECAD-MCAD Library Creator Tutorial

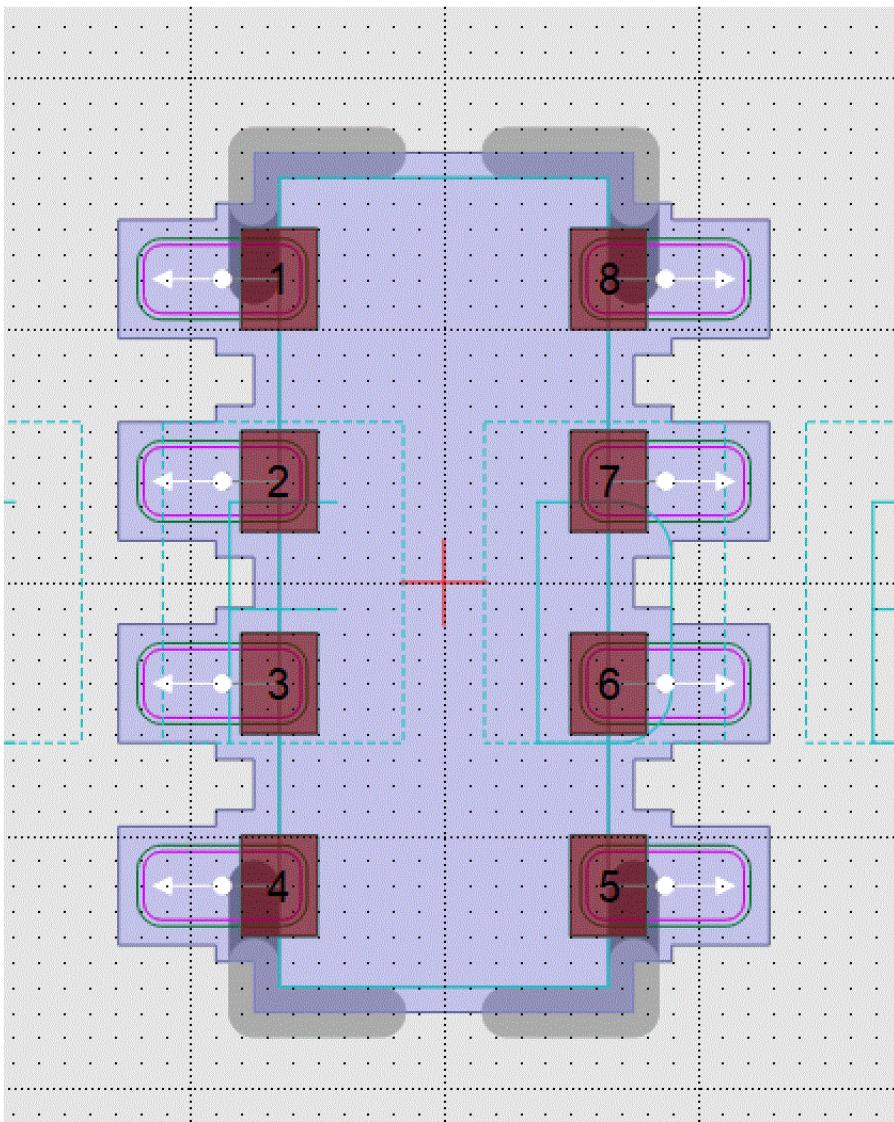
Creating a Footprint from Packages--Applying Rule to the Package



You can also apply different rules to the package.

3. Choose *Apply - Training/IPC-C [Design Technology]*.

Notice that the padstacks get smaller as shown in the following figure.



When in the *Footprint (2D)* view, you can change colors, transparencies, or filled/unfilled areas using the Layer Control tab of the *Explorer* panel.

Layer Control

Labels	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Padstack Placements	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Connect Points	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Contact Orientations	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Contact Areas	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Assembly_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Place_Bound_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Silkscreen_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Pastemask_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Soldermask_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Etch_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Dfa_Bound_Top	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Package Area	<input checked="" type="checkbox"/>	<input type="checkbox"/>	
Scratch - Silkscreen	<input checked="" type="checkbox"/>	<input type="checkbox"/>	

4. Choose *Apply - Training/IPC-A [Design Technology]* to revert to the IPC-A rule.

Exporting the Footprint to Allegro PCB Editor

Allegro Library Creator allows you to import and export padstacks, STEP models, footprints, to and from Allegro PCB Editor. This supports the population and synchronization of the PCB library.

 Before initiating the import or export, ensure that all the settings in the Allegro Export dialog are configured your library in one location. For ore information about setting the dialog, refer [Setup the Tutorial Database](#).

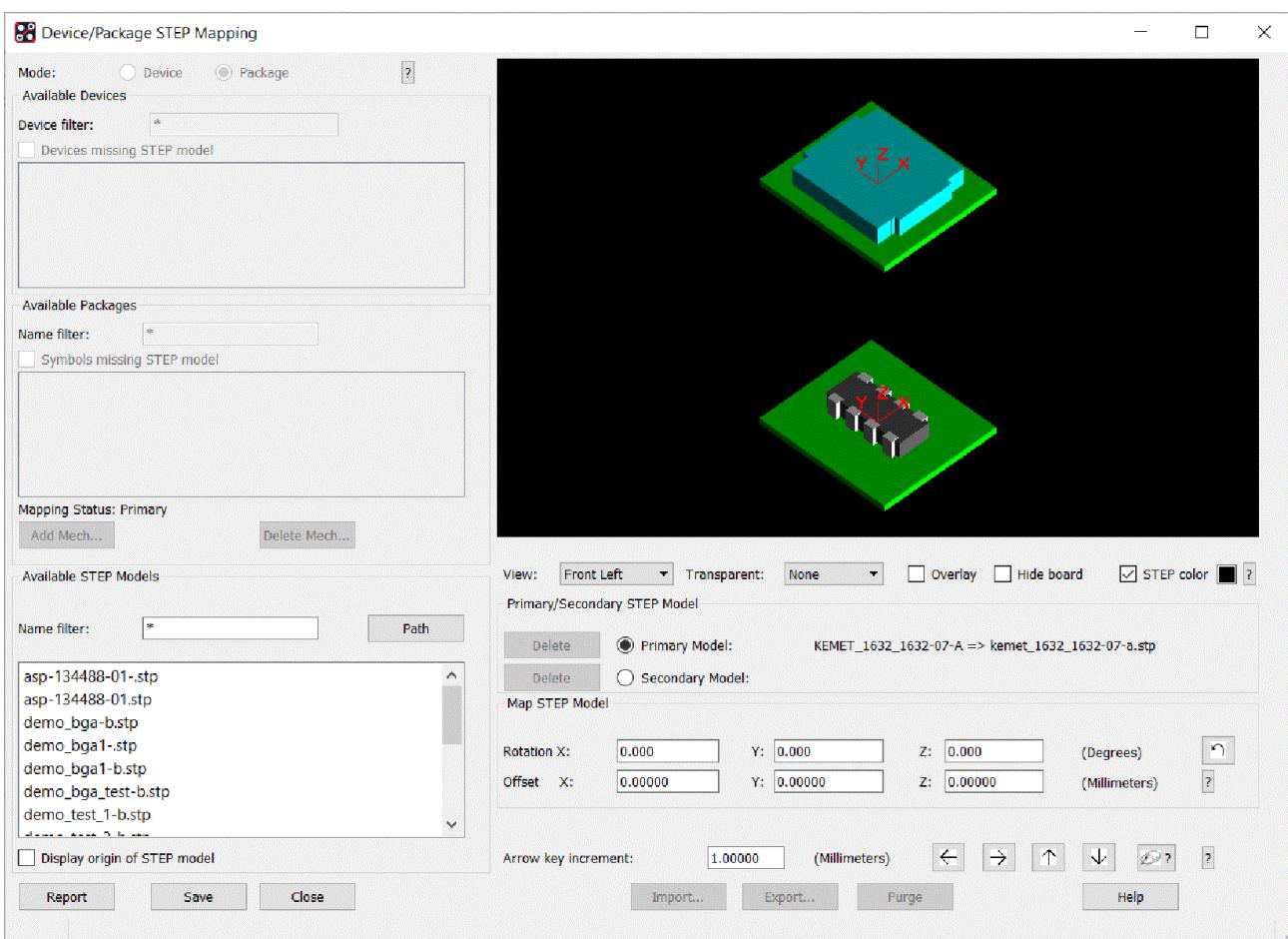
To export the footprint to Allegro PCB Editor:

1. Choose *File - Export - Allegro* to export the footprint and STEP model to Allegro PCB Editor.
 The *Waiting for task to complete* dialog is displayed.

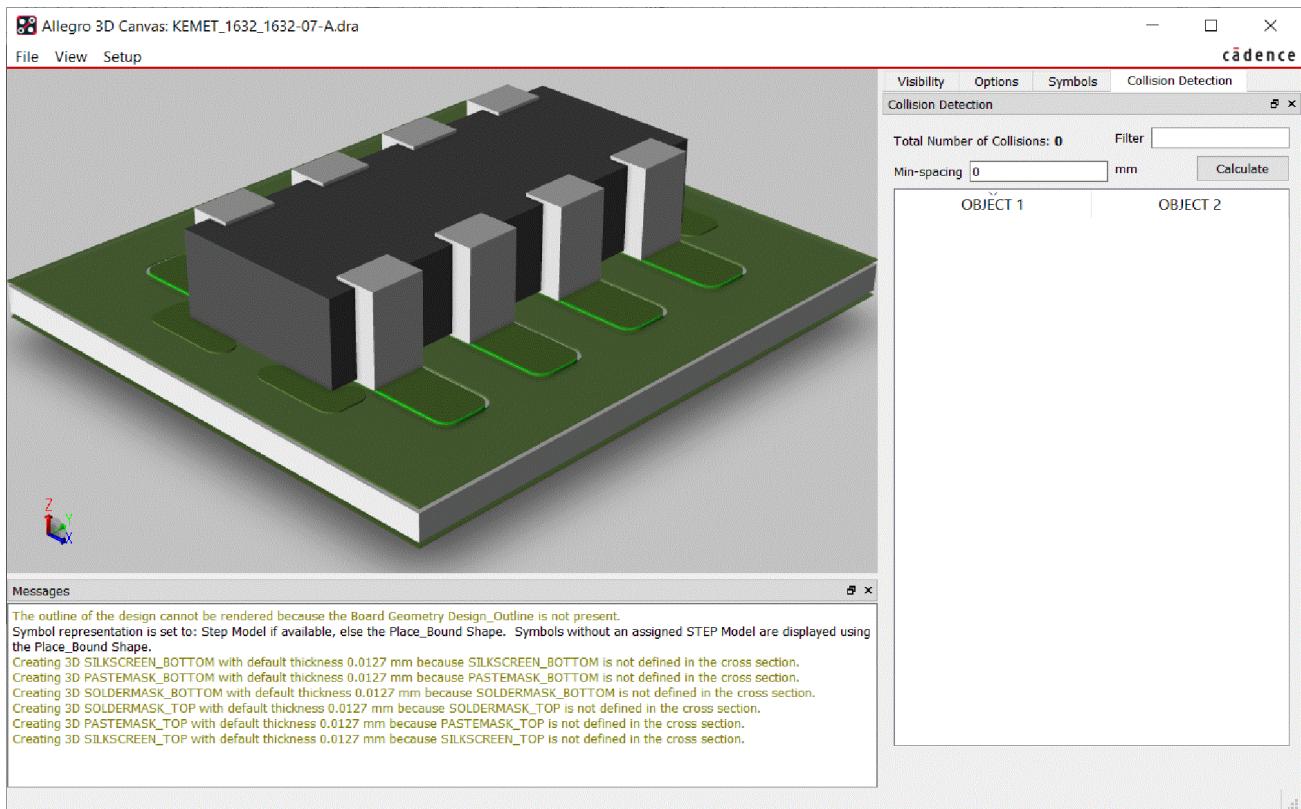


A *KEMET_1632_1632-07-A.dra* file is created at the *Package Symbols* location selected in the Allegro Settings dialog.

2. Open the *KEMET_1632_1632-07-A.dra* file in Allegro PCB Editor to see that everything transferred over correctly.
3. Choose *Setup - STEP Package Mapping*.
 You can see that both the PLACEBOUND shape and STEP model have been added to the footprint we just created.



4. Close the *Device/Package STEP Mapping* dialog.
5. Click on the *3D* icon to see this footprint and STEP model in the PCB Editor 3D canvas.



Summary

In this section, you learned how to:

- Load a package
- Apply rules to the package
- Export the footprint to Allegro PCB Editor

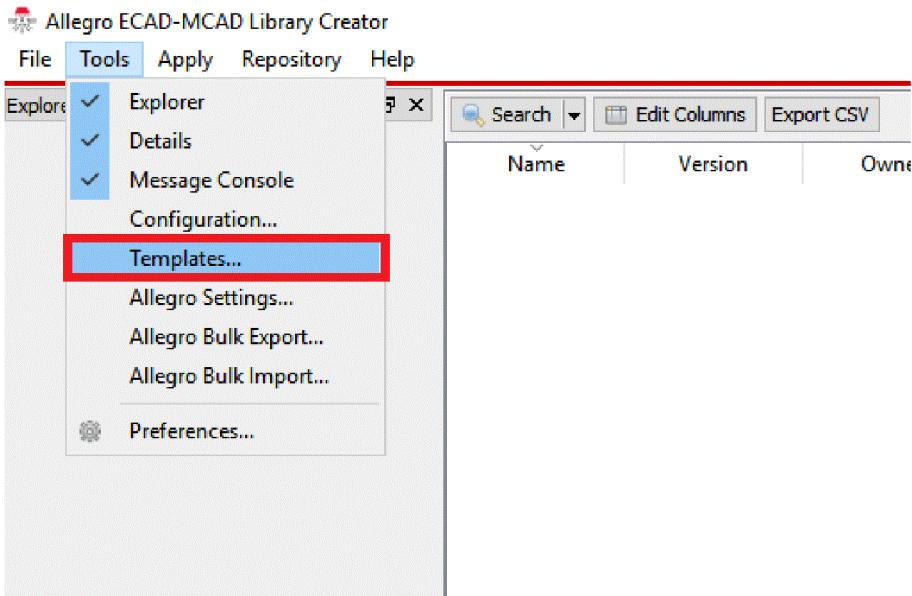
Creating a Footprint from Templates

What You Will Learn

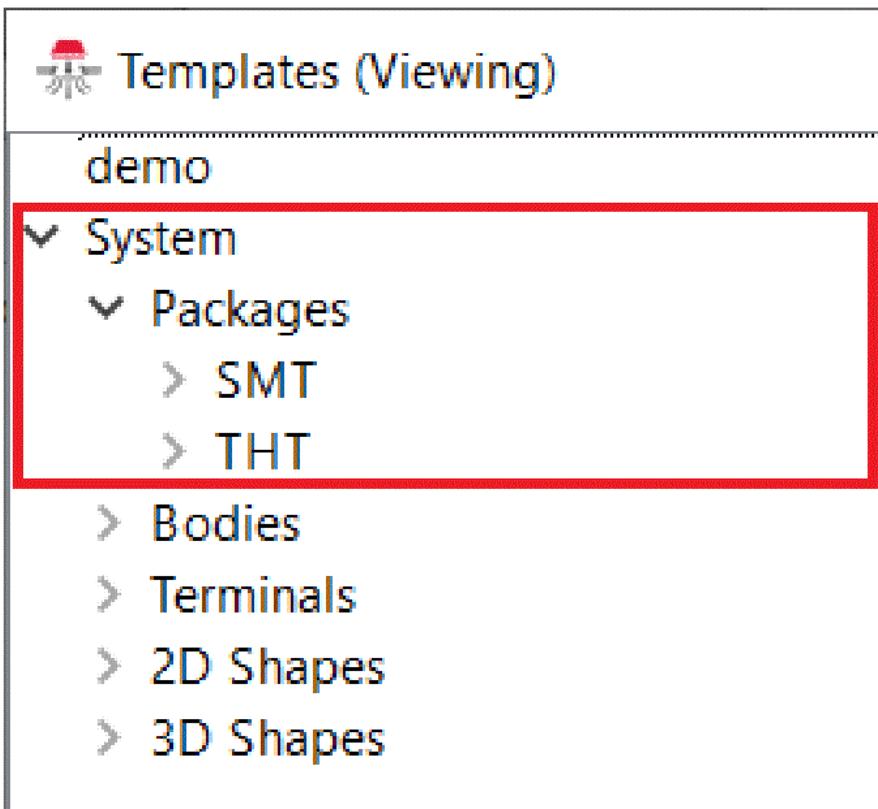
In this module, you will learn how to use an existing template to create a new instance of the template, apply a set of supplied rules to the package to generate a footprint, save the package to the repository, and then export the footprint and a detailed 3D STEP model to Allegro PCB Editor.

Accessing the Package Templates

1. Choose *Tools - Templates* to access the package templates.



2. Expand the System section and then expand the Packages section in the *Templates (Viewing)* dialog.



Templates are organized by categories.

3. Expand the SMT section and then expand the Area Array.

The templates tree also displays a thumbnail view of the available templates. For this tutorial, we will use the BGA (Ball Grid Array) template.

4. Click *BGA*.

Template details are displayed. Next we will change the necessary values in the parametric table.

Creating a New Instance of Template

1. Double-click the *Value* column of the *name* field in the parametric table of the *Templates (Viewing)* dialog.
The field becomes editable.
2. Enter `demo_bga`.
3. Enter `1.61/0.19` in the *Value* column of the *package_height* field.

⚠️ Tolerances can be added simply by typing the values and then adding a forward slash in between the value.

Change all the parametric values as displayed in the following figure.

	Value
name	demo_bga
lead_form	Ball
units	Millimeters
n_x	14
n_y	14
pitch_x	1
pitch_y	1
package_height	1.61 ± 0.19
standoff	0.4 ± 0.1
body_x	15 ± 0.2
body_y	15 ± 0.2
terminal_diameter	0.5 ± 0.1

[Previous](#)

[Next](#)

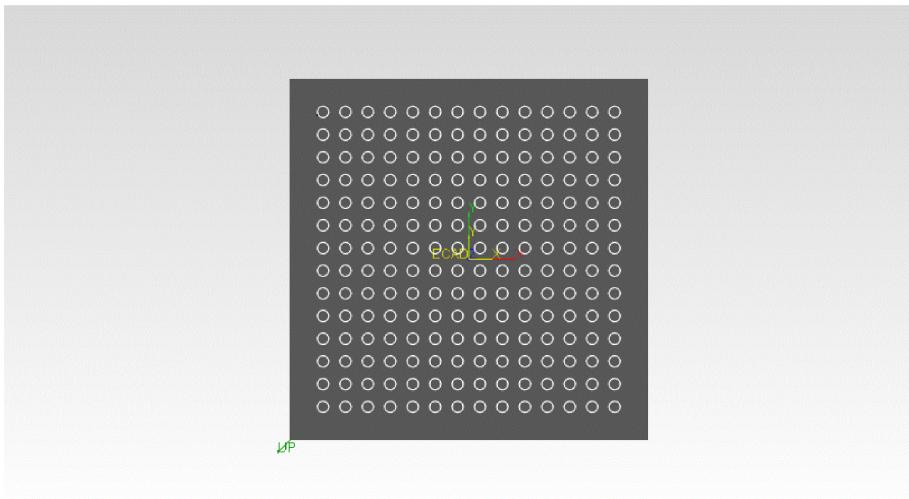
[New Instance](#)

[View Family](#)



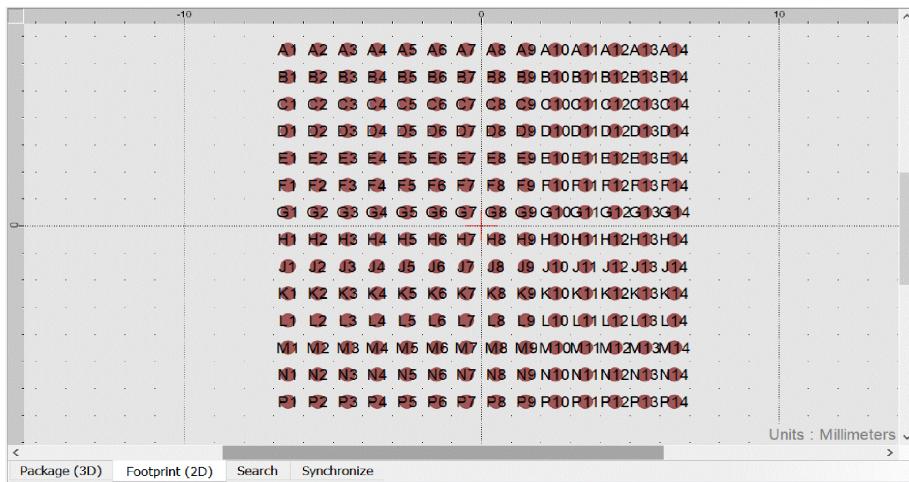
4. Choose New Instance and close the *Templates (Viewing)* dialog.

The BGA package is created and displayed in the Package (3D) view.



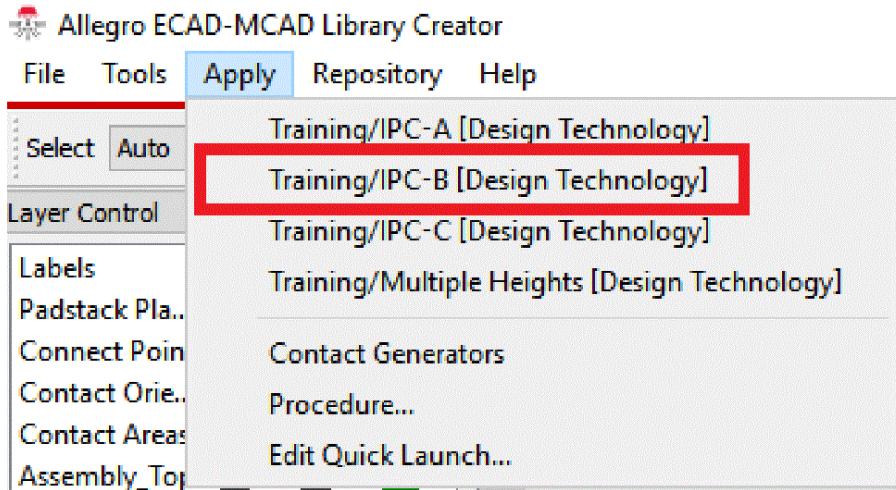
5. Click Footprint (2D) to view the contact areas for each pin.

Notice how the BGA pins are already numbered as per the PDF spec sheet. Note that the PDF spec sheet shows the Bottom View of the pin array, while the Library Creator (2D) view is from the top side – as viewed on the PCB board.

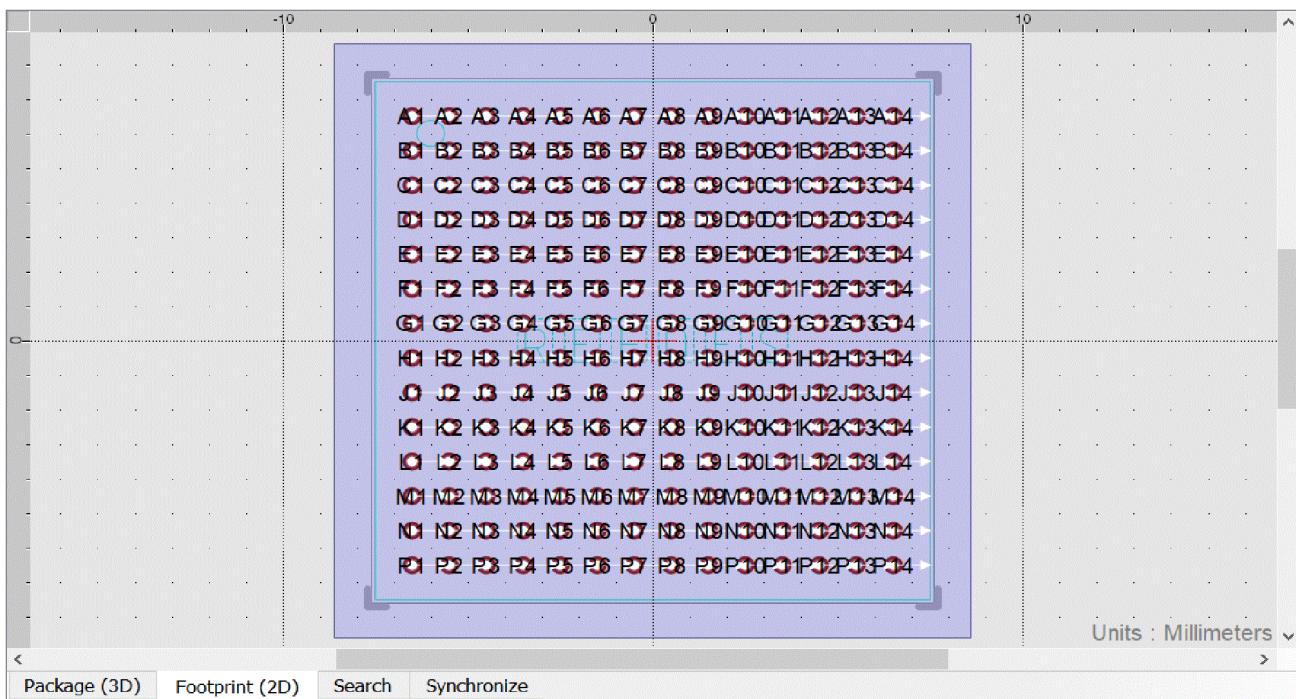


Applying Rules to Template

1. Choose Apply - Training/IPC-B (Design Technology) to create the footprint.



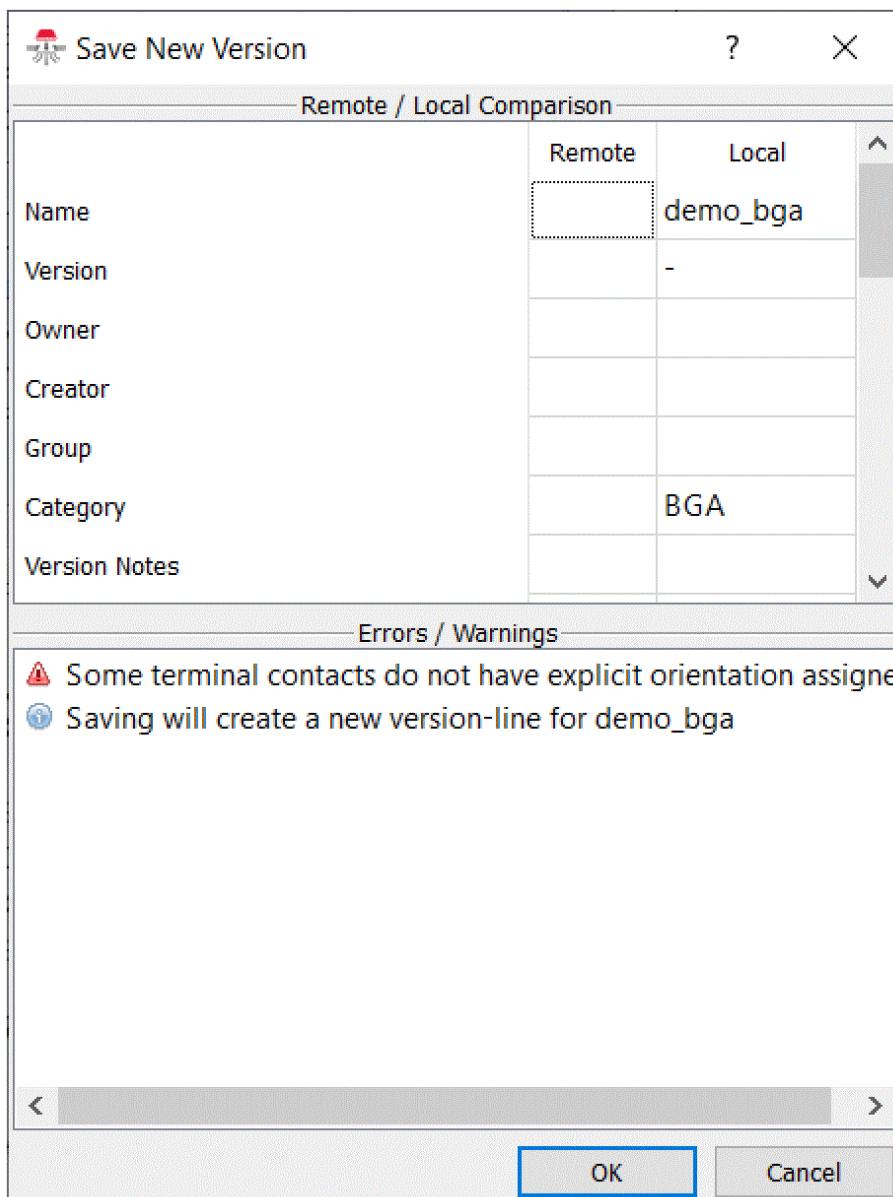
The footprint matching the IPC-B rule is created.



Saving the Footprint

Exporting the footprint at this stage to Allegro PCB Editor without saving it will display an error message. The reason for this error is that a relationship between the package and the footprint cannot be created because the package is not saved. It is important that you save a new version of the footprint before exporting it to PCB Editor.

1. Choose *File - Export - Allegro* to export the footprint and STEP model to Allegro PCB Editor.
An error message is displayed.
2. Click OK to exit the error message.
3. Choose *File - Upload - Save New Version* to save the package to the repository.
The *Save New Version* dialog appears. Here, additional information can be added to if required as your company design practices.



Ignore the errors and warnings displayed in the *Errors/Warnings* section.

4. Click OK.

Version Notes dialog box is displayed.

5. Enter `demo_bga_v1` and click OK.

A success message is displayed in the *Message Console* window.

Exporting Footprint to Allegro PCB Editor

1. Choose *File - Export - Allegro*.

After export is completed, a file named *demo_bga1-B.dra* is created at the *Package Symbols* location selected in the *Allegro Settings* dialog.

2. To verify the footprint, open the *demo_bga1-B.dra* file in Allegro PCB Editor to see that everything transferred correctly.

Creating a Footprint from Package Templates

What You Will Learn

In this chapter, you will learn how to load a package from the repository, create a new instance by changing some of the values in the underlying template, upload the new instance to the repository, apply a set of rules to the package to generate a footprint and then export both the footprint and a detailed 3D STEP model to Allegro as was shown in the first lab.

Loading a Package

The first step in creating a footprint from a package is searching and loading a package from the repository.

To load a package:

1. Select *Pin Count* from the *Property* drop-down list and *equals* from the *Constraint* drop-down list in the *Search* tab.
2. Type *8* in *Value* and press *Enter*.

Ensure that the *Latest versions only* check box is selected.

The screenshot shows a search interface with the following details:

Property	Constraint	Value
Pin Count	equals	8

Below the search bar, there are buttons for "Displaying 93 results", "Search", and "Synchronize". At the bottom right, there are checkboxes for "Only search Noida_Training packages" and "Latest versions only", with the latter being checked.

Search results are displayed.

3. Click the *Name* column to sort the package names alphabetically in ascending order.

Allegro X ECAD-MCAD Library Creator Tutorial
Creating a Footprint from Package Templates--What You Will Learn

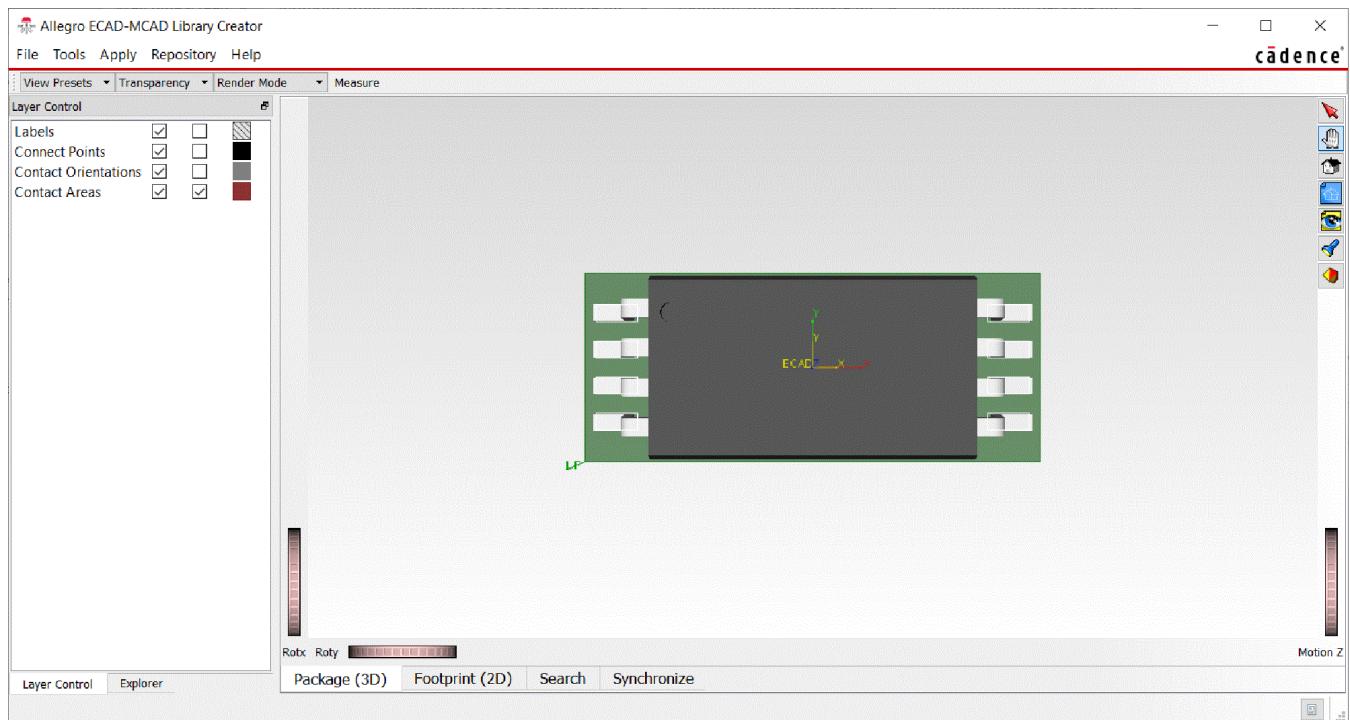
The screenshot shows the Allegro X library manager interface. At the top, there are buttons for 'Search', 'Edit Columns', and 'Export CSV'. Below the search bar, a table displays package information with columns: Name, Version, Owner, Creator, Category, Last Edit, Pin Count, Lead Form, and Pitch (mm). Nine packages are listed, with rows 1 through 9 highlighted by a red box. Row 10 is the current selection, indicated by a blue background. A constraint builder window is open at the bottom, showing a dropdown for 'Property' set to 'Pin Count', an operator 'equals', and a value '8'. There are also '+' and '-' buttons for modifying the constraint. At the bottom of the screen, there are status messages: 'Displaying 92 results', 'Only search Cadence packages' (unchecked), and 'Latest versions only' (checked).

4. Next, press *M* to automatically scroll down to the package names starting with M.

This screenshot shows the Allegro X library manager after applying a filter. The search results grid now only displays packages with 8 pins. The first three packages listed are EPCS_QCC8B, EPCS_QCC8C, and MO-002_AA. The 'MO-002_AA' row is currently selected, highlighted with a blue background. The constraint builder window at the bottom remains the same as in the previous screenshot, with 'Pin Count' set to '8'. The status bar at the bottom indicates 'Displaying 92 results' and includes the same filter checkboxes as the previous screenshot.

5. Find and load the package *MO-150_AA*.

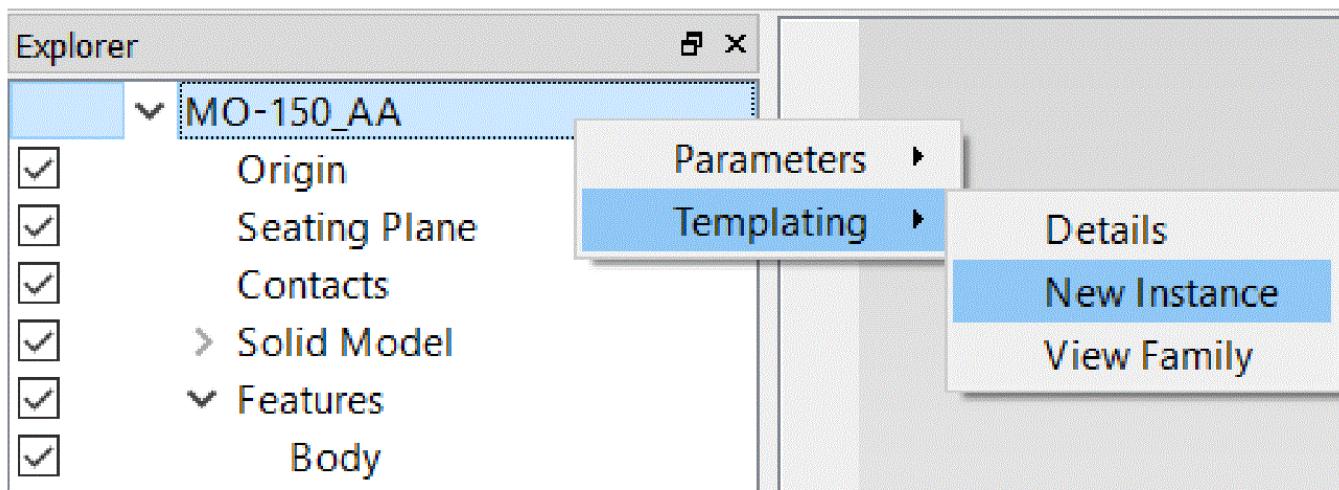
The package *MO-150_AA* loaded and displayed on the grid as well as *Explorer*.



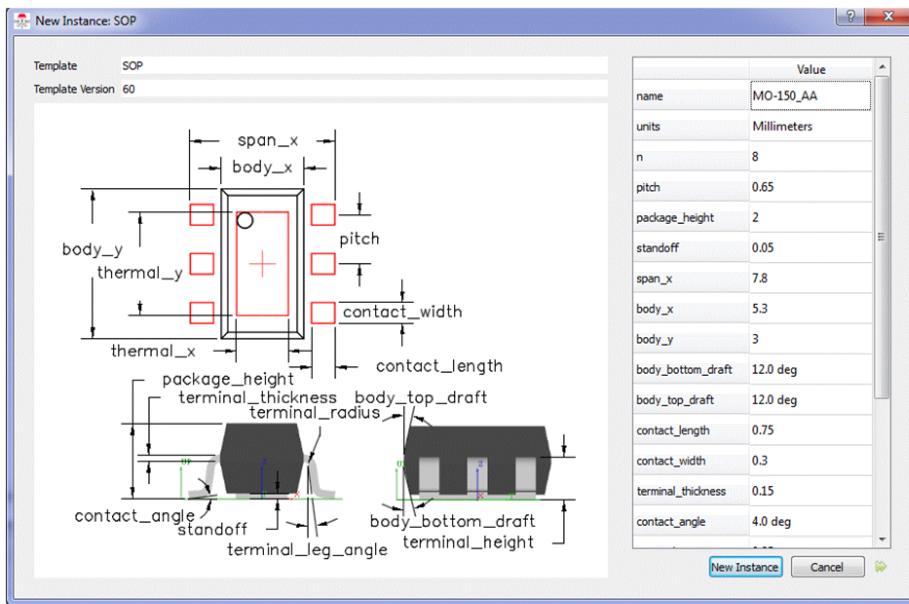
Next we will create a new instance of the package and change the necessary values in the parametric table.

Creating a New Instance of Package

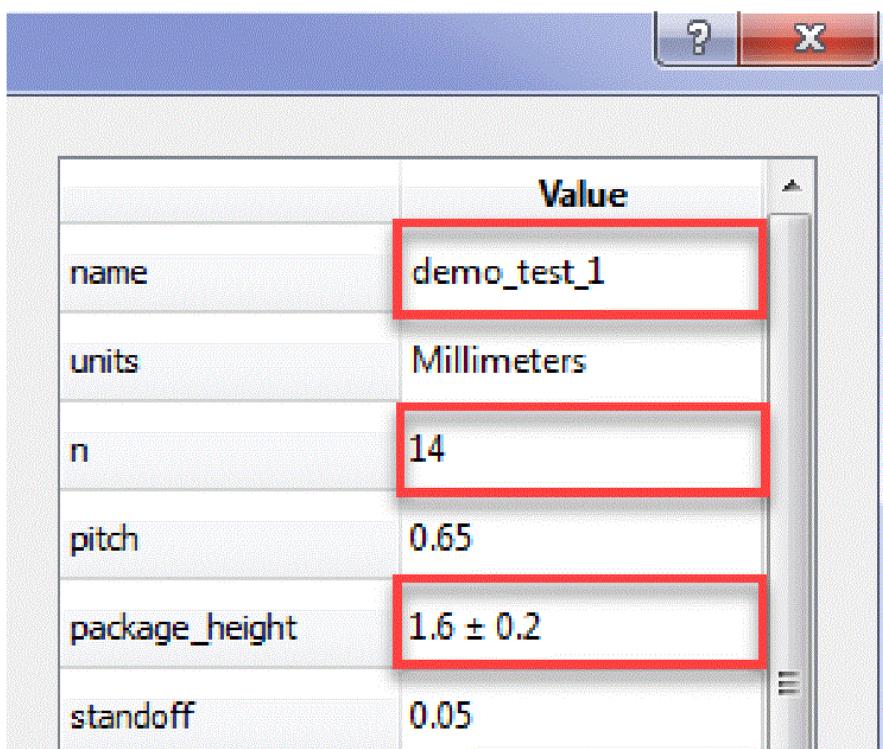
1. Right-click *MO-150_AA* in *Explorer* and choose *Templating - New Instance*.

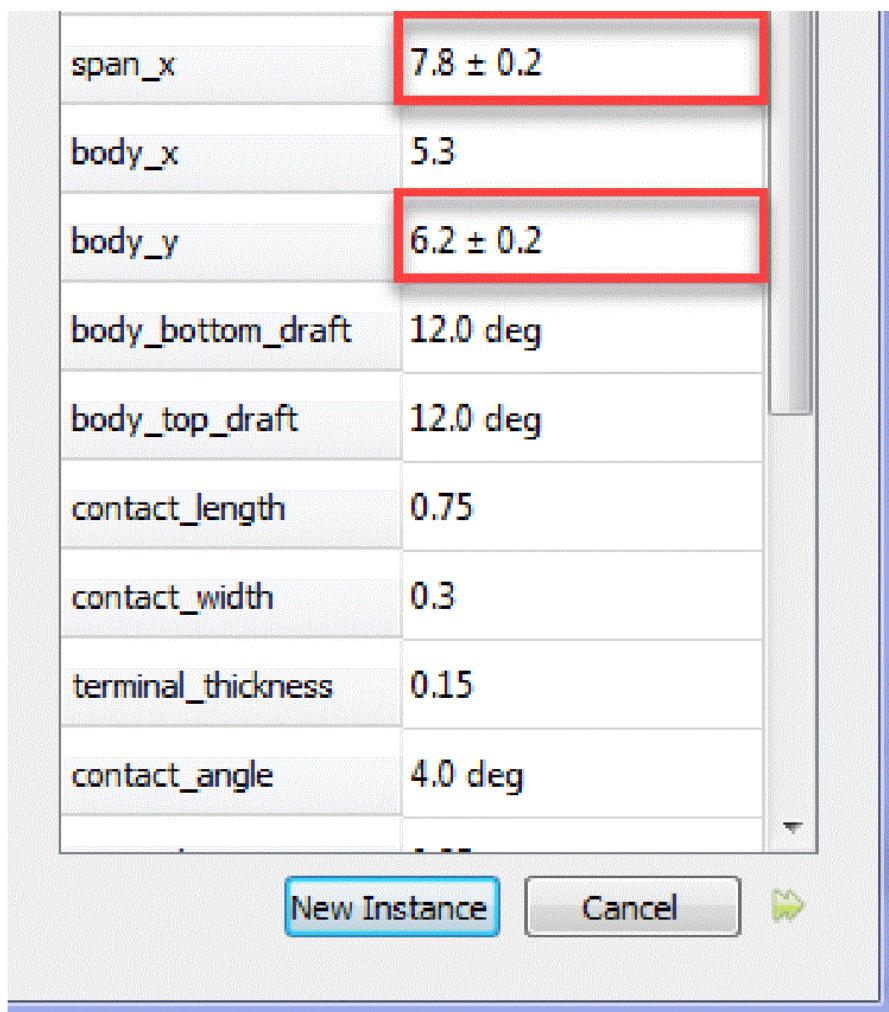


New Instance dialog appears.



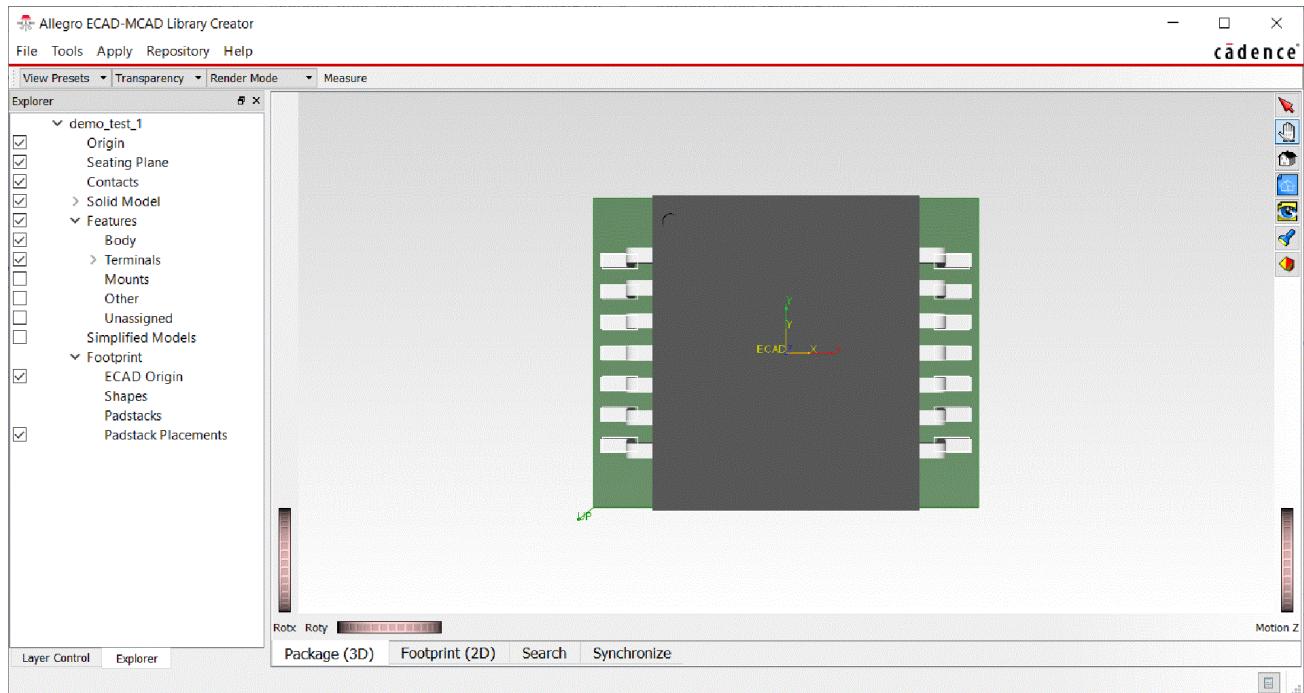
2. Double-click the *Value* column of the *name* field in the parametric table and enter `demo_test_1`.
3. Change the value of *n* field to `14` to create a similar 14 pin package.
4. Next, double-click the *package_height* field and enter `1.6 ± 0.2`.
5. Similarly change the values of *span_x* and *body_y* to `7.8 ± 0.2` and `6.2 ± 0.2` respectively.





6. Click *New Instance*.

Notice that the *Package (3D)* image changes from an 8-pin part to 14 pin part.



Saving the Footprint

It is important that you save a new version of the footprint before exporting it to PCB Editor.

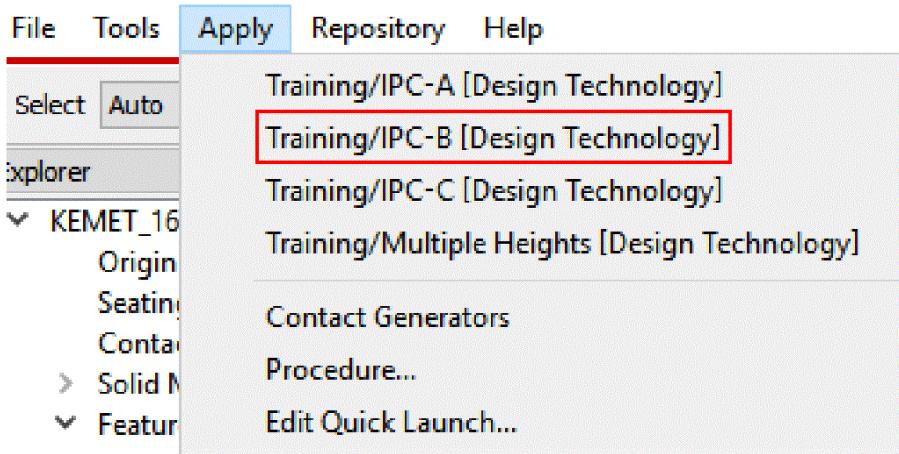
For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

Applying Rules to Package

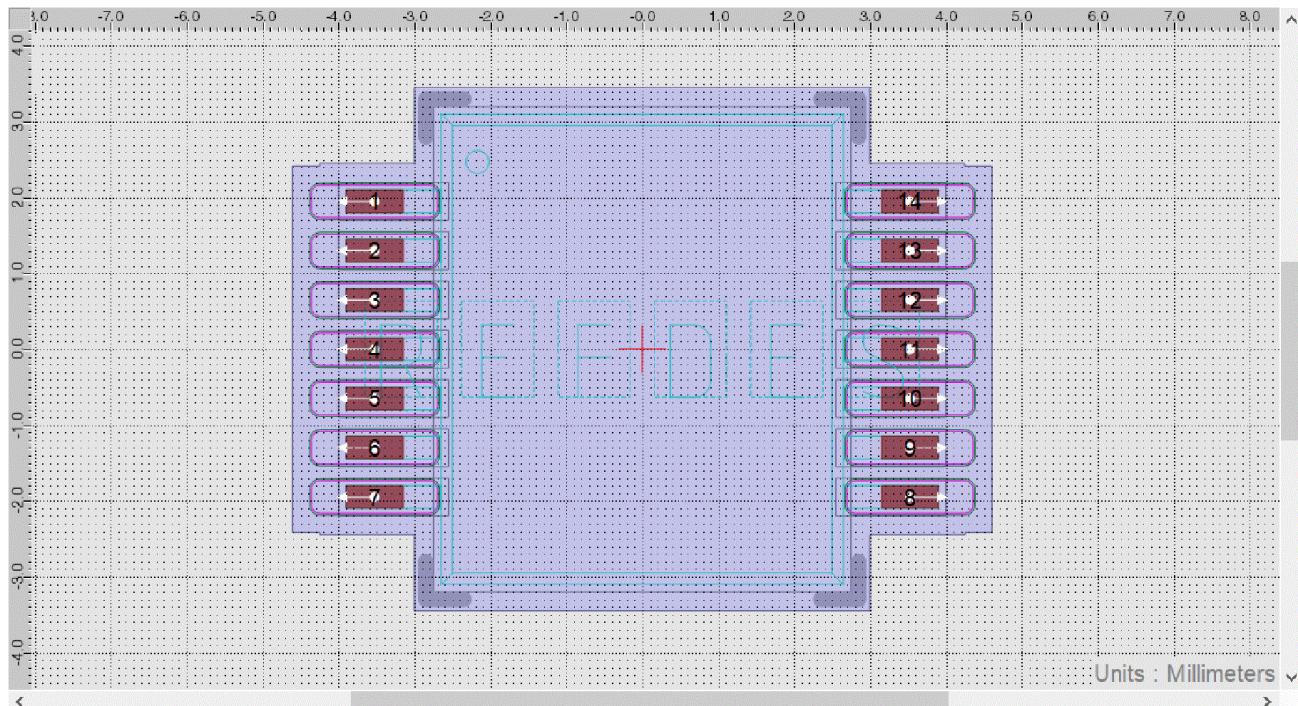
You can apply rules to packages from the *Quick Launch* list of the *Apply* menu. Depending on the context, different types of rule objects can be applied.

For more information about adding a specific rule to the *Quick Launch* list, refer [Adding Rules to Apply Menu](#).

1. Choose *Apply - Training/IPC-B [Design Technology]*.



Footprints matching the rule are created.



Exporting Footprint to Allegro PCB Editor

For more information about exporting the footprint to Allegro PCB Editor, follow the steps mentioned in the [Exporting the Footprint to Allegro PCB Editor](#) section.

Summary

In this module, you learned to create a 14 pin package from an underlying 8 pin package by changing its values and uploading the package to repository.

Creating Custom Footprints

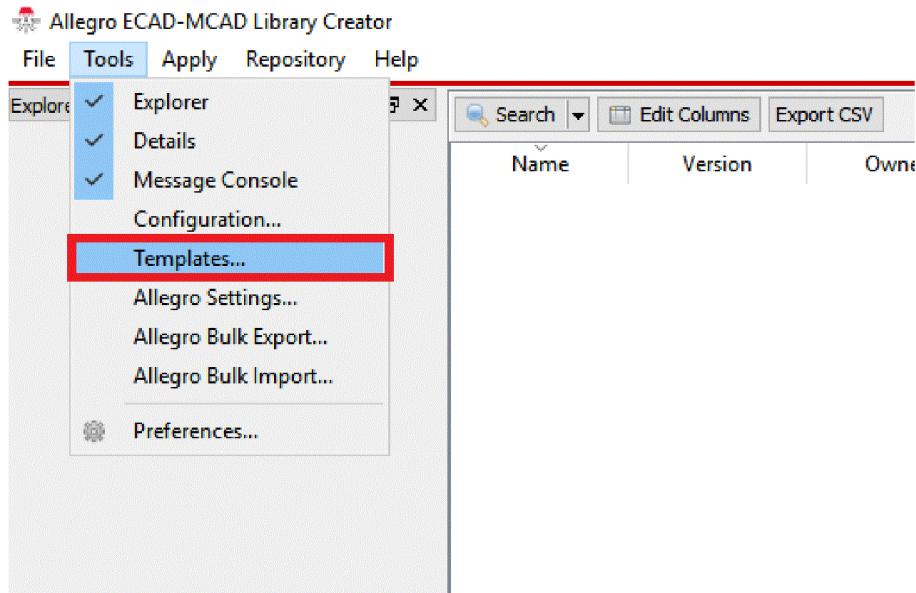
What You Will Learn

In this module, you will learn to create a non-standard footprint with a custom thermal pad underneath it using a provided template as a starting point. You will also learn to use simple drafting techniques to speed up the creation of this custom footprint.

Loading the Template

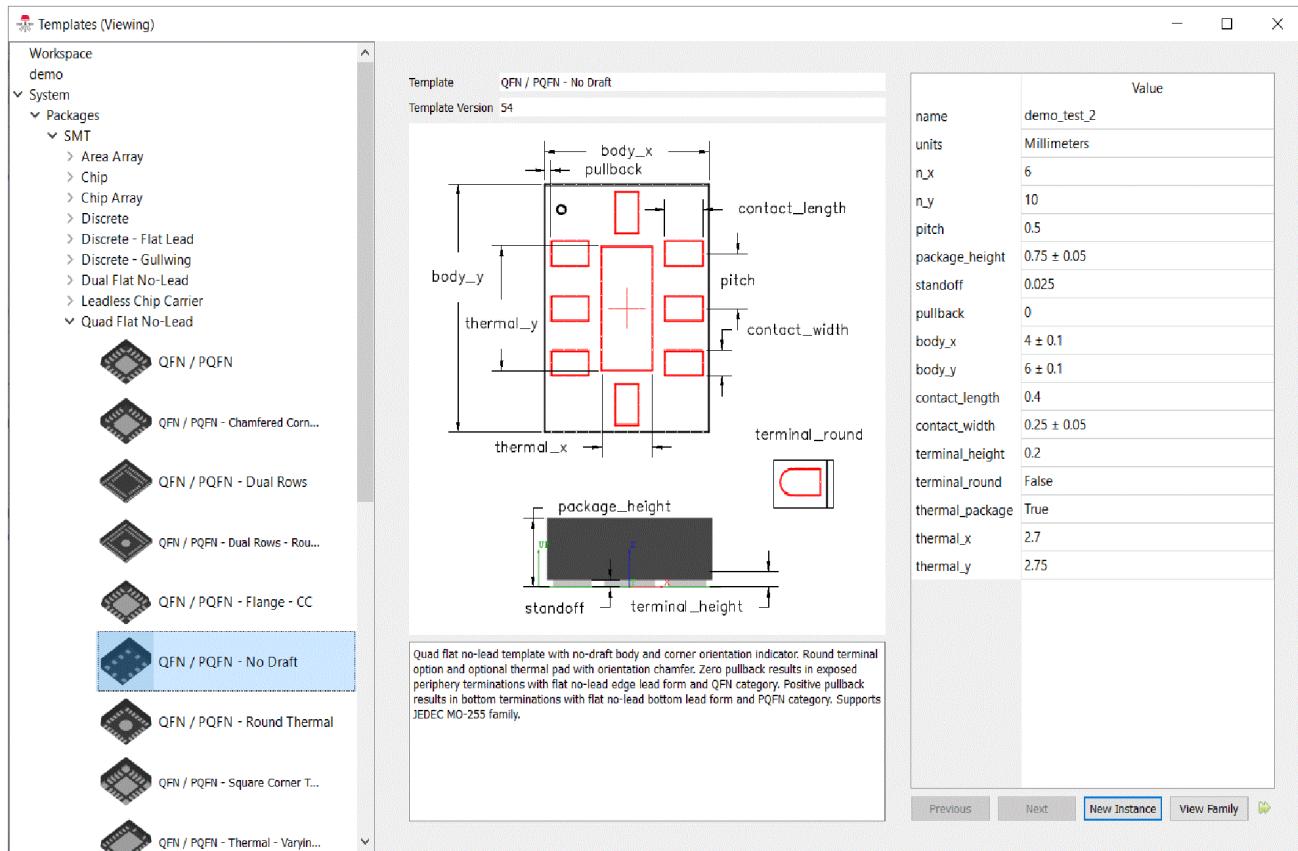
1. Choose *Tools - Templates*.

Templates (Viewing) dialog is displayed.



2. Expand the System - Packages - SMT - Quad Flat No-Lead in the *Templates (Viewing)* dialog.
3. Click *QFN / PQFN – No Draft*.

The selected template and its parametric values are displayed.



Creating a New Instance of Template

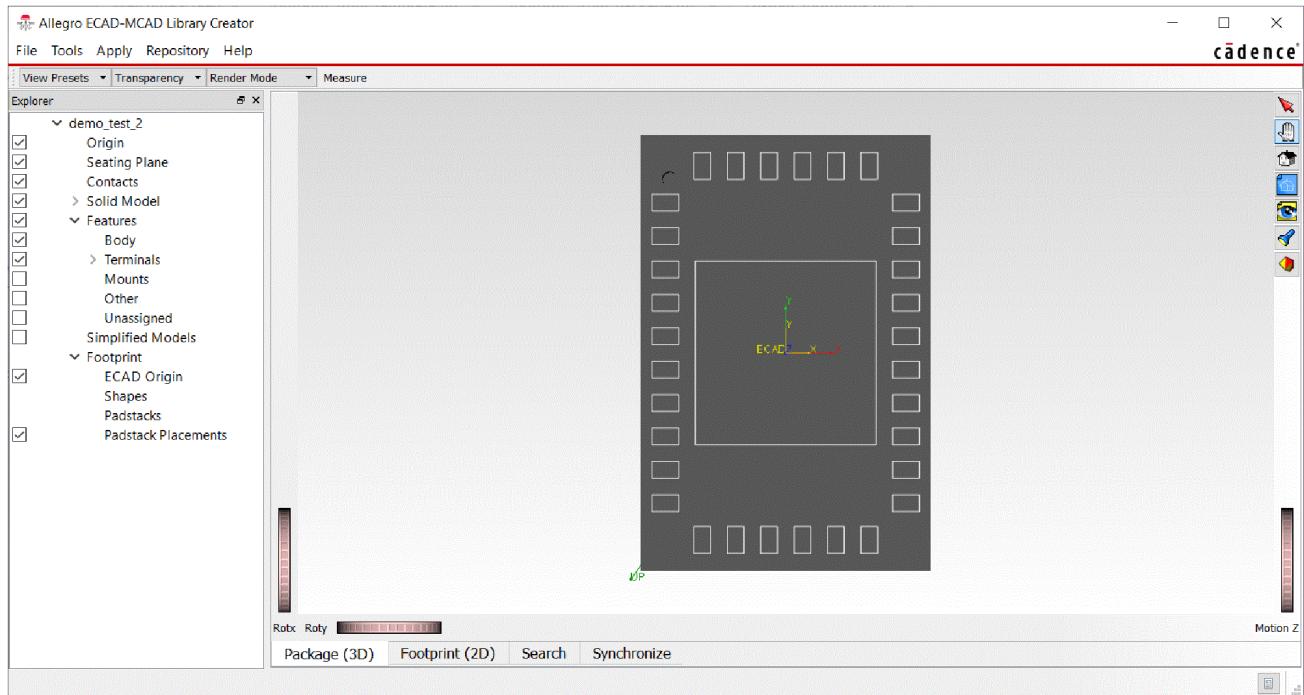
To create a new instance of a template:

1. Change the name in the parametric table to `demo_test_2`.
2. Change the values in the parametric table if required and click **New Instance**.

For this tutorial, we will keep the parametric values as it is.

3. Close the *Templates (Viewing)* dialog.

The Package (3D) view displays the template including a single large thermal pad that we can use as a starting point.



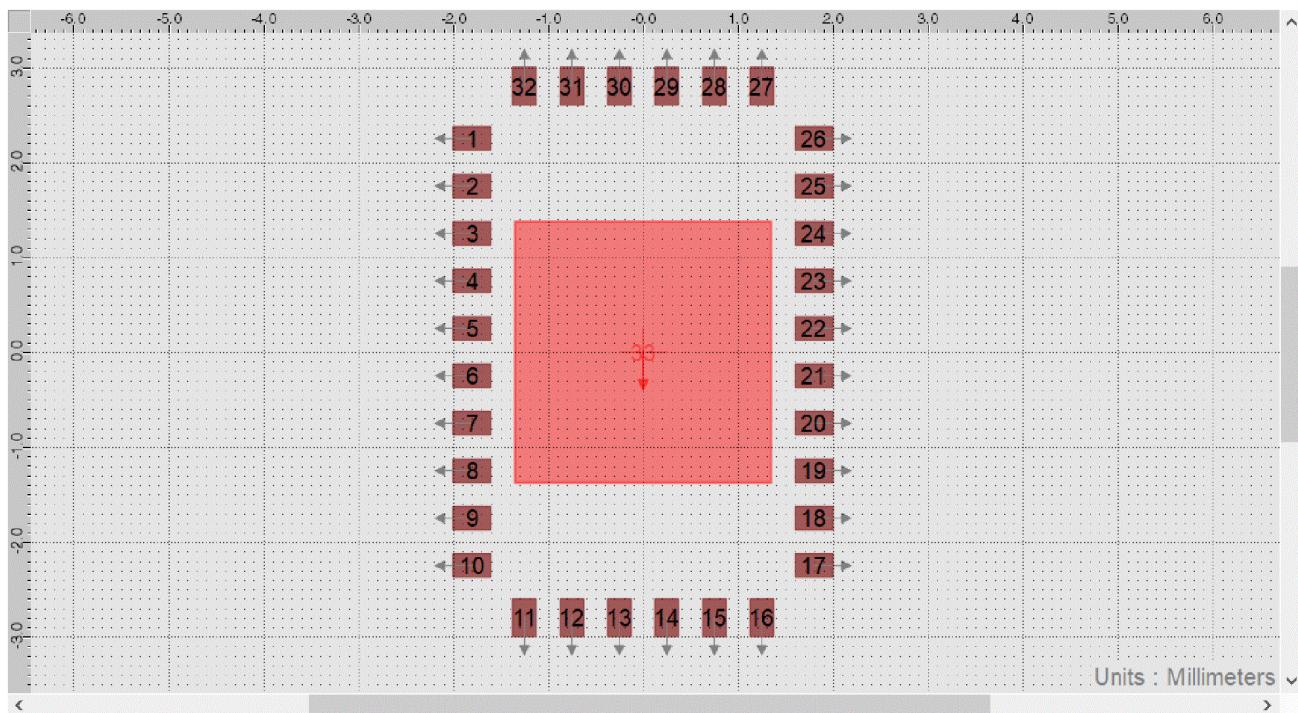
4. Click the Footprint (2D) view tab.

You will now work on editing the thermal pad.

Editing Thermal Pads

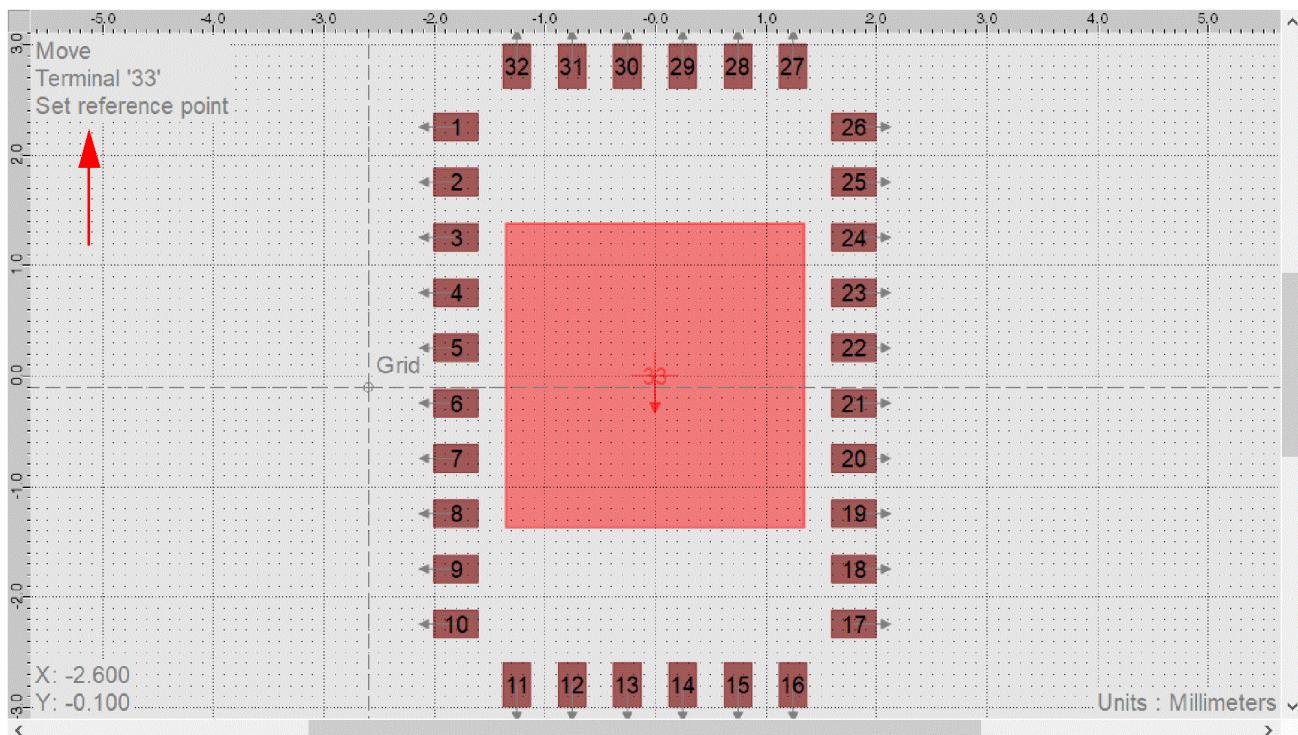
To edit thermal pads:

1. Click to select the middle thermal pad labeled 33.
The selected thermal pad turns red upon selection.



- Right-click the selection and choose *Move* or press *Ctrl + M*.

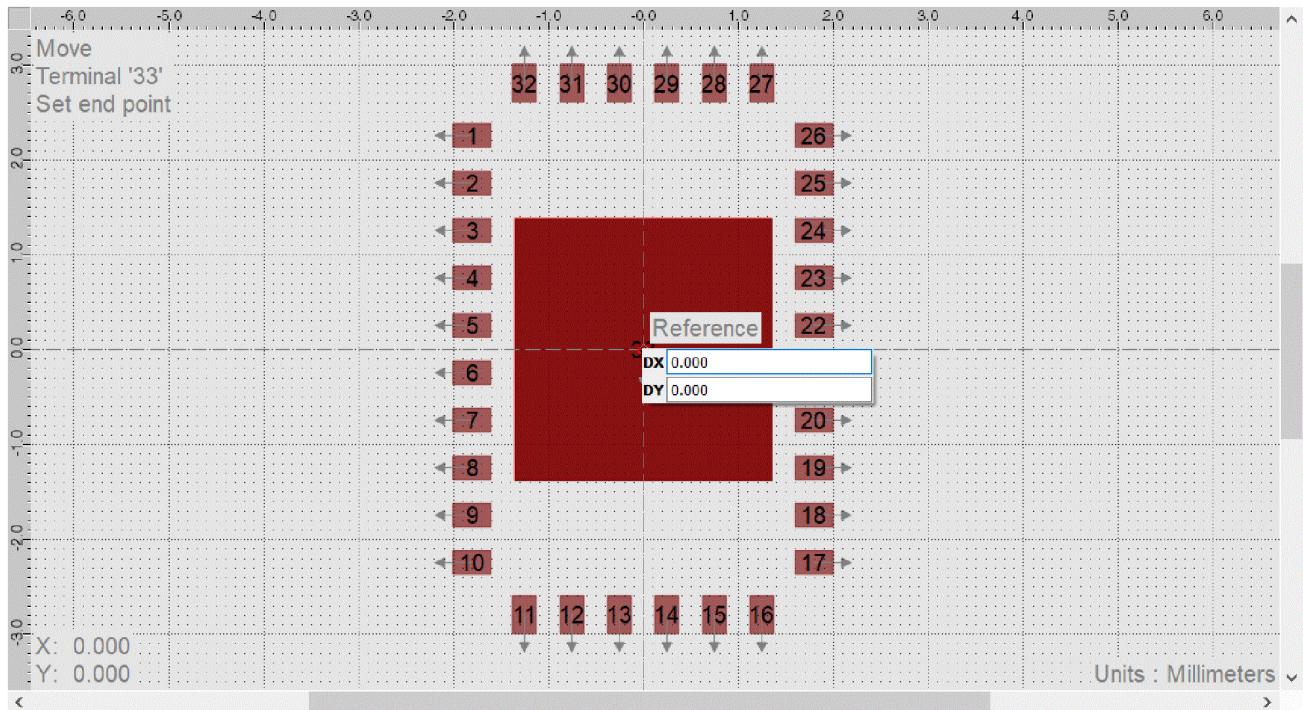
A X and Y location of the cursor appears at the bottom left of the canvas. The upper left corner displays the command in progress, the object originally selected and a guide as to the next step to be done – in this case *Set reference point* is displayed as the next step.



3. Click on the center point of the thermal pad 33.

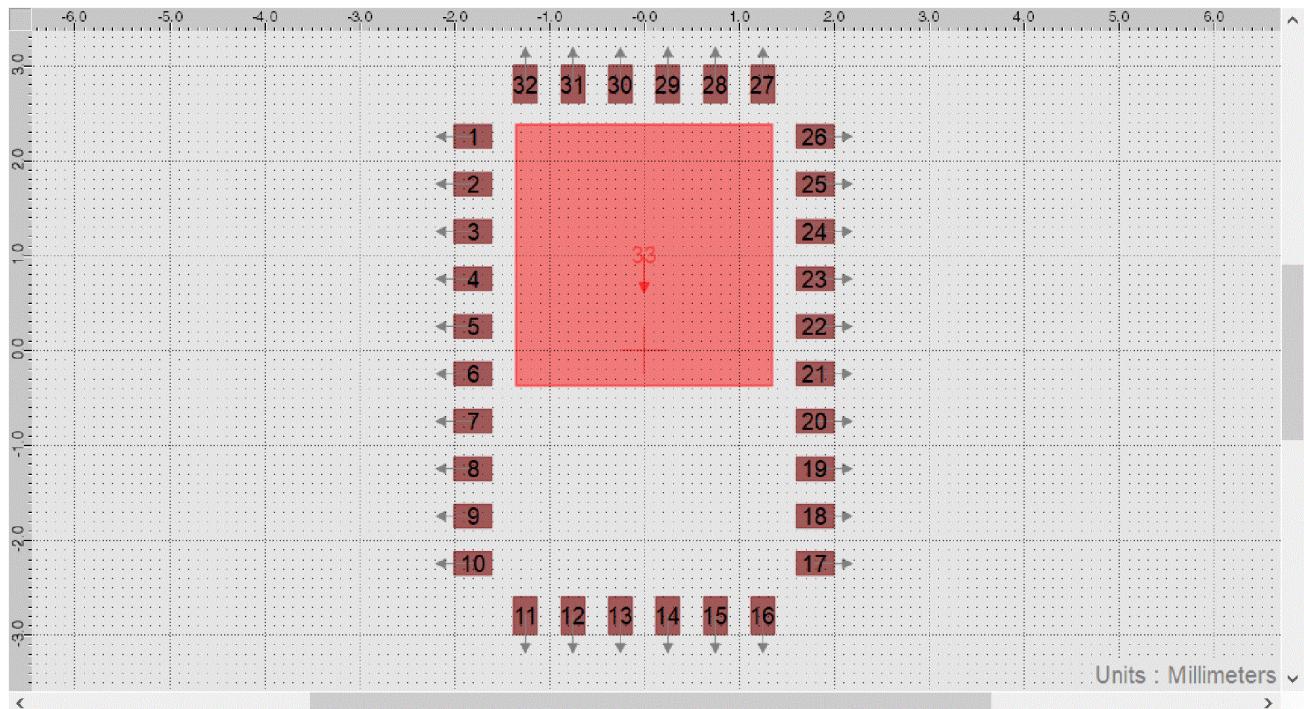
Library Creator provides an option to move the thermal pad using either a *Coordinate* or a *Delta*. For this module, we will use *Delta*.

4. Right-click and choose Delta.



5. Enter the value of DY as 1.0 and press Enter.

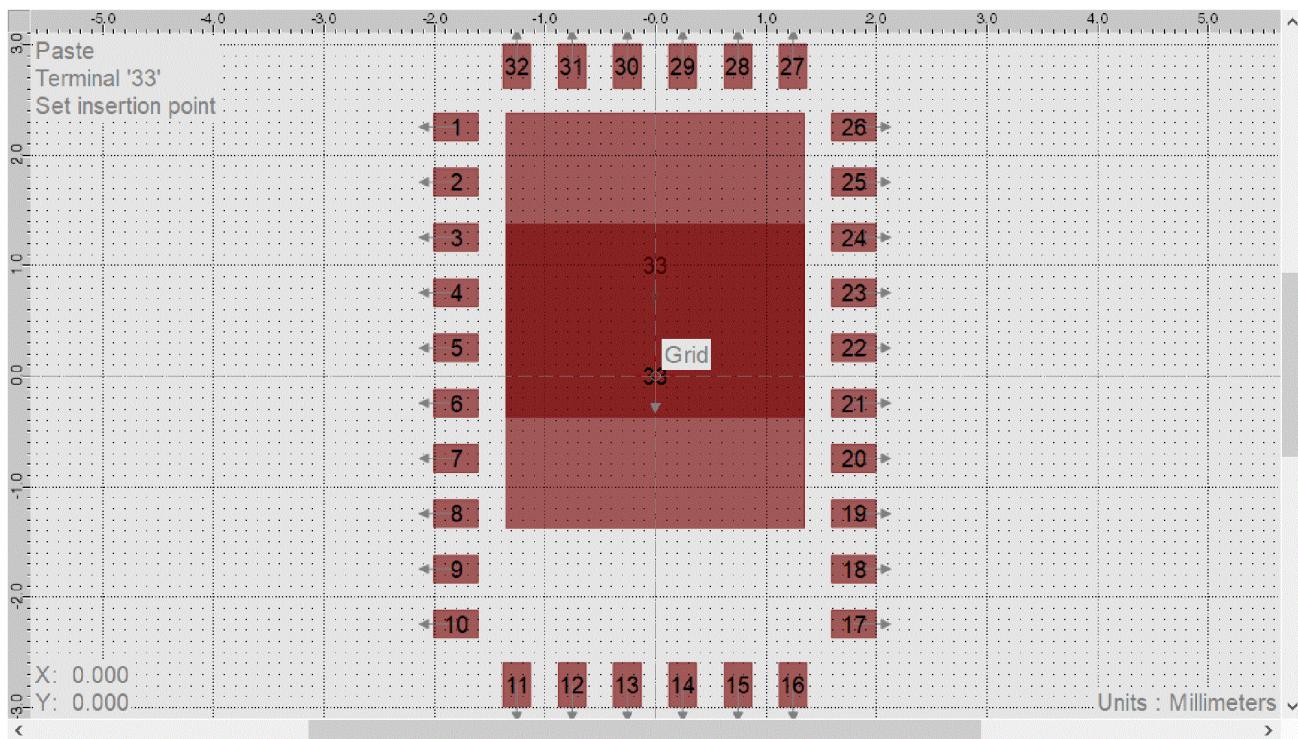
The thermal pad moves upwards in the footprint.



As the thermal pad is still red which means that it is still selected and can be acted upon.

6. Right-click and select Copy
7. Right-click and select Paste.
A copy of the thermal pad is attached to the cursor.
8. Locate the center of the footprint and click to place the copied thermal pad.

You can locate the center of the footprint by noticing the values of X and Y location at the bottom left of the canvas. These values must be zero.

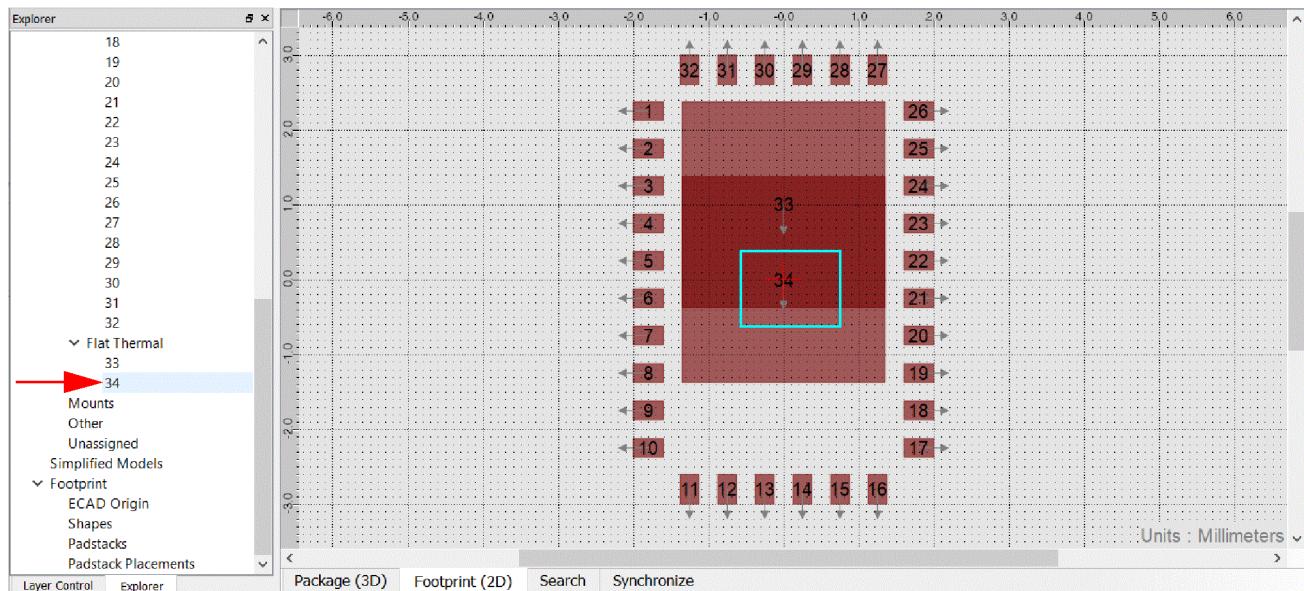


A new thermal pad 33 is added under *Flat Thermal* in *Explorer*. Next, we will change the name of the newly added thermal.

9. Right-click the newly added Flat Thermal 33 and select *Rename* in *Explorer*.
Enter New Name dialog box appears.
 10. Change the name to 34 in the *Name* field.
 11. Click *OK*.
- The name of the newly added thermal pad is changed to 34 in *Explorer* and is also reflected on the canvas.

Allegro X ECAD-MCAD Library Creator Tutorial

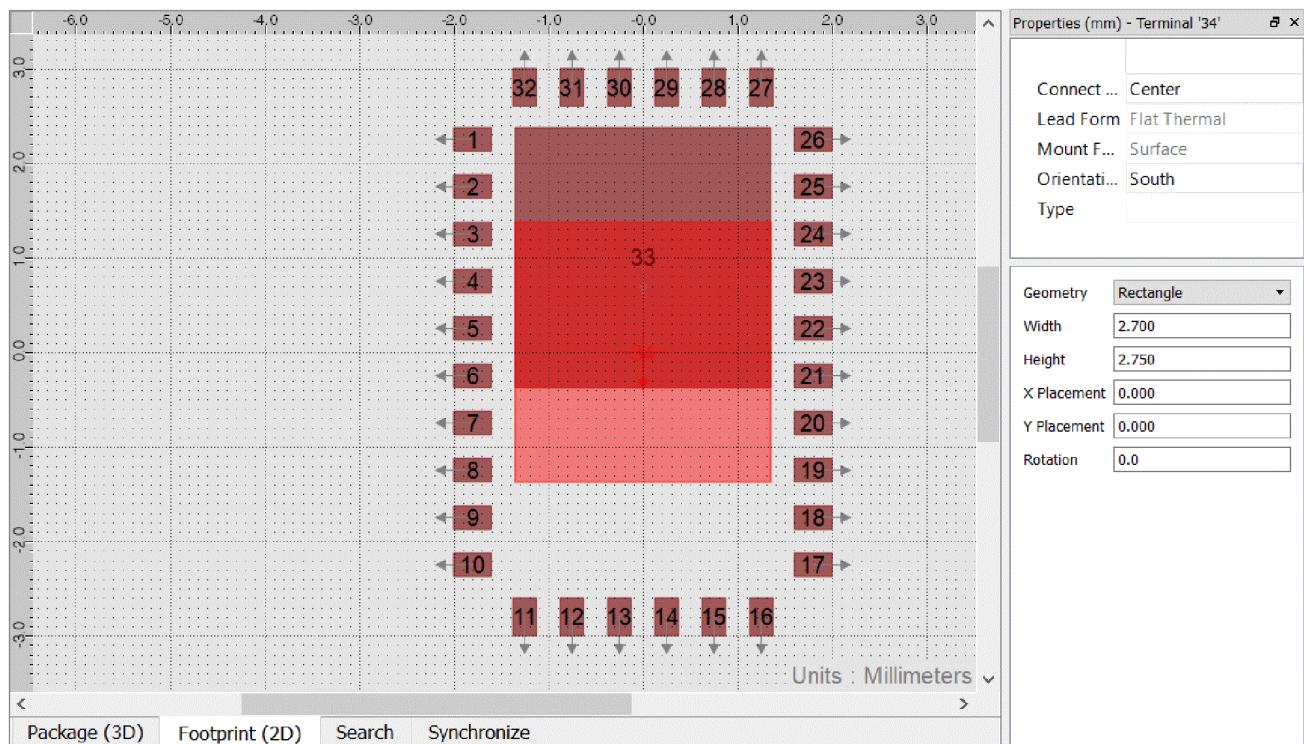
Creating Custom Footprints--Editing Thermal Pads



Next, we will change the height of the newly added thermal pad from the *Properties* panel.

12. Click *Tools - Properties*.

The *Properties* panel is displayed. As the thermal pad 34 is selected on the canvas, properties and its values related to the selected object (thermal pad 34) are displayed.



13. Change the value of Height to 1.7 and press *Enter*.

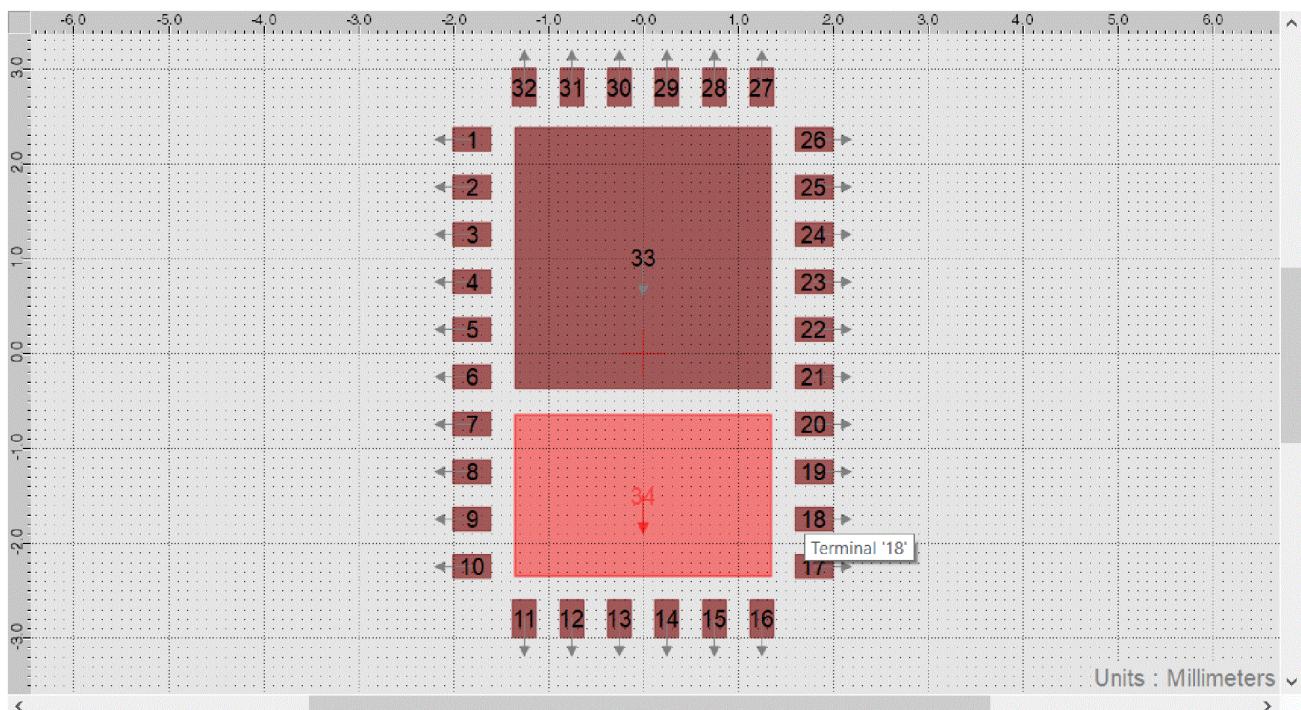
14. Right-click the thermal pad 34 and choose *Move* or press *Ctrl+M*.

15. Click on the center point of the thermal pad 34.

16. Right-click and choose *Delta*.

17. Change the value of DY to -1.5 and press *Enter*.

The thermal pad has moved to its new location as shown in the following figure.



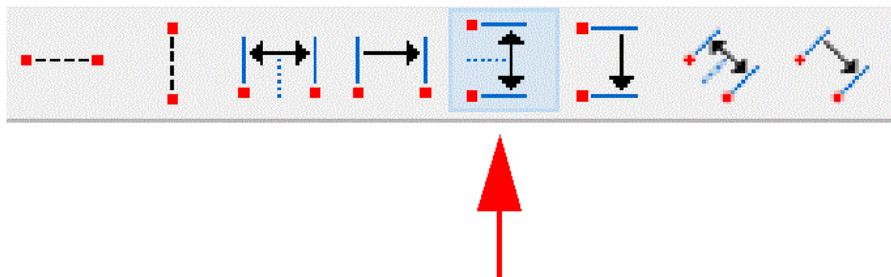
Now we will change the shape of the thermal pad 33 so that it matches the shape H.

Changing Thermal Pad Shape

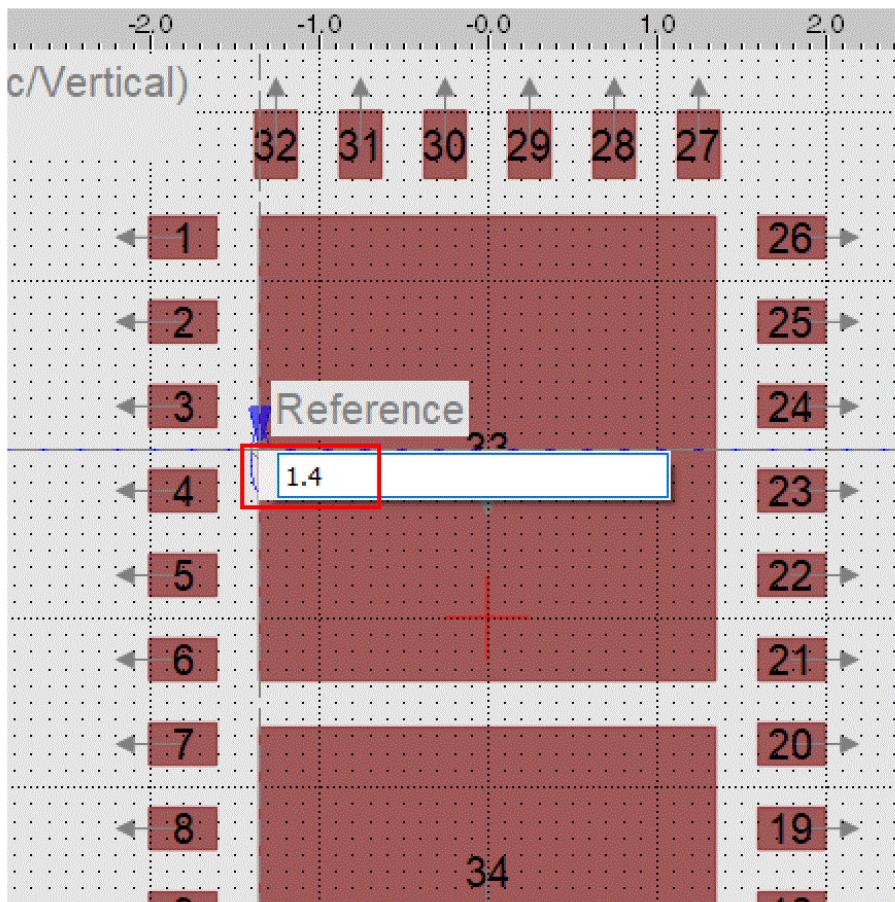
Library Creator allows you to easily change the shape of a thermal pad. You can do this by adding reference dimensions that are used as aids in sketching a shape outline of the thermal pad. In this section, you will change the shape of thermal pad 33 to H.

To change the shape of thermal pad 33:

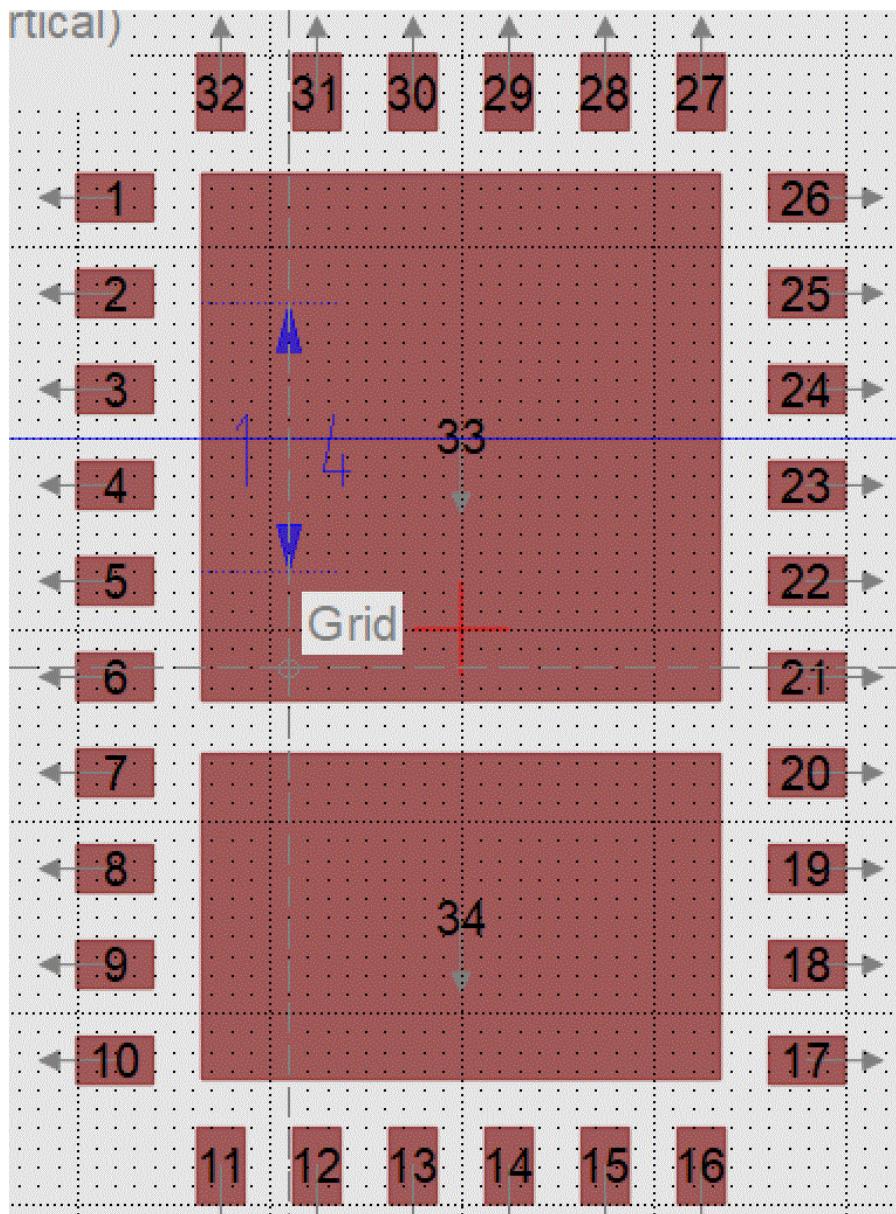
1. Right-click anywhere on the canvas and choose *Selection Filter - Shapes*.
2. Click on the *Add a Vertical Symmetrical Dimension* icon on the toolbar.



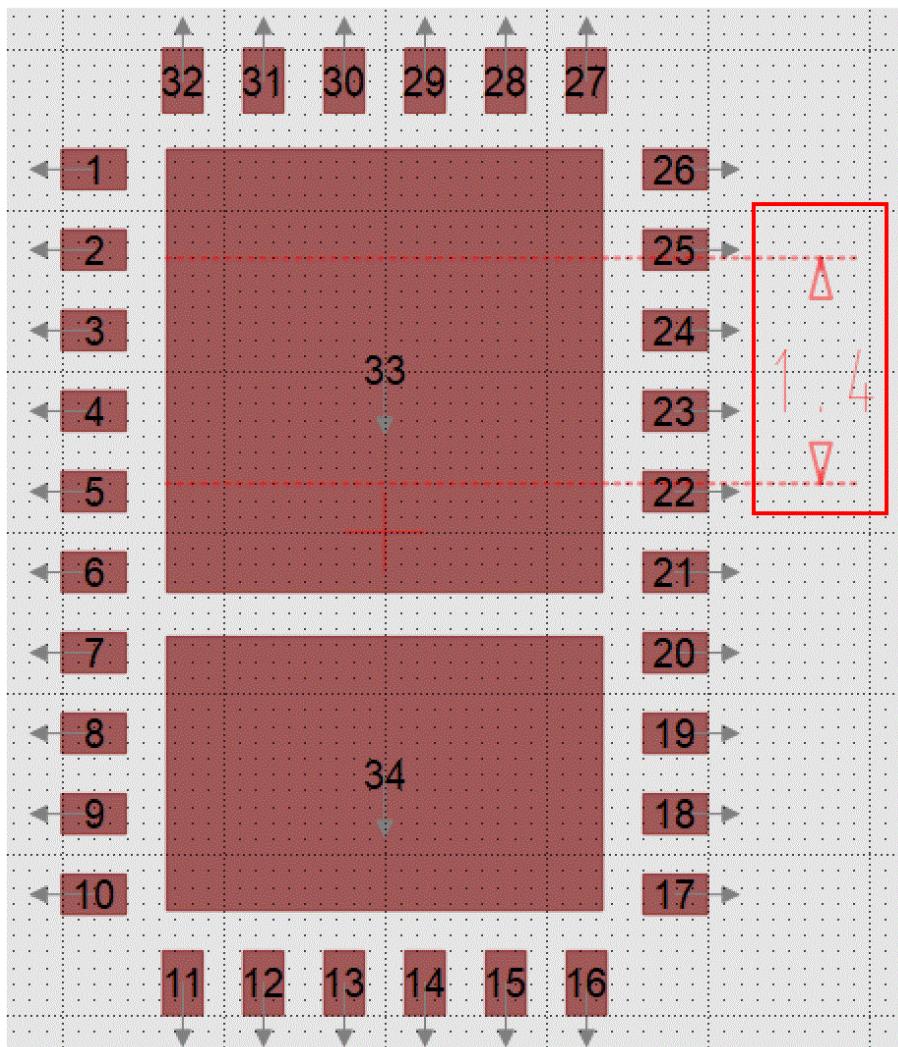
3. Click the middle left edge of the thermal pad 33 and move the mouse cursor slightly above.
4. Right-click to choose Set Dimension.
5. Type `1.4` and press *Enter*.



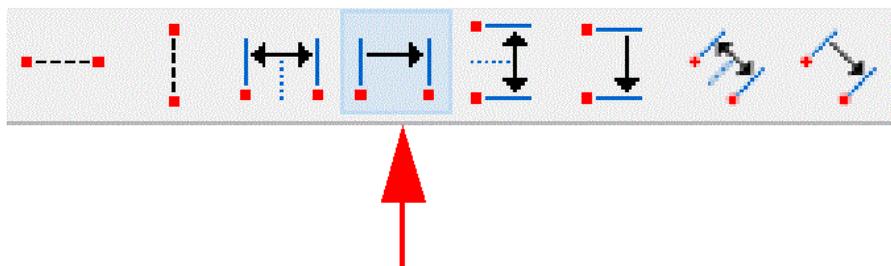
A blue dimension with value `1.4` is attached to the cursor.



6. Click on the right side of the footprint to place the dimension.



7. Click the Add a horizontal relative dimension icon on the toolbar.

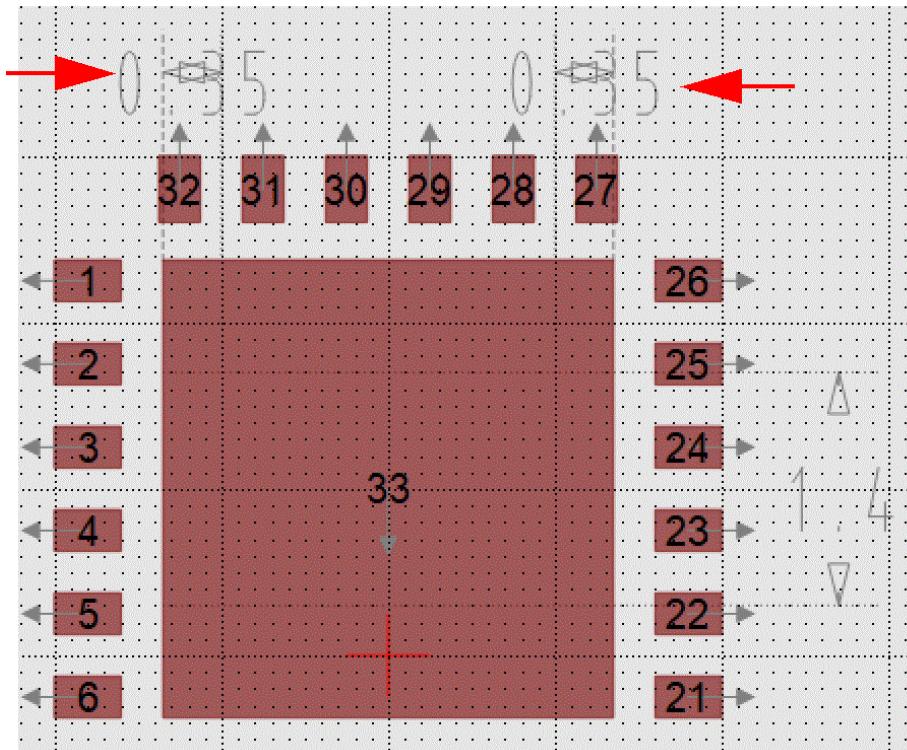


8. Click the top left corner of the thermal pad 33 and move the cursor to the right.
9. Right-click to choose Set Dimension.
10. Type `0.35` and press `Enter`.

A blue dimension with value `0.35` is attached to the cursor.

11. Click on the top left of the footprint to place the dimension.

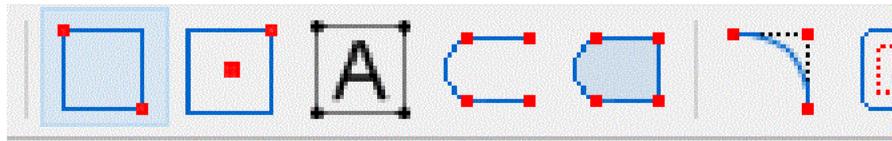
Similarly, add a horizontal relative dimension with value 0.35 to the top right corner of the footprint.



The reference dimensions are used as aids in sketching the H shape outline of the thermal pad. Next we will add some shapes to the thermal pad 33.

12. Select the thermal pad 33.

13. Click the *Add a new standard shape object* icon on the toolbar.

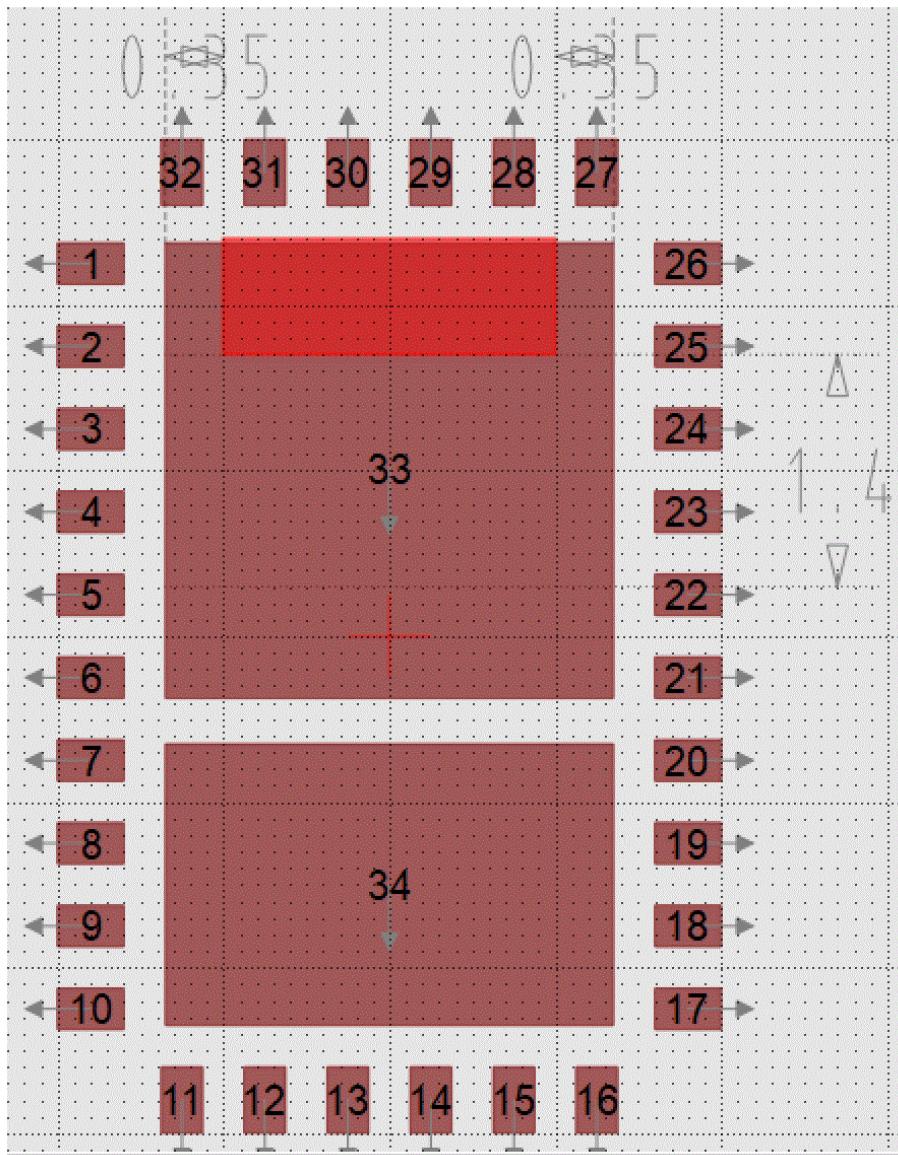


14. Move the mouse cursor to the top of the thermal pad and click when the values X and Y location displays -1.0 and 2.4 respectively.

15. Move the mouse towards the right-side of the footprint and click when the X and Y location

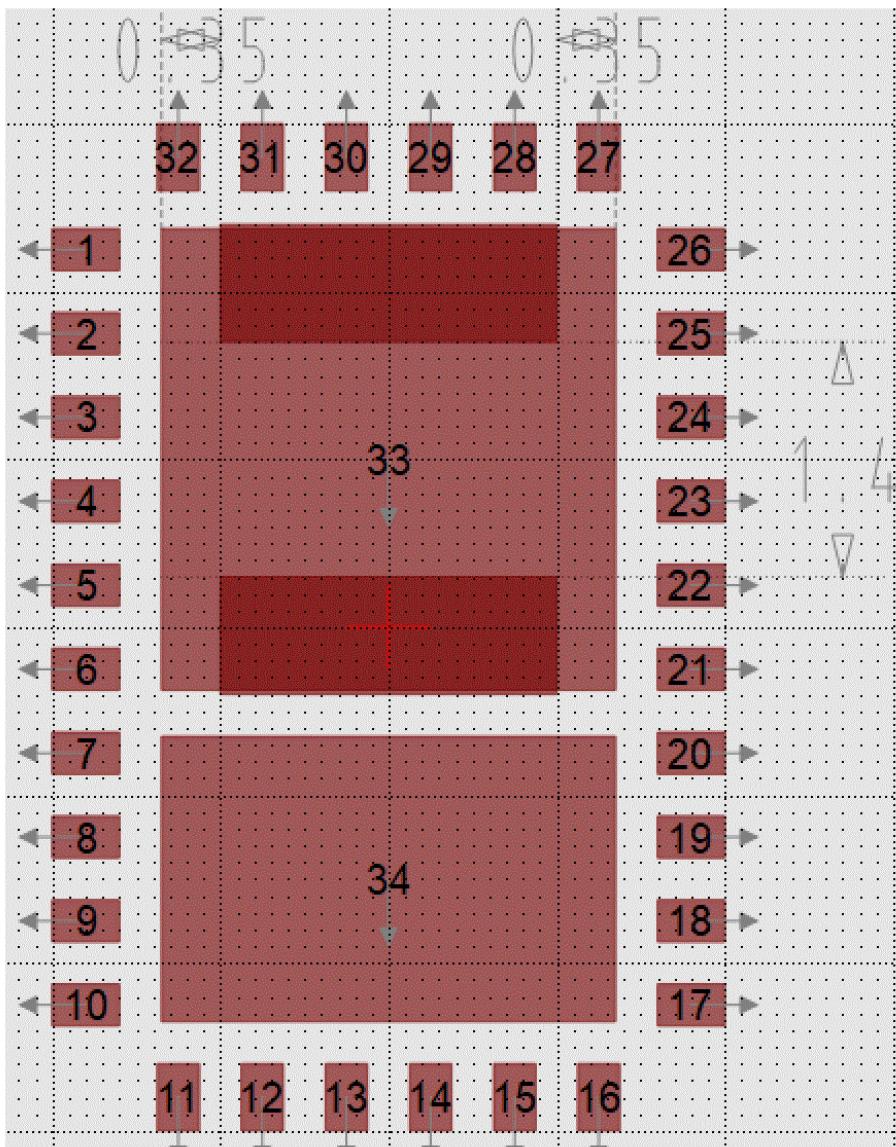
displays 1.0 and 1.7 respectively.

A rectangle is created as displayed in the following figure.

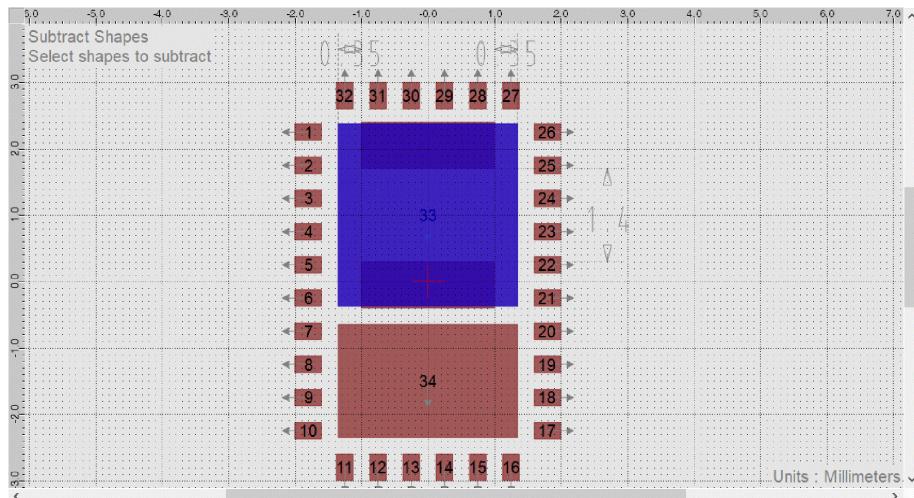


16. Similarly, create a rectangle at the bottom of the thermal pad 33.

When completed you should have two rectangles within the thermal pad 33.

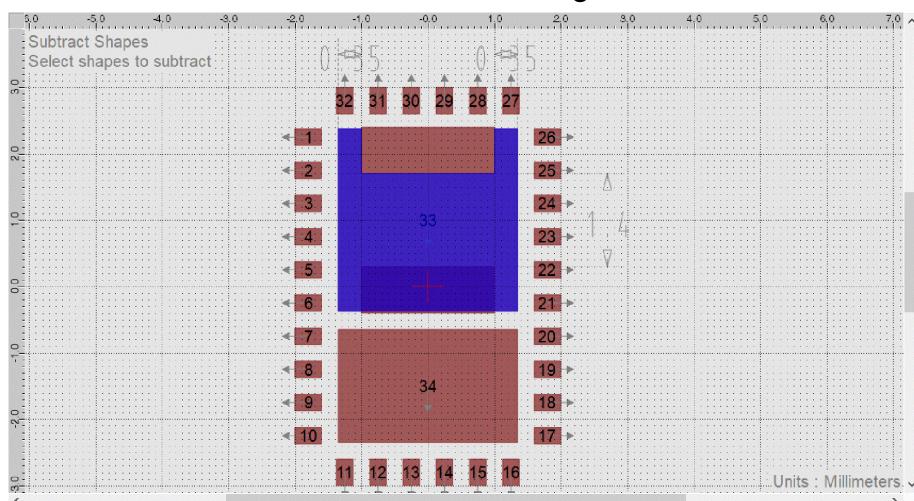


17. Click anywhere on the canvas to deselect the thermal pad 33.
18. Select the thermal pad 33 again.
To ensure that you select the thermal pad 33, hover cursor over datatip which says *Rectangle of Contact of Terminal '33'*.
19. Click  icon on the toolbar.
The selected thermal pad 33 turns blue and a message appears on the canvas letting you know that Library Creator is a subtract shapes mode.

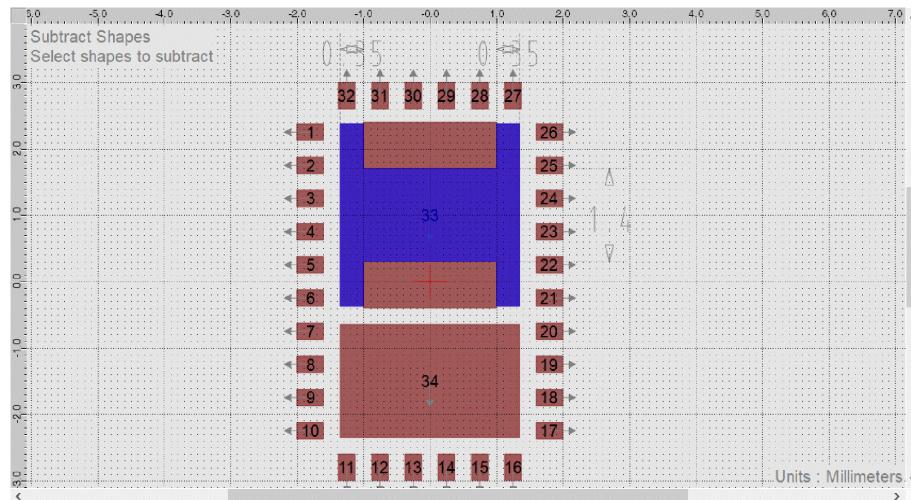


20. Click the rectangle created on the top of the thermal pad 33.

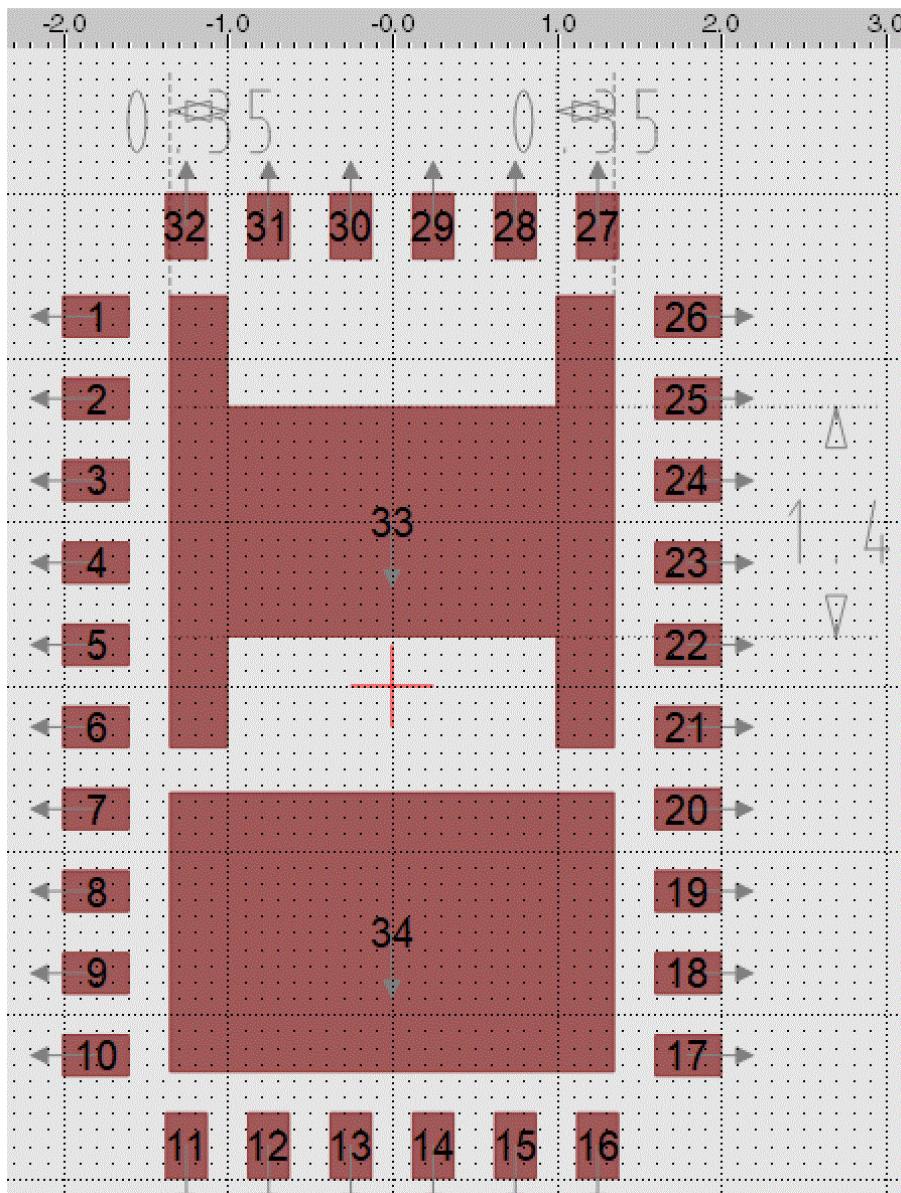
This removes the area under the rectangle.



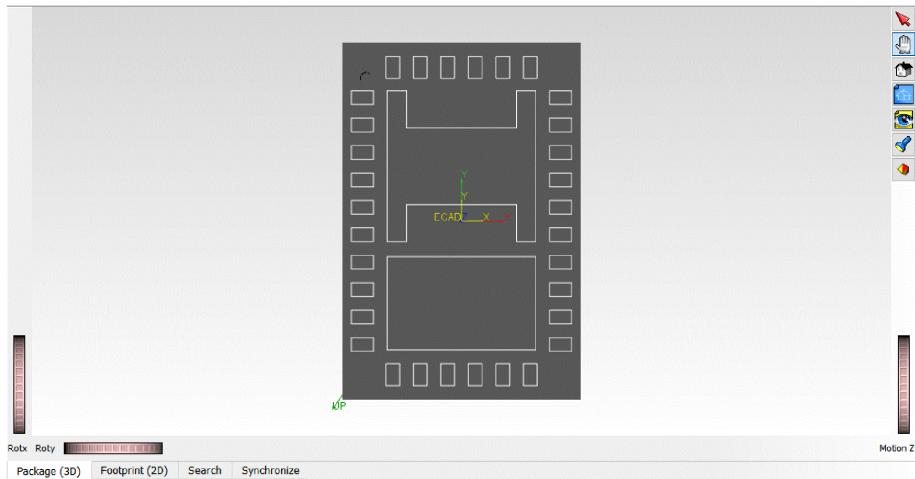
21. Similarly, subtract the rectangle created on the bottom of the thermal pad 33.



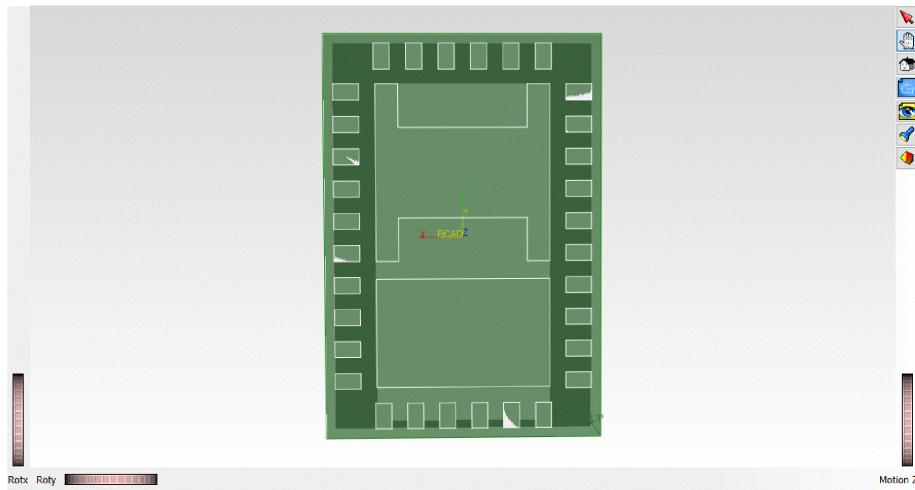
22. Click anywhere on the canvas to deselect the thermal pad 33.
23. Select the rectangle on the top and bottom of the thermal pad 33. To do this:
 - a. Hover cursor over the rectangle on the top and click to select it.
 - b. Now, keeping the *Ctrl* key pressed, select the rectangle on the bottom.
24. Press Delete on the keyboard.
The selected rectangles are deleted.



25. Right-click anywhere on the canvas and choose *Selection Filter - Auto*.
26. Click to select the reference dimensions and press *Delete*.
27. Click *Package (3D)* to see the changes reflected in the *3D* view.

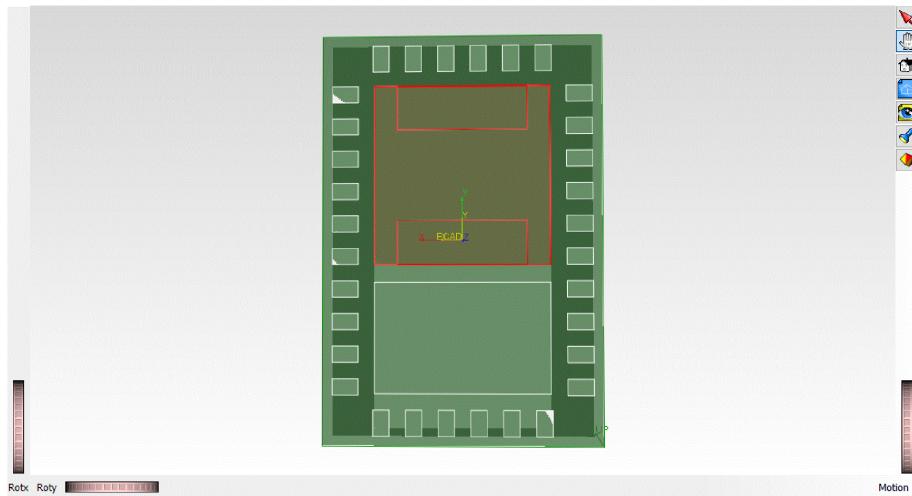


But if you spin the 3D image around to the back side, you will notice that only the contact has been updated the H shape. The actual thermal pad on the 3D STEP model has not yet been updated as shown in the following figure.



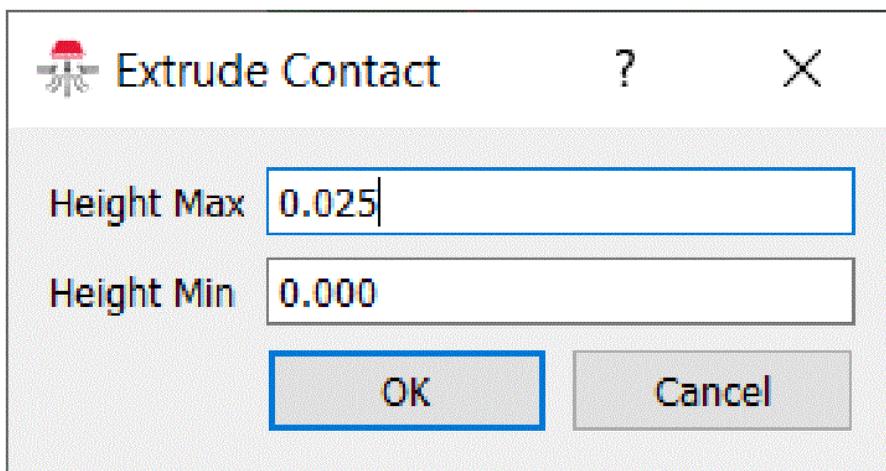
Updating Thermal Contacts

1. In the *Explorer* panel, under Flat Thermal, select 33.



2. Right-click and choose Extrude Contact.

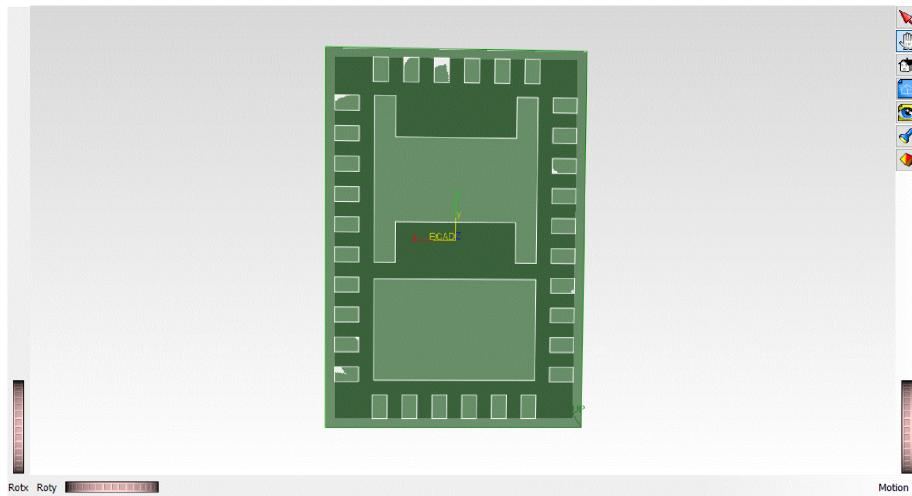
Extrude Contact dialog is displayed.



3. Click *OK*.

Similarly, extrude contacts for pin 34.

The underside of the STEP model now matches the new contacts that were added.



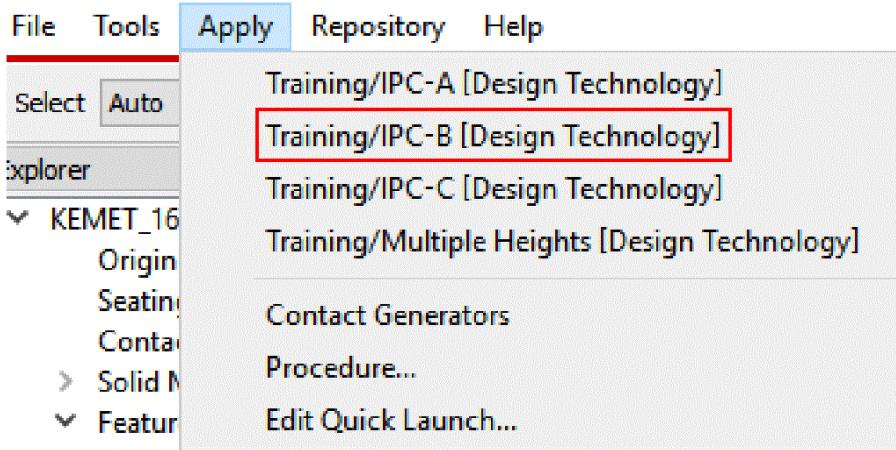
Saving the Footprint

For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

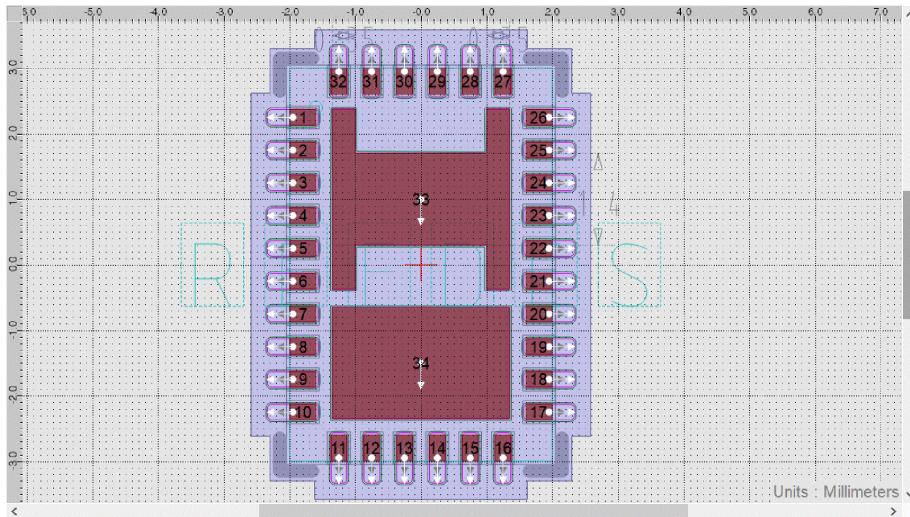
Applying Rules to Package

To apply rules:

1. Choose *Apply - Training/IPC-B [Design Technology]*.



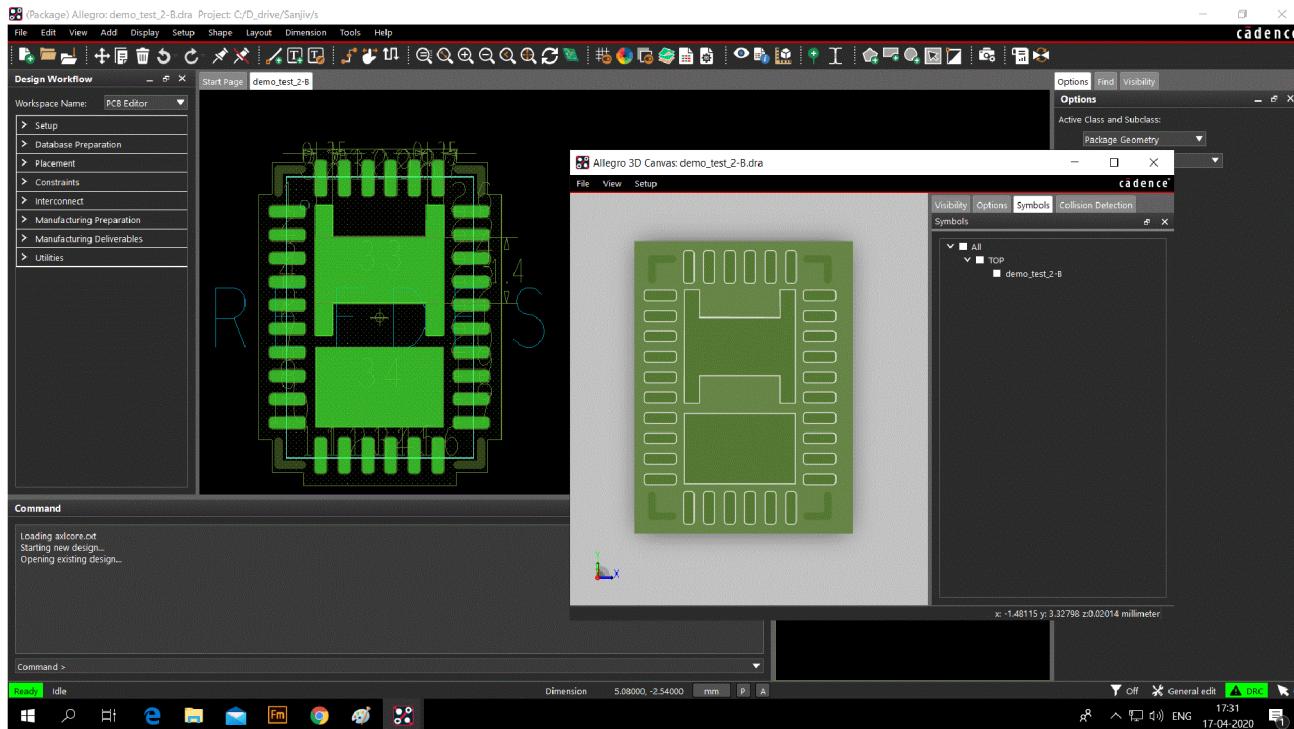
Footprints matching the selected rule are created.



Exporting Footprint to Allegro PCB Editor

To export the footprint to Allegro PCB Editor, refer to the steps mentioned in the [Exporting the Footprint to Allegro PCB Editor](#) section.

The custom thermal shapes are reflected in the footprint exported to Allegro PCB Editor.



Summary

In this module, you learned loading a template, modifying and creating thermal pads, adding reference dimensions, and changing shape of thermal pads.

Creating Footprints from a STEP Model

What You Will Learn

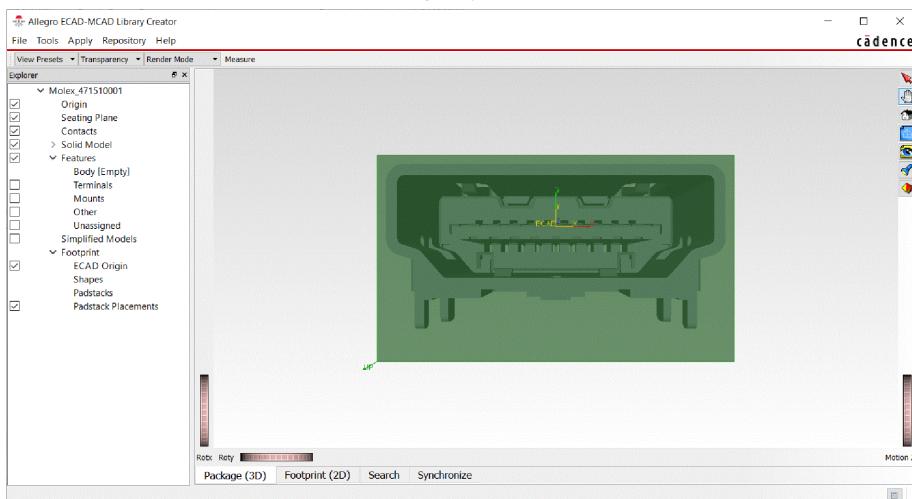
In this module, you will learn how a vendor provided STEP model can be used to create an Allegro PCB Editor footprint and then exported to Allegro PCB Editor along with the original STEP model.

Importing STEP Models

In this section, you will import the step model molex_471510001.stp from
<your_install_dir>/doc/lc_tut/tutorial_examples/STEP_models.

To import the step model molex_471510001.stp:

1. Choose File - Import - STEP.
The *Select STEP file* window is displayed.
2. Locate and select the model molex_471510001.stp file and click Open.
The Molex_471510001 is displayed in *Explorer* as well as on the canvas.

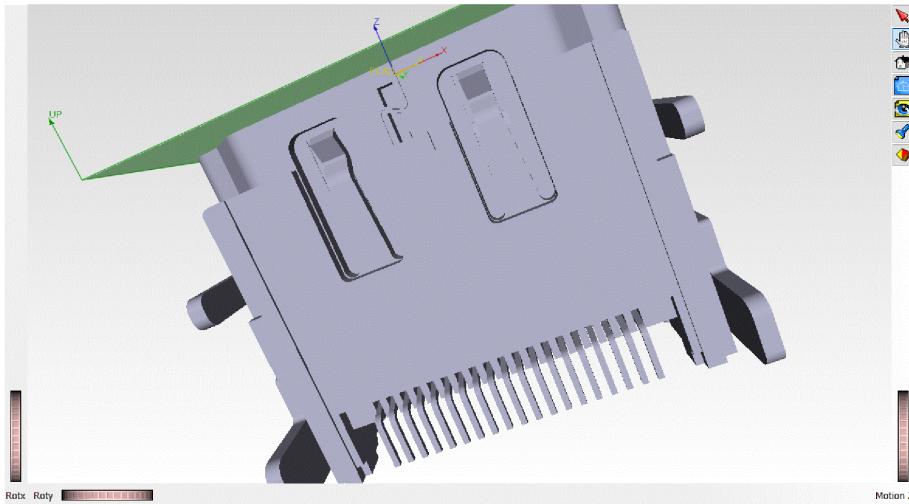


Notice that the green Seating Plane is not located on the correct face of the connector. The Seating Plane refers to the surface of the PCB that the component will rest on after

installation.

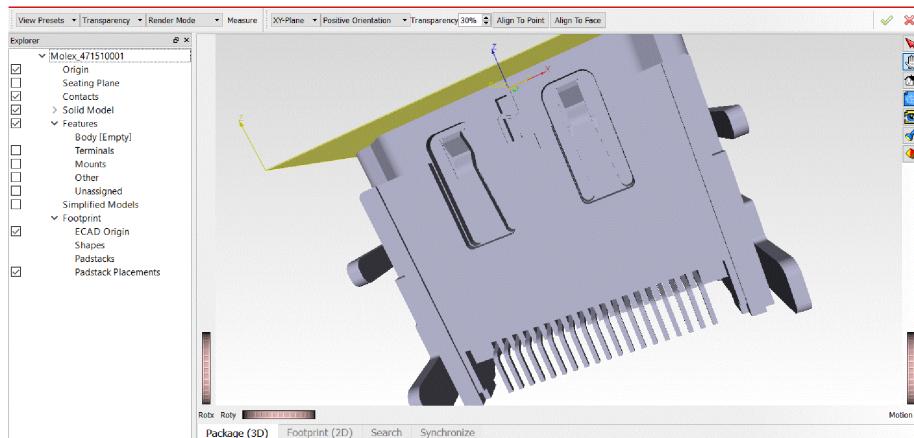
Editing Seating Pane

1. Spin the STEP Model to match the view as shown in the following figure.

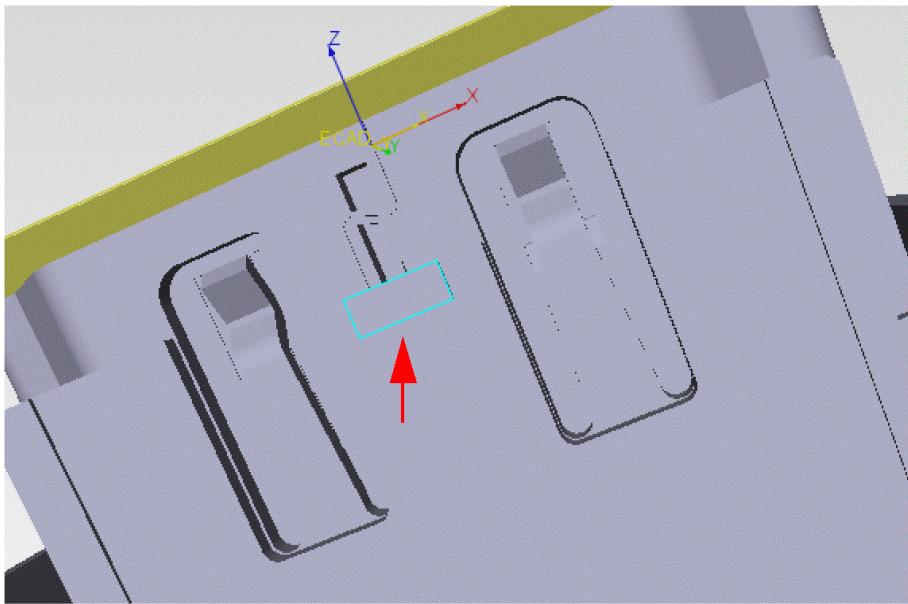


2. Right-click Seating Plane and choose Edit in Explorer.

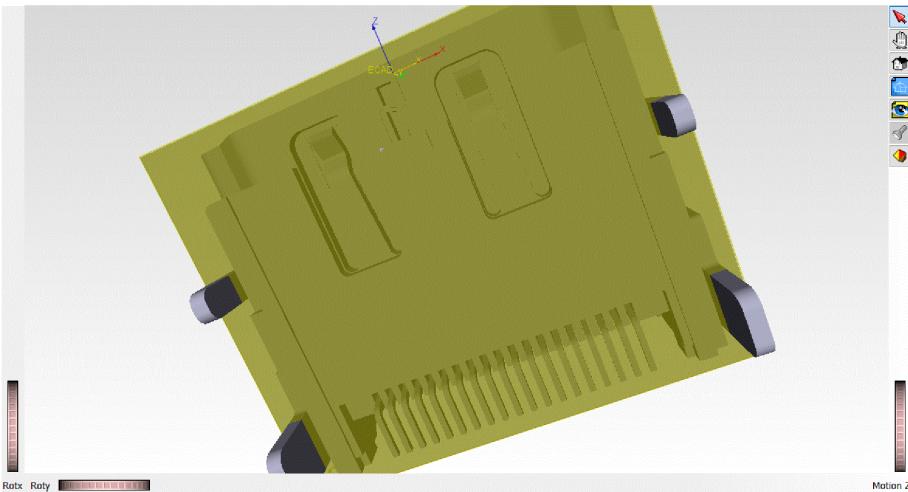
The color of the seating plane changes signifying that it is now in edit mode.



3. Click the Align To Face button on the toolbar and then click the red selection arrow toolbar icon on the right side.
4. Move the cursor around the underside and click the connector highlighted in red in the following figure.



The seating plane repositions itself so that it is parallel with that surface.

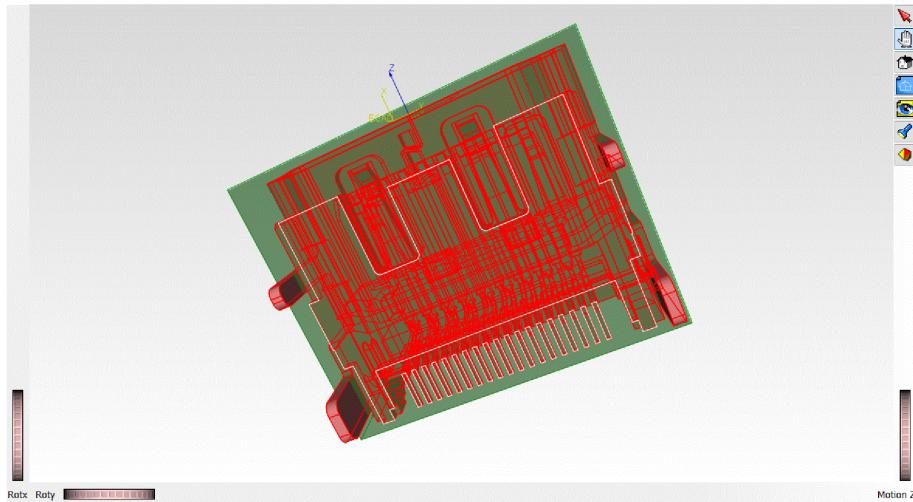


5. Click the *Accept* button on the top right of the toolbar.

The seating plane returns to its original color.

Adding Contact Features

1. In Explorer, expand right-click *Solid Model - Solid (Unassigned)*.
2. Right-click *Solid (Unassigned)* and choose *Contact Features - Training/SMT Features*. All SMD contacts that touch the seating plane, 19 pins and one large metal area, are highlighted in red.



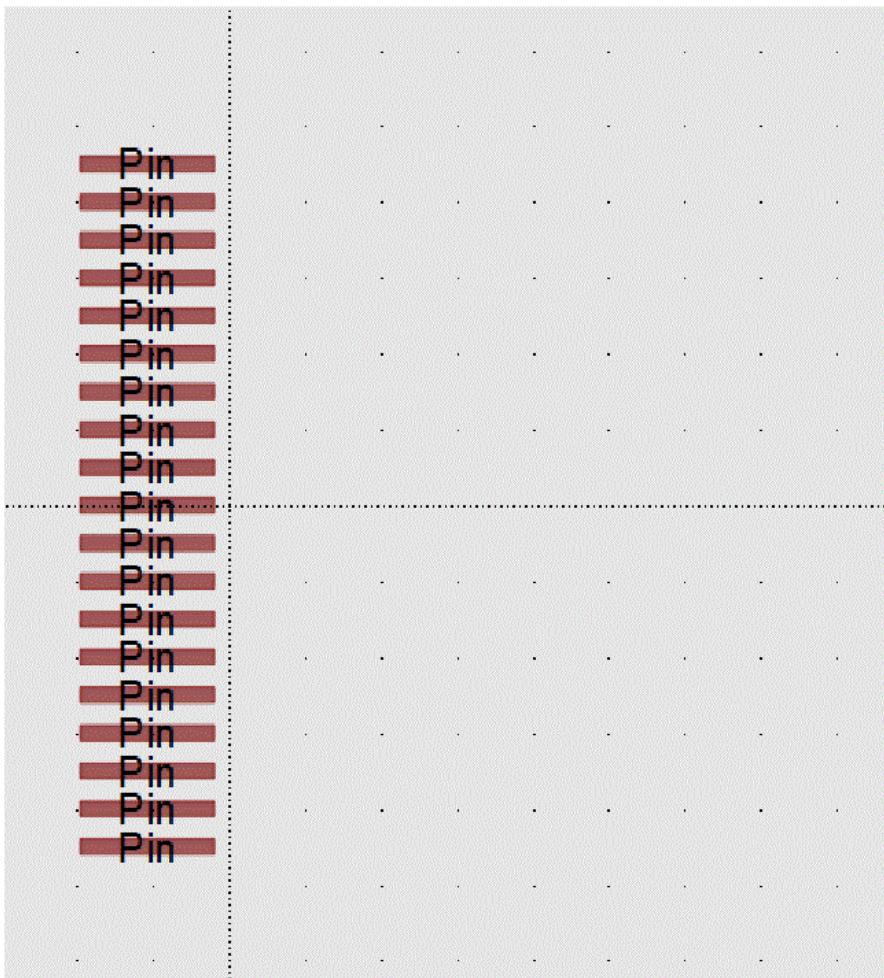
3. Click the Footprint (2D) view.

The SMT Features are automatically added under Feature - Unassigned in Explorer.

Also, Library Creator automatically groups all the 19 objects that are of the same size and the single large contact area is added as Feature.

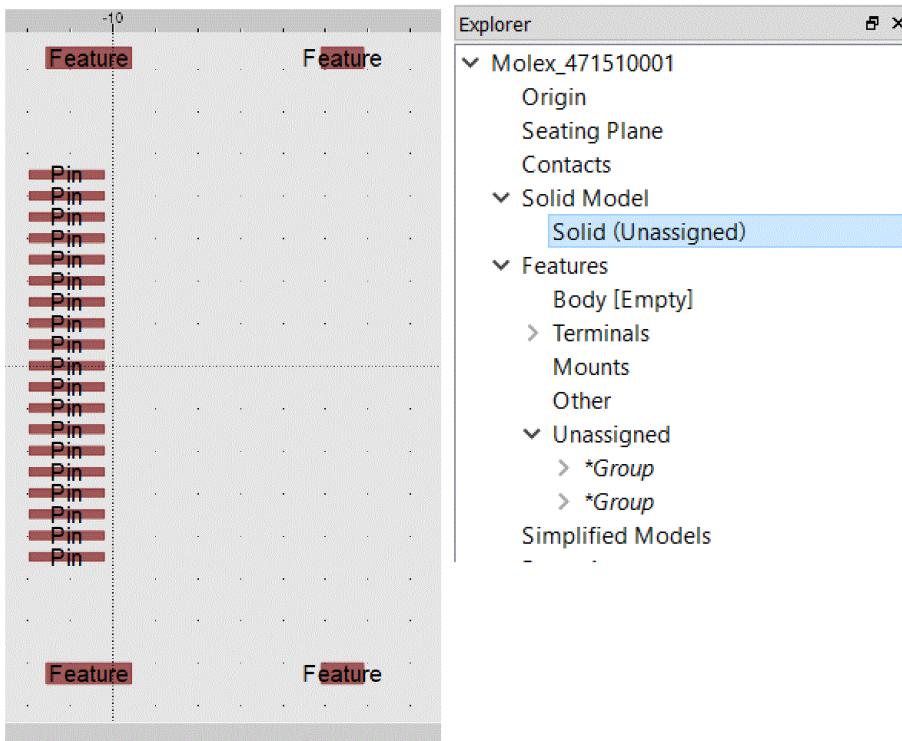
4. Select and drag Feature - Unassigned - Group in *Explorer* and drop it to Feature - Terminals.
5. Right-click Feature - Unassigned - Group and choose *Delete*.

Only the 19 contact areas remain on the canvas.

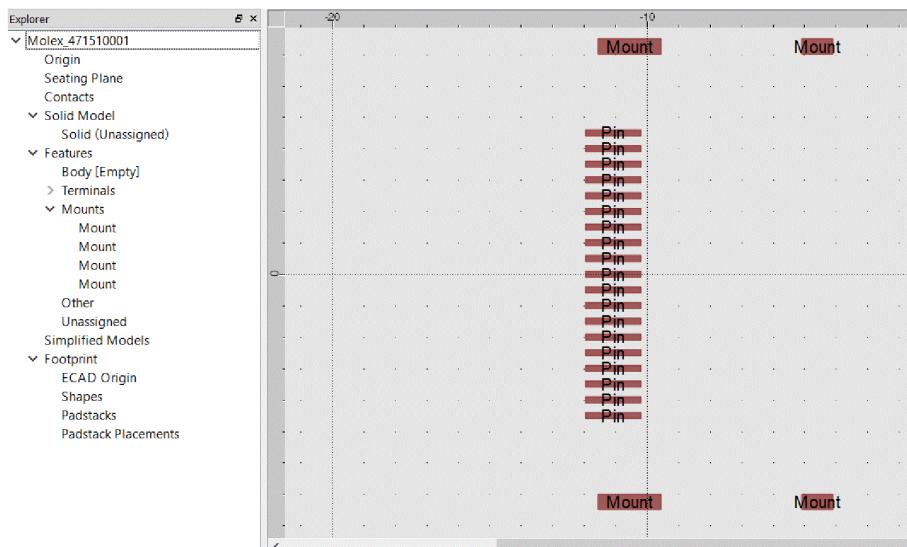


Next, you will adding contact features by adding the thru-hole features of this connector.

6. In Explorer, expand right-click *Solid Model* - Solid (Unassigned).
7. Right-click Solid (Unassigned) and choose Contact Features - *Training/THT Features*.
Four thru-hole contact areas are added and the four features are also added to the *Features - Terminals - Unassigned* in *Explorer*.
In this case, two groups are added because the two contacts are of the same size and the other two smaller contacts are the same size.



8. Select and drag both the *Group* under *Unassigned* and drag them to the *Features - Terminals - Mounts*.



⚠️ Mounts is the terminology used by Library Creator for mechanical pins while Terminals corresponds to signal pins.

9. Select all the four Mounts

10. Right-click the selection and choose *Set Type – PTH - Slot* to change the type of mounts to *Plated Thru-Slots*.

Next you will assign terminal types to the 19 contacts.

Assigning Terminal Types

1. In *Explorer*, under *Features - Terminals*, choose *Group*.
2. In the *Properties* panel, double-click the value of *Lead Form* and select *Gull Wing* from the drop-down list.
3. Click the Package (3D) view tab.
4. In the Explorer panel and select and drag the *Solid Model - Solid (Unassigned)* to the *Features - Body [Empty]* node.
The *Features - Body [Empty]* node will change to *Feature - Body*. This tells Library Creator that the complete 3D shape should be used to calculate the placement keepout.
When featuring a STEP model, it's important to assign the model height parameter.

Assigning Model Height

1. Right-click *Molex_471510001* and choose *Parameters - Edit* in *Explorer*.
The *Edit* dialog is displayed.
2. Click the Height check-box and then click the Compute button corresponding to *Height*.
Library Creator calculates and displays the height automatically as 6.23.
3. Type *+/- 0.05* after 6.23 in the *Height* field.
4. Click OK.

Assigning Pin Numbers

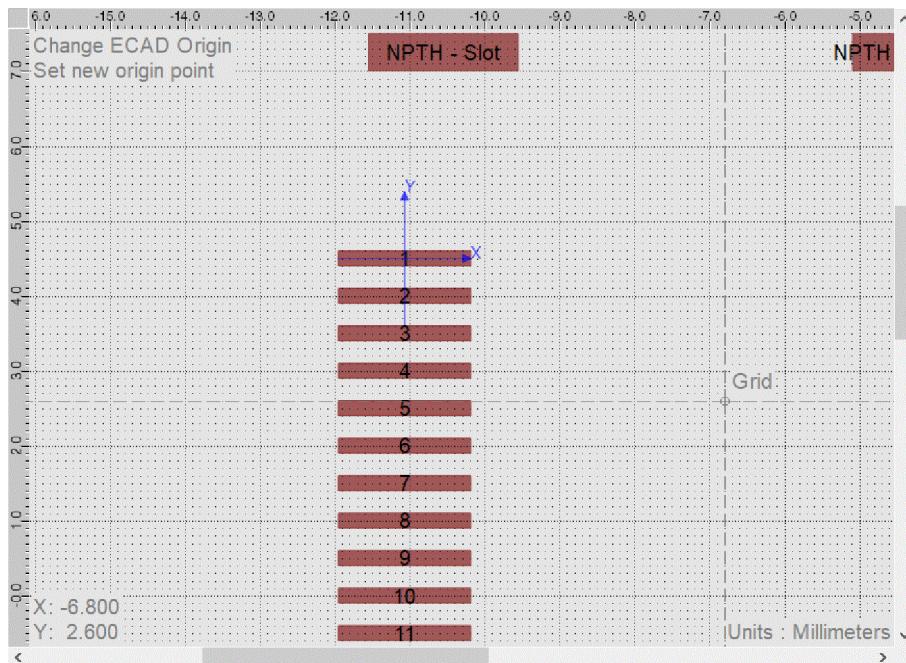
In this section, you will assign pin numbers to the 19 pins.

1. In the 2D view, select the 19 contact areas by creating a selection box around the contact areas.
The *Features - Terminals - Group – Gull Wing* node expands displaying the 19 unnumbered pins.

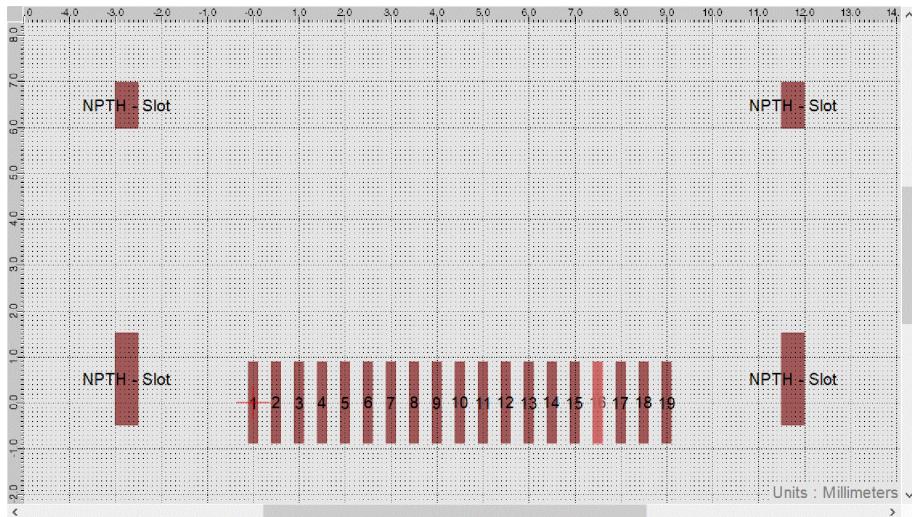
2. Click on the toolbar.
3. Type **1** in the *First* field on the toolbar and click the *Accept* button.
The pin numbers in *Explorer* indicate Pin 1 as the primary pin.

Editing the Origin

1. Right-click Footprint - ECAD Origin in Explorer and choose Change ECAD Origin.
This helps to better align a part when using in a design. The cursor changes to cross hair.
2. Zoom in to Pin 1.
3. Move the cross hair over pin 1 and click to select that pin as the new origin.
The X and Y cross hair should also now be on Pin 1.

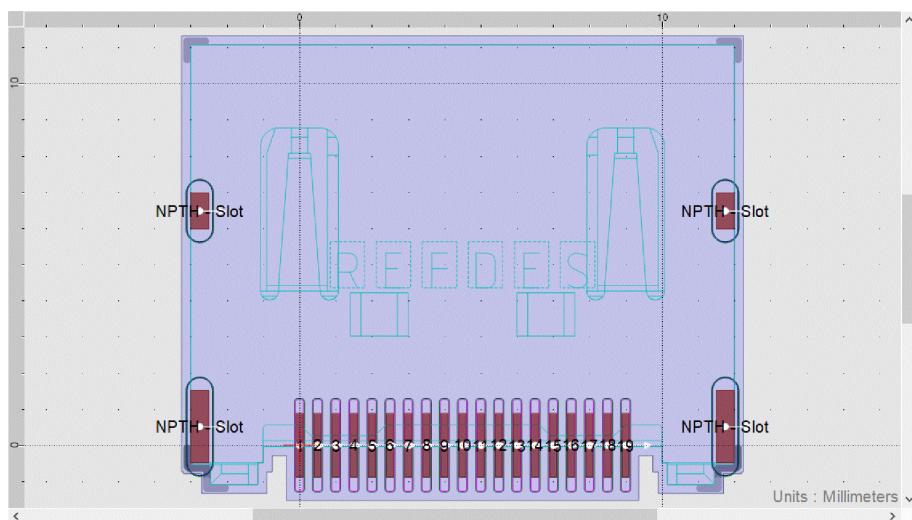


4. Select *Sync 3D Origin* check box.
5. Click on the toolbar to rotate the package and align it with the STEP model.
6. Click the *Accept* button on the toolbar.
7. Click to fit the footprint on the canvas.



Applying Rules

In this section, you will apply the Training/IPC-B (Design Technology) rule to create the footprint. For more information about applying rules, refer to the [Applying Rules to Template](#) section. The footprint matching the IPC-B rule is created.



Next, you will upload this featured version of your package to the repository.

Saving the Footprint

It is important that you save a new version of the footprint before exporting it to PCB Editor.

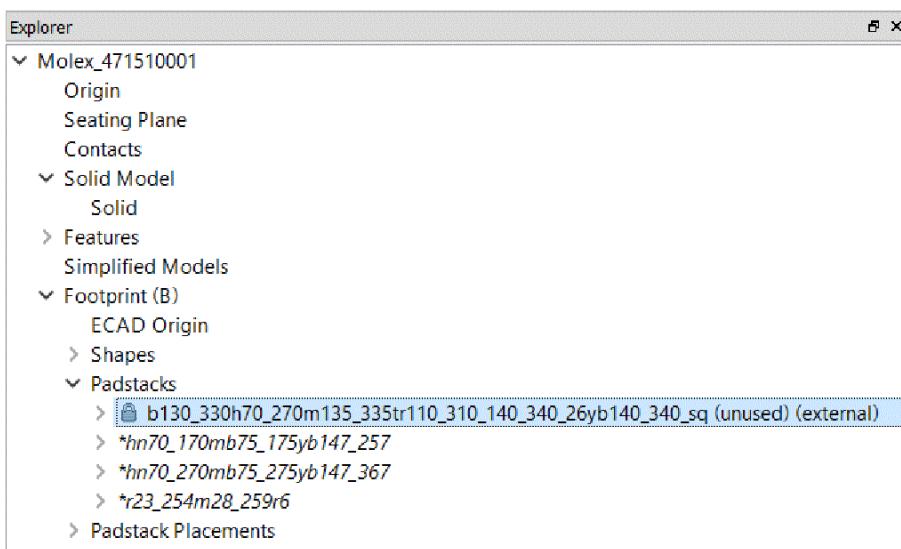
For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

Reusing Existing Padstacks

In this section, you will reuse the existing padstack *b130_330h70_270m135_335tr110_310_140_340_26yb140_340_sq.pad*. You will also replace the two larger slotted plated holes with a padstack that has external rectangle pads.

Importing External Padstack

1. Right-click *Footprint - Padstacks* and choose *Import*.
Import Padstack dialog is displayed.
2. Type b130.
Padstacks matching b130 are listed.
3. Select *b130_330h70_270m135_335tr110_310_140_340_26yb140_340_sq* and click OK.
The selected padstack is listed under the *Padstacks* in *Explorer*.



Notice the following:

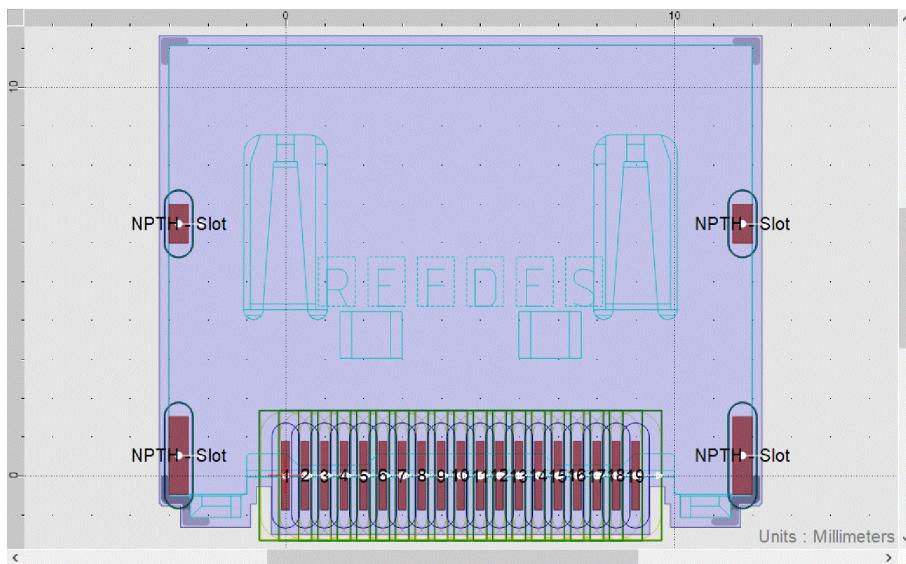
- A lock symbol and *external* appear at the start and end of the newly imported padstack name respectively. This indicates that this padstack cannot be changed from within Library Creator that the padstack is generated from an external source.
- (unused) in the name of the padstack indicates that while we have imported this padstack into Library Creator, it has not yet been used in the footprint under development.

You can use external padstacks in the footprint under development in two ways –

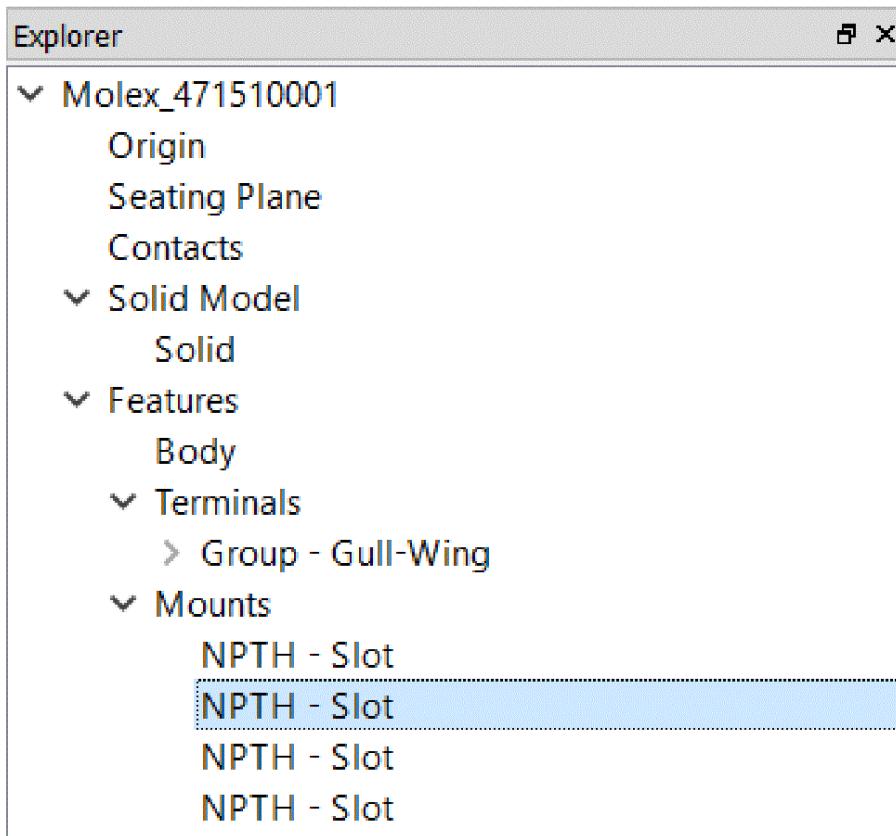
- by replacing one padstack with this new padstack. In this case, every pad instance using the original padstack is replaced with the new padstack.
- by replacing the pads on an instance basis. In this case, you can replace only the pads that require the new padstack.

Replacing Pads with Imported Padstack

1. Select the *r23_254m28_259r6* padstack from the Explorer - Footprint - Padstacks list.
2. Right-click the selection, choose Replace With and then select *b130_330h70_270m135_335tr110_310_140_340_26yb140_340_sq*.
Both pads of the footprint change to the new padstack as shown in the following figure:



3. Right-click anywhere on the canvas and select Undo Replace Padstack.
Next, you will update only one pad in a footprint.
4. Select the NPTH - Slot from the Explorer - Features - Mounts list.



5. Right-click the selection and choose *Set Padstack -*

b130_330h70_270m135_335tr110_310_140_340_26yb140_340_sq

The result is that only one pad is updated to the imported padstack. This may be beneficial if your company standards require a special pad for a certain situation.

Now, you can export to this footprint to Allegro PCB Editor and view the footprint in Allegro STEP Mapper as well as the 3D Viewer.

Creating Footprint of a Mechanical Part

What You Will Learn

In this module, you will learn how a footprint can be created from a detailed 3D STEP model of a non-standard mechanical component to generate a mechanical symbol in Allegro PCB Editor.

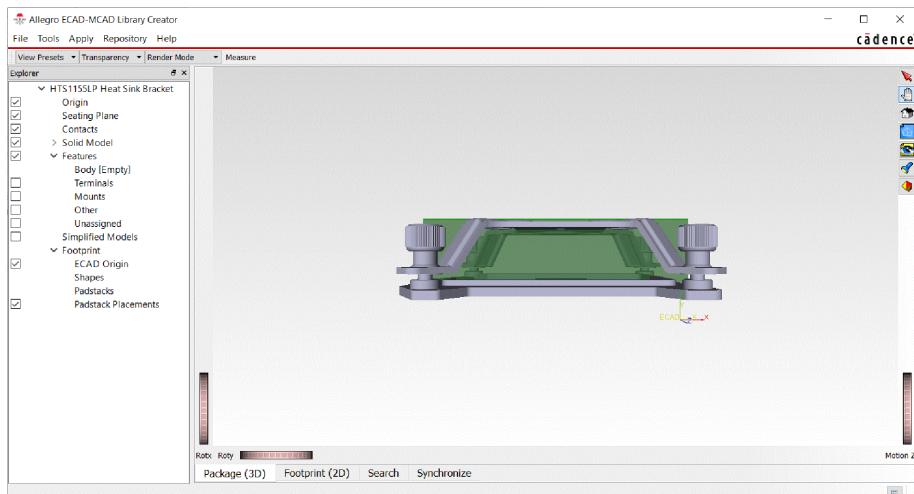
Also, you will learn to create a single mechanical symbol that has both top and bottom PLACEBOUND regions that will define the constraints to allow the PCB design to work around and over other components.

Importing a Step Model

In this section, you will import the step model *HTS1155LP Heat Sink Bracket.stp* from <your_install_dir>/doc/lc_tut/tutorial_examples/STEP_models.

For more information about importing a STEP Model, refer to the [Importing STEP Models](#) section.

The *HTS1155LP Heat Sink Bracket* is displayed in *Explorer* as well as on the canvas.



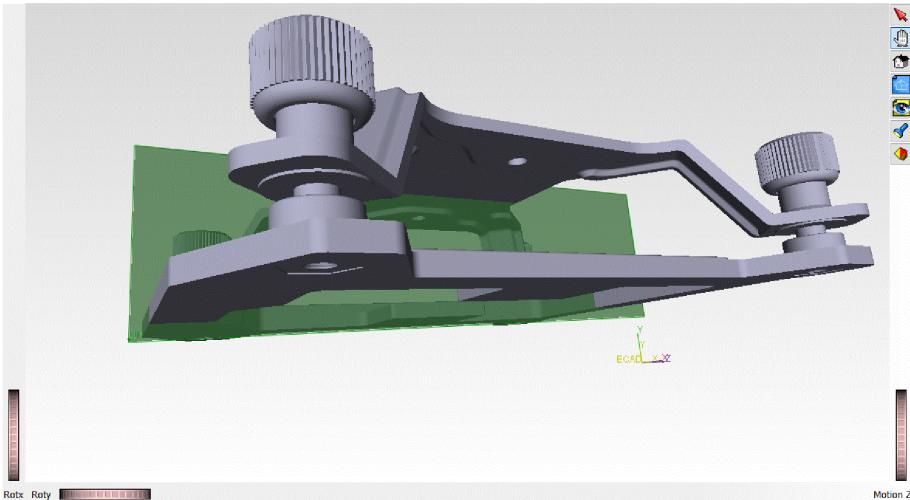
1. Choose *File - Import - STEP* to load the provided model of a heat sink *HTS1155LP Heat Sink*

Bracket.stp

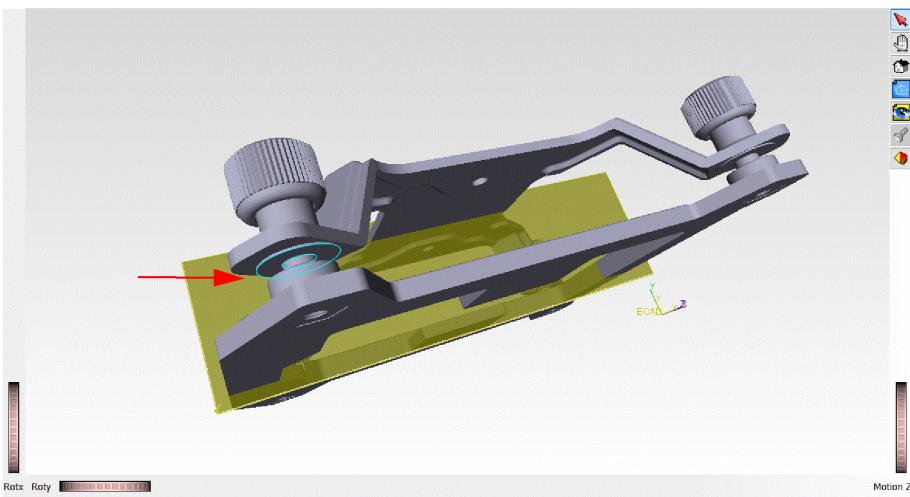
This component has both top and bottom hardware that will sandwich the PCB.

Editing Seating Pane

1. Position the 3D viewing area as shown below.



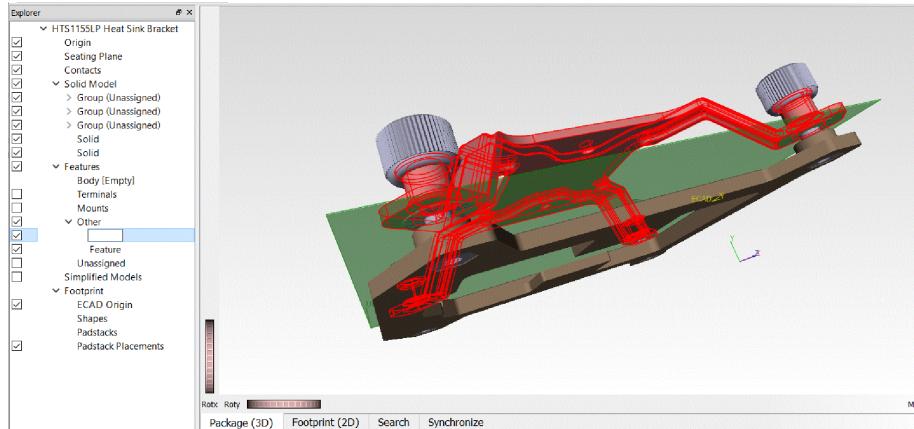
2. Right-click Seating Plane and choose Edit in Explorer.
The color of the seating plane changes signifying that it is now in edit mode.
3. Click the Align To Face button on the toolbar and then click the red selection arrow toolbar icon on the right side.
4. Move the cursor and select the face as displayed in the following figure.



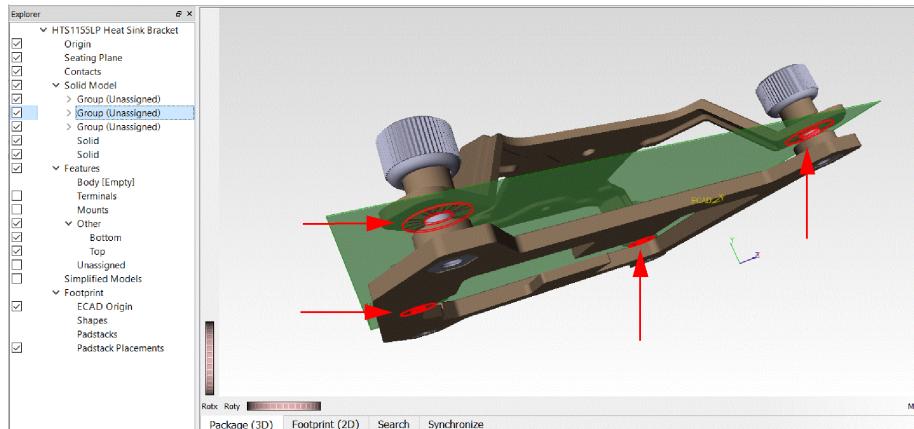
5. Click the Accept button.
The seating plane is now positioned so that the bottom of the top hardware is located face to face with the top face of the PCB.

Renaming Features

1. In the *Explorer* pane, select and drag both *Solid (Unassigned)* nodes under *Solid Model* and drop to *Features - Other* node.
Two new items called *Feature* are listed under *Features - Other* node in *Explorer*.
2. Identify the top bracket and double-click to change the name.
The name becomes editable.



3. Type *Top* and press *Enter*.
4. Similarly, rename the other feature as *Bottom*.
5. Click each of the *Solid Model - Group [Definition]* nodes to identify the mounting holes in *Explorer*.



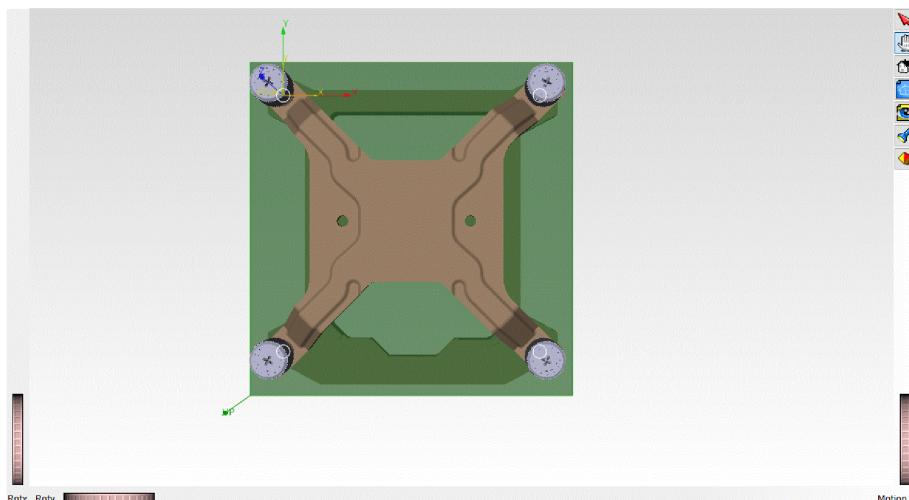
6. Expand the identified *Group*.
7. Select all the four *Solid (Unassigned)* nodes.
8. Right-click the selection and choose *Contact Features - Training/Hole Features*.
Four entries are created under *Features - Unassigned - Group*.

9. Select and drag the *Group* to the *Features - Mounts* category.
Four Mount nodes are created under *Features - Mounts*.
10. Select the four mounts, right-click and choose *Set Type – PTH-Drill*.
Four holes displayed in the Footprint (2D) view.

Editing the Origin

1. Right-click Footprint - ECAD Origin in Explorer and choose Change ECAD Origin.
This helps to better align a part when using in a design. The cursor changes to cross hair.
2. Zoom in to the top left hole.
3. Move the cross hair over to the top left hole and click to select that hole as the new origin.
The X and Y cross hair is on the top left hole.
Ensure the *Sync 3D Origin* check-box is selected.
4. Click *Accept*.

5. In the *Package (3D)* view, click .

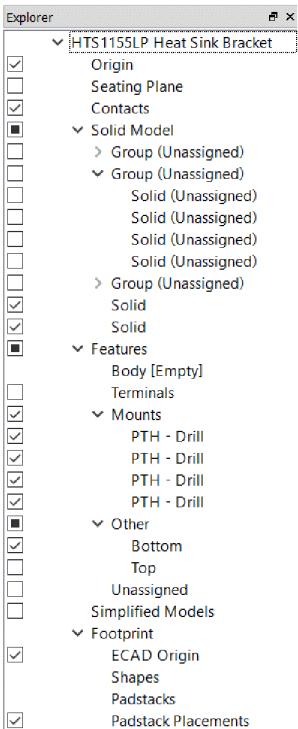


To ensure that the bottom PLACEBOUND is set correctly, you need to set the correct PCB thickness for this part. You can use the measure tool to calculate the distance between the bottom and top parts to determine the maximum PCB thickness.

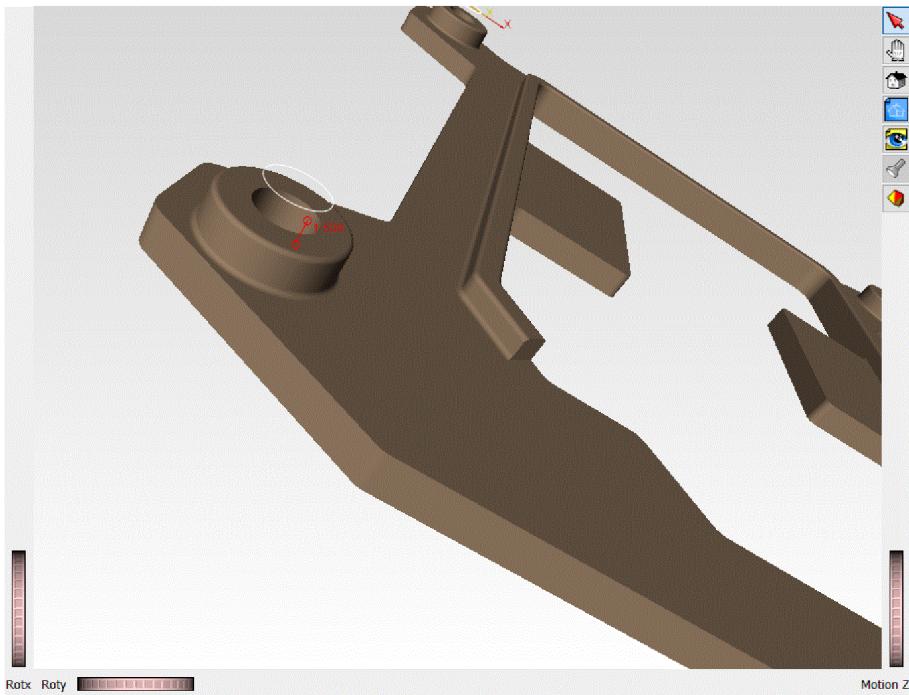
Measuring Thickness

Before we can measure let's put the canvas in a state that will make it easier to achieve a measurement.

1. Select the check boxes in the *Explorer* pane as displayed in the following figure.



2. Select the *Measure* toolbar icon across the top.
Ensure that the *Distance Mode* is set to *Height*.
3. Click the red *Selection* arrow at the top of the right-side toolbar.
Now you can select any surface to measure its height.
4. Click on the upper face of any of the four elevated mounting platforms.
The height of the elevated mounting platforms is displayed as **1.500**. This means that **1.500** is the assumed PCB thickness for this heatsink bracket.



5. Click the Accept button.

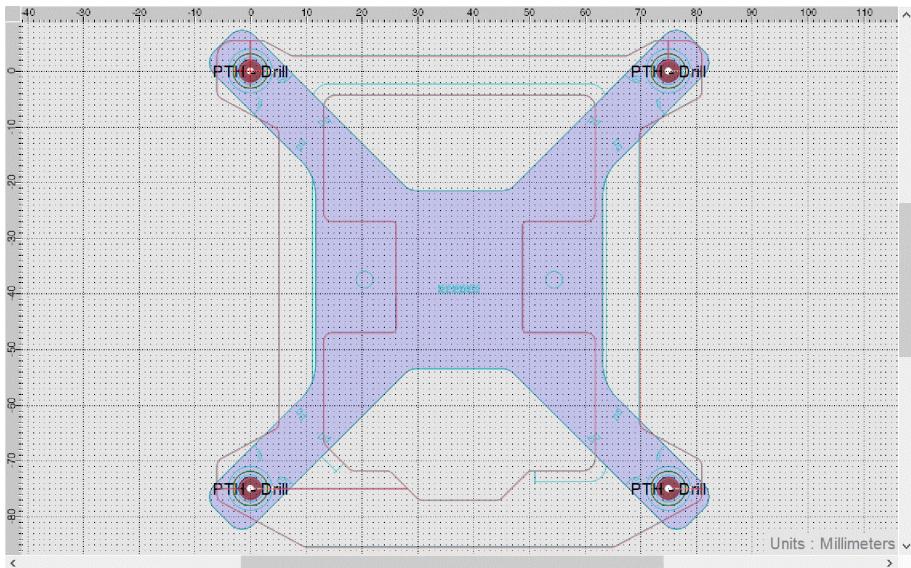
Now, you will set the PCB thickness as well as the overall height of the part.

6. Right-click *HTS1155LP Heat Sink Bracket* and select Parameters - Edit in *Explorer*.
7. In the *Editing* dialog, select the *Height* check box.
8. Set the value of Height to 16.35 and click *Compute*.
9. Select the *PCB Thickness* check box and set its value to 1.500.
10. Click *OK*.
11. In the *Explorer* pane, select all the check boxes.

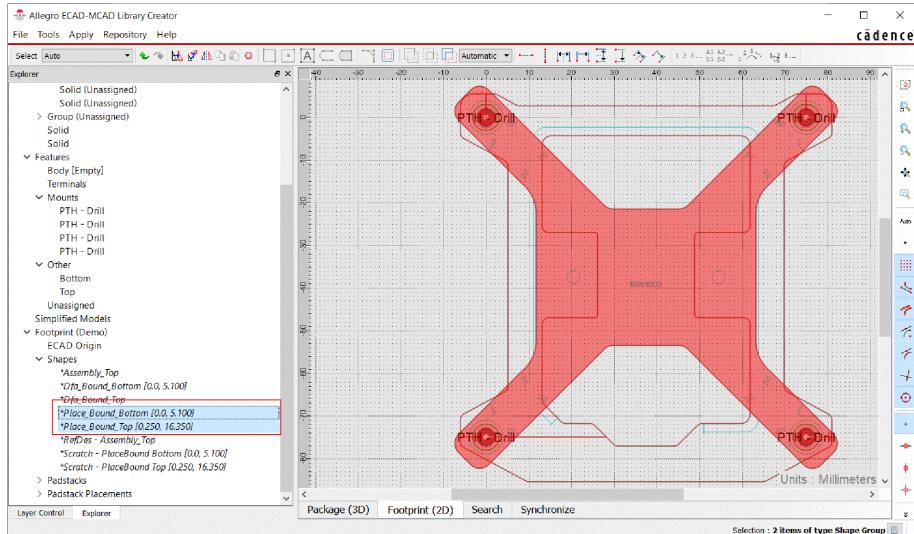
Applying Rules

In this section, you will apply the *Apply - Training/Multiple Heights* rule to create the footprint.

For more information about applying rules, refer to the [Applying Rules to Template](#) section. The footprint matching the *Apply - Training/Multiple Heights* rule is created.



In *Explorer*, you can see that the PLACEBOUND shapes are added along with all the other automatically created footprint features. Selecting the object in the Explorer pane causes the same object to be selected in the Footprint (2D) view.



Next, you will upload this featured version of your package to the repository.

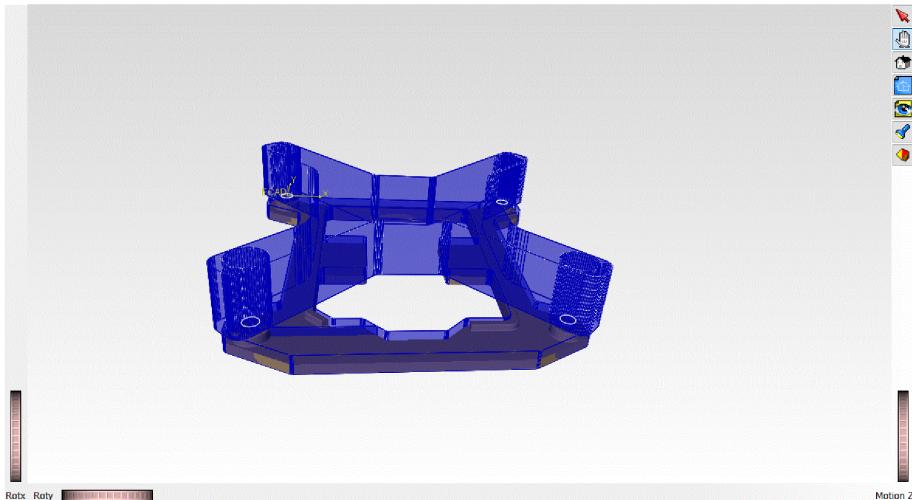
Creating 3D Placebound Shapes

In this section, you will create the 3D PLACEBOUND shapes by applying the *Training/Extrude – PlaceBound [Simplified Model]* rule.

For more information about adding the *Training/Extrude – PlaceBound [Simplified Model]* to the *Apply* menu, refer to the [Adding Rules to Apply Menu](#) section.

1. Choose *Apply - Training/Extrude – PlaceBound [Simplified Model]*.

The PLACEBOUND shapes are now part of the 3D model.



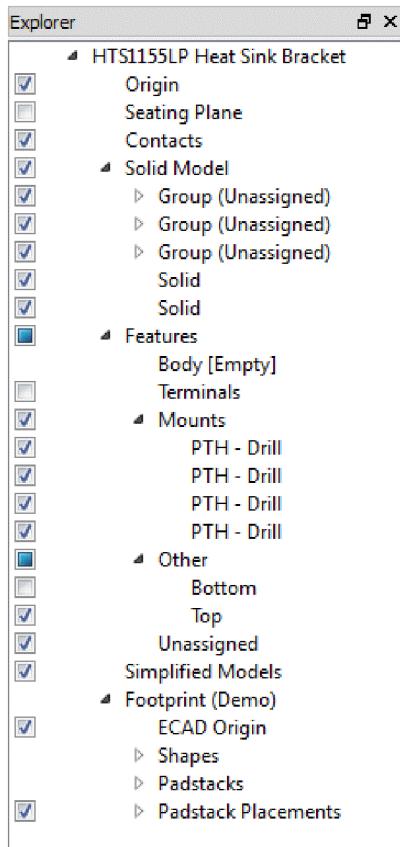
These solid PLACEBOUND shapes make it impractical to use this component in a real application as the PLACEBOUND shapes will create errors if components such as the main IC are placed under it.

2. Right-click *Simplified Model - PlaceBound* choose *Delete*.

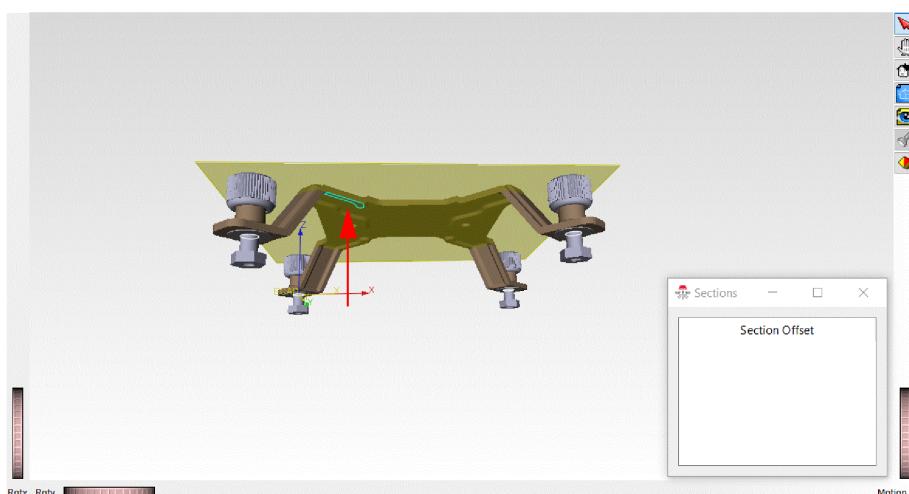
You will now add section offsets to the upper bracket that will aid us in creating proper PLACEBOUND regions.

Adding Section Offsets

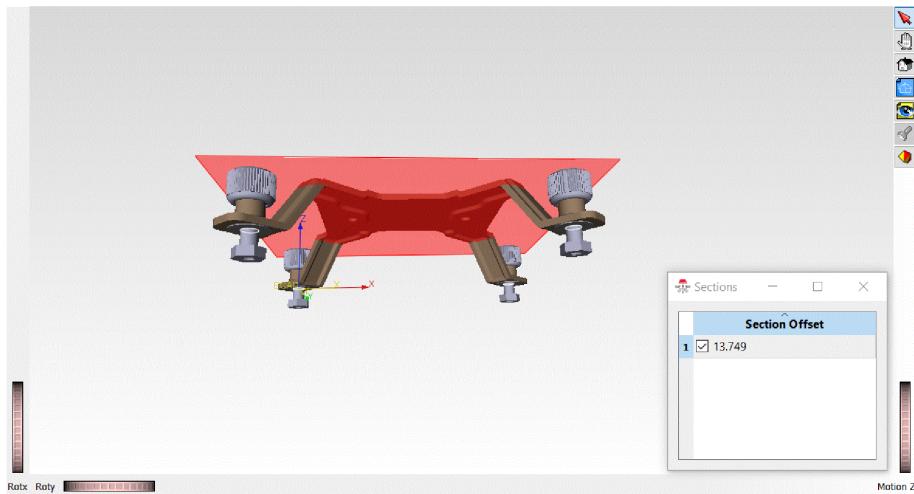
1. Select the check boxes in the *Explorer* pane as displayed in the following figure.



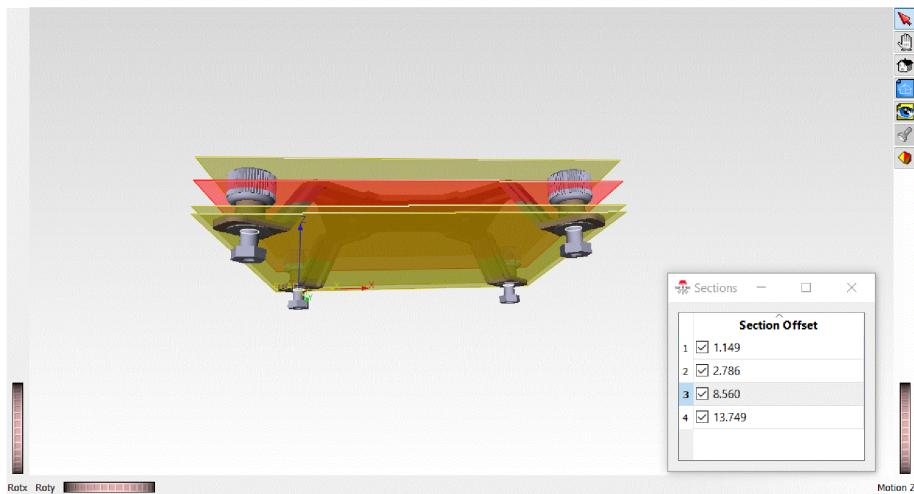
2. Right-click *Features - Other - Top* and choose *Edit Sections*.
3. Click the red selection arrow toolbar icon on the right side.
4. Click one of the faces on the underside as shown in the following figure:



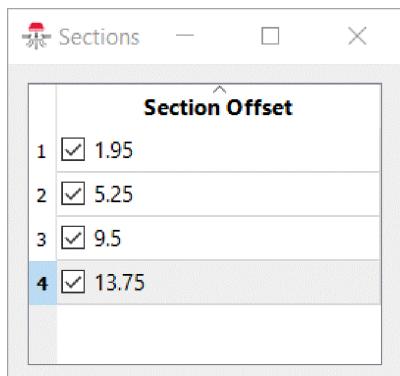
A colored section plane on the canvas as well as an entry in the *Section* dialog is added.



5. Move the cursor down the angled section of the bracket and click to add three more sections as shown in the following figure.



6. Double-click on each entry in the *Sections* dialog and change the original values to match as shown in the following figure:



Setting these values moves the recently added sections at the specified heights off the

seating plane which gives better control on the stair-step effect created by the Place_Bound shapes.

7. Click *Accept* on the toolbar.
8. Choose *Apply - Training/Multiple Heights*.
Top Placebound regions in the *Explorer* panel are added.

Saving the Footprint

It is important that you save a new version of the footprint before exporting it to PCB Editor.

For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

Exporting Footprint to Allegro PCB Editor

For more information about exporting the footprint to Allegro PCB Editor, follow the steps mentioned in the [Exporting the Footprint to Allegro PCB Editor](#) section.

Adding STEP Model to Existing Allegro PCB Editor Footprint

What You Will Learn

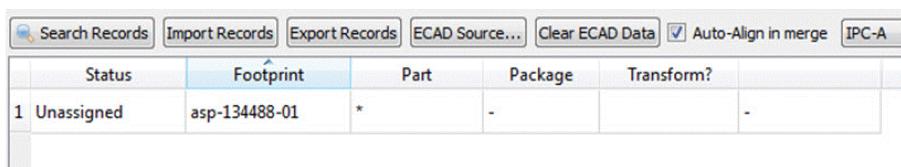
In this module, you will learn to import an existing, previously created 2D centric footprint for a complex component and make it 3D centric by using Library Creator.

For this module, you will use the footprint *asp-134488-01.dra* and the STEP model *asp-134488-01.stp*.

Importing Footprints

In this section, you will import the *asp-134488-01.dra* footprint from
`<your_install_dir>/doc/lc_tut/tutorial_examples/Synchronization_Examples`.

For more information about importing a footprint, follow the steps mentioned in the [Importing Footprints](#) section.

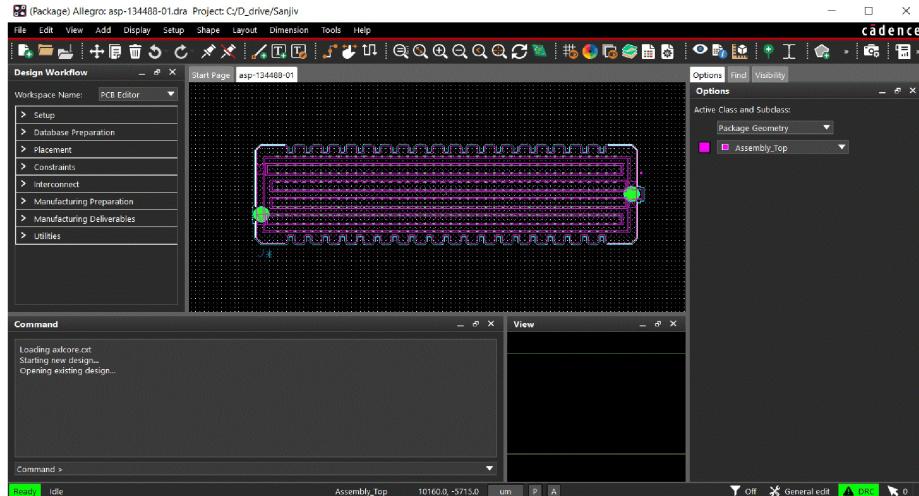


The screenshot shows the 'Import Records' dialog in the Allegro X software. The top bar includes buttons for 'Search Records', 'Import Records', 'Export Records', 'ECAD Source...', 'Clear ECAD Data', a checked checkbox for 'Auto-Align in merge', and 'IPC-A'. The main area is a table with columns: Status, Footprint, Part, Package, and Transform?. A single row is present, showing 'Unassigned' in the Status column, 'asp-134488-01' in the Footprint column, and asterisks (*) in the other columns.

Status	Footprint	Part	Package	Transform?
1 Unassigned	asp-134488-01	*	-	-

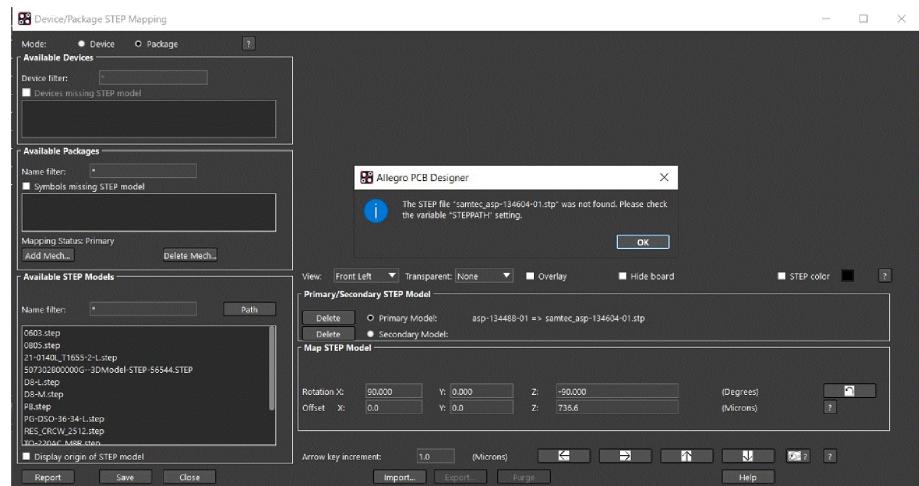
Checking STEP Model

1. Open the footprint in Allegro PCB Editor.



2. In Allegro PCB Editor, choose *Setup - STEP Package Mapping*.

An error message is displayed stating that there is no STEP model assigned. This confirms that this footprint is 2D centric and does not have a 3D model assigned to it.

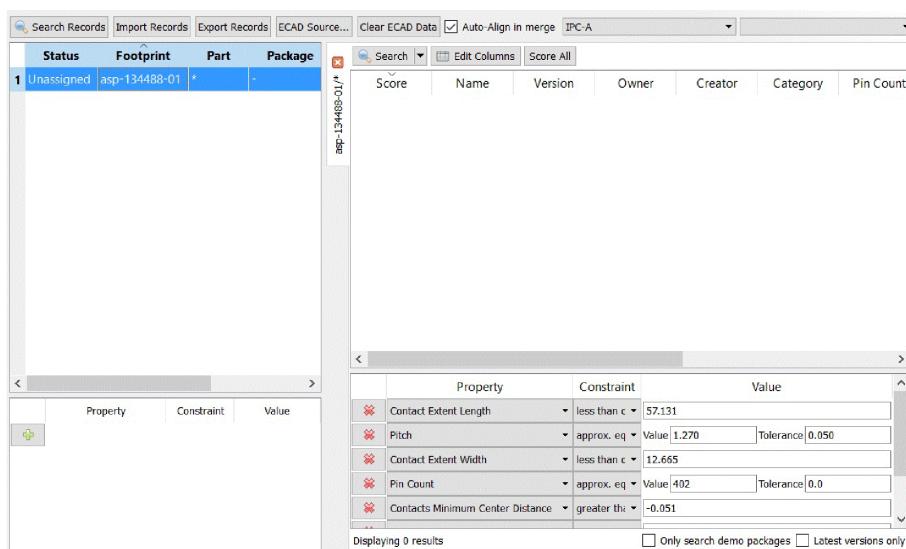


3. Close Allegro PCB Editor.

Searching Packages

1. In Library Creator, right-click *asp-134488-01* and choose *Auto Search*.

A search criteria panel with search constraints appears.



2. Click the *Search* button.

The search in the Library Creator repository turns up zero matching results.

	Property	Constraint	Value
✖	Contact Extent Length	less than c	57.131
✖	Pitch	approx. eq	Value: 1.270 Tolerance: 0.050
✖	Contact Extent Width	less than c	12.665
✖	Pin Count	approx. eq	Value: 402 Tolerance: 0.0
✖	Contacts Minimum Center Distance	greater than	-0.051

Displaying 0 results Only search demo packages Latest versions only

If no matching packages are found, you can use the Library Creator parametric templates to create the required package. In case of complex components, such as connectors, the 3D model can be obtained from the vendor, imported into Library Creator, turned into a package, saved into the repository and then matched up with the footprint.

3. Close the search panel.

Importing STEP Models

In this section, you will import the STEP Model *asp_134488-01.stp*.

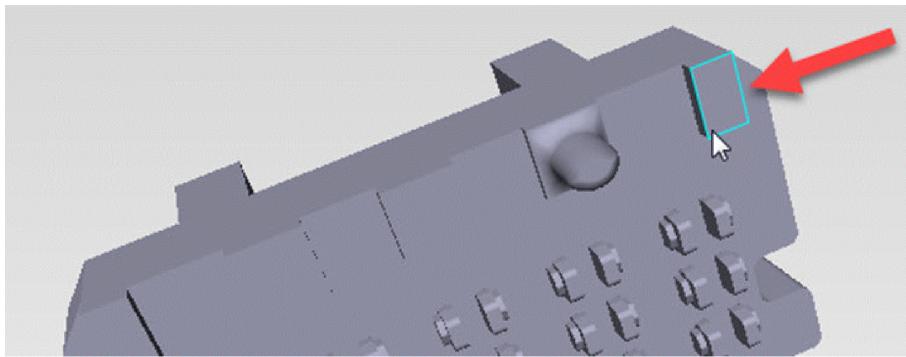
For more information about importing a STEP Model, follow the steps mentioned in the [Importing STEP Models](#) section.

After the import is complete, notice that the green *Seating Plane* is not located on the correct face of the connector.

Next we will change the seating pane to the correct face of the connector.

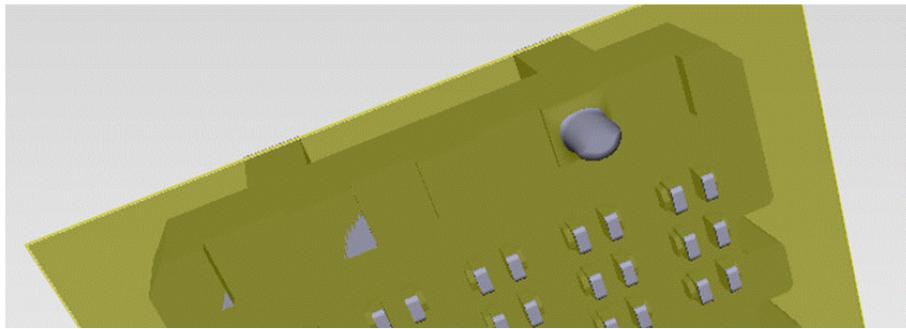
Editing Seating Pane

In this section, you will edit and move the seating pane to the connector highlighted in the red in the following figure.



For more information editing a seating pane, follow the steps mentioned in the [Editing Seating Pane](#) section.

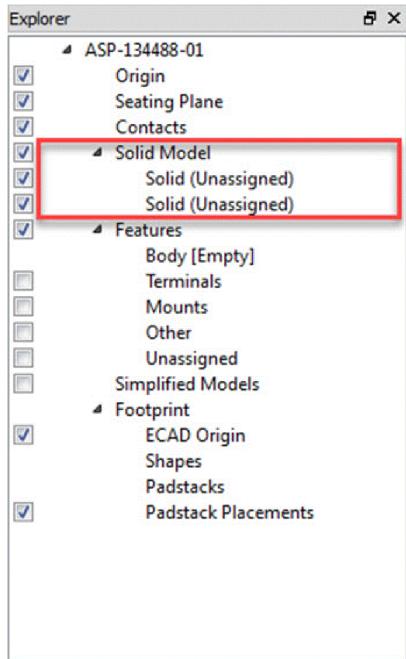
The seating pane is repositioned as displayed in the following figure.



Adding Contact Features

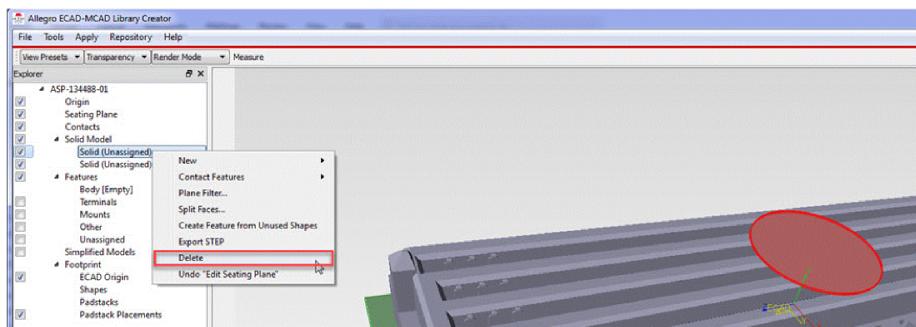
In this section, you will remove the circular object at the center of the connector and then assign contact features.

1. In the *Explorer* pane, expand *Solid Model*.



The imported STEP model has two entities. The first *Solid (Unassigned)* entity reflects the circular object in the center of the connector. This flat circular object is a disposable object used by the assembly line pick and place machines that allows a vacuum suction to grab the part at its centroid location and then place the connector in its proper location on the board. This circular object can be removed as it does not add value to the connector.

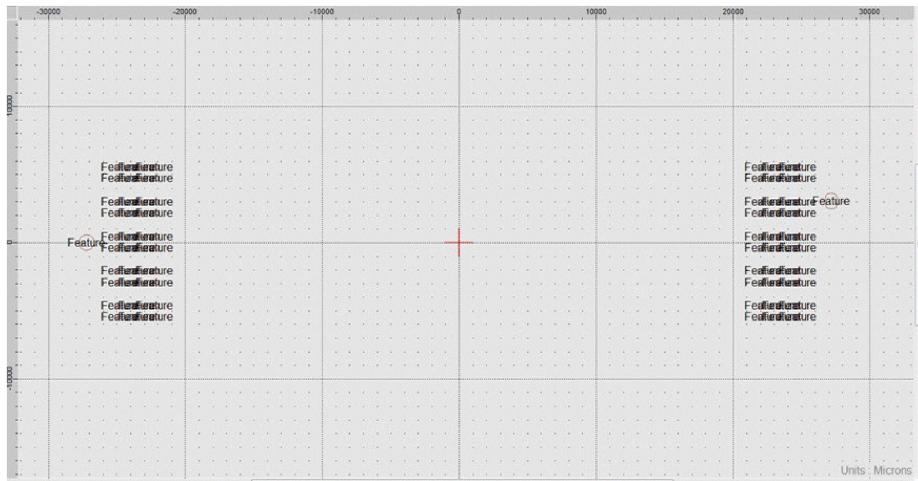
2. Right-click *Solid Model - Solid (Unassigned)* corresponding to the circular part and choose *Delete*.



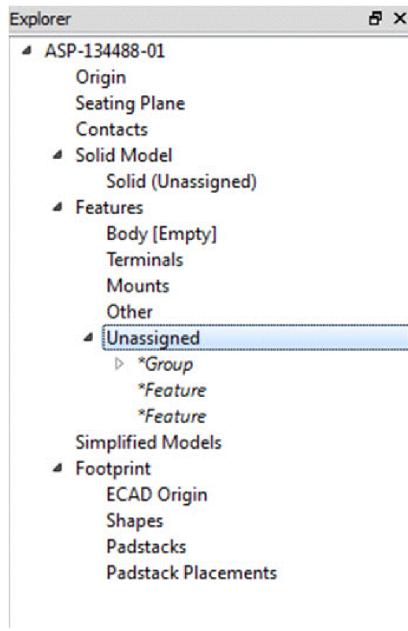
Next, you will add *Training/THT Features* contact feature to *Solid (Unassigned)* under *Solid Model*.

For more information about assigning contact features, refer to the steps mentioned in the [Adding Contact Features](#) section.

3. Right-click *Solid (Unassigned)* and choose *Contact Features - Training/THT Features*. The end contacts and the two mounting holes are added.



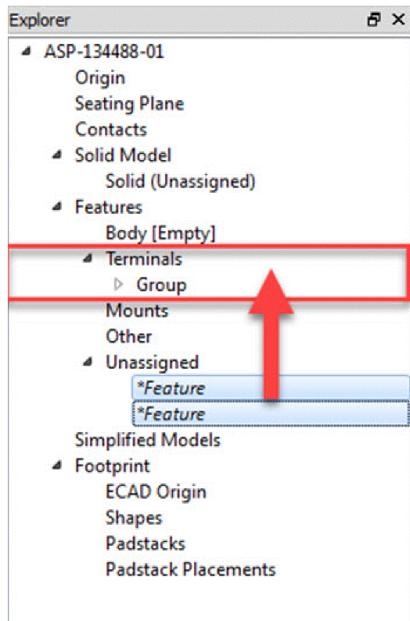
In the *Explorer* pane, the just added THT Features are automatically added under *Features - Unassigned*.



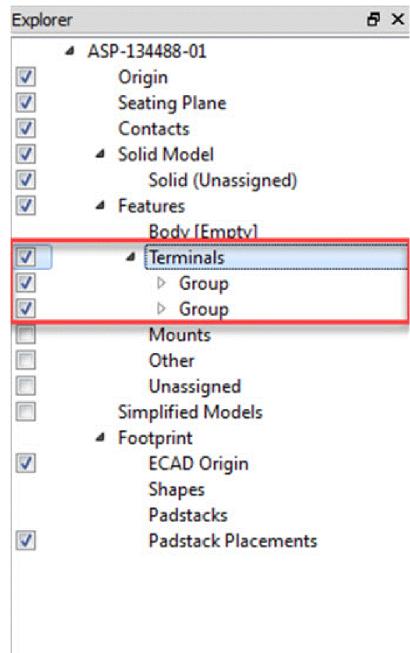
Library Creator intelligently creates a group of end pins that are of the same size and the two mounting holes are referenced as *Feature*.

4. Select and drag Feature - Unassigned - Group in *Explorer* and drop it to Feature - Terminals. Usually, the mounting holes are part of the schematic and are connected to a signal when used in a circuit.
Next, you will move the contact points for these two mounting holes to the *Terminals* node because Mounts is the terminology used by Library Creator for mechanical pins while Terminals corresponds to signal pins.
5. Select and drag both *Unassigned - Feature* items and drop them to the *Features - Terminals* -

Groups node.



Once complete, the Terminals node now includes two Groups – one for the pins and one for the two mounting holes.



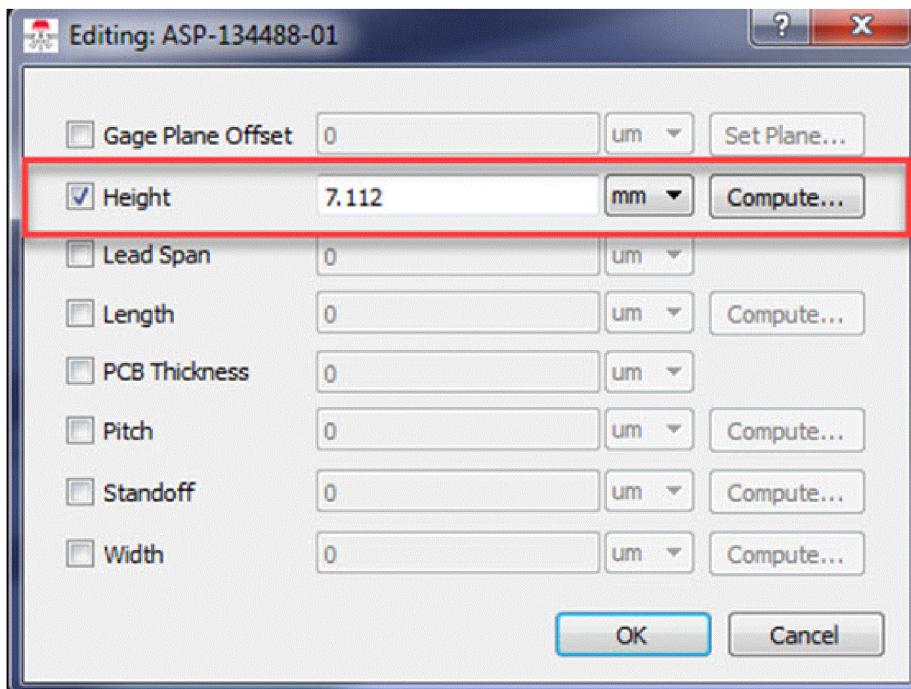
6. Right-click the *Group* that corresponds to end pins and choose *Set Lead Form - Other Surface*.
7. Right-click the *Group* that corresponds to the mounting holes and choose *Set Lead Form - Through-Round*.

Assigning Terminal Types

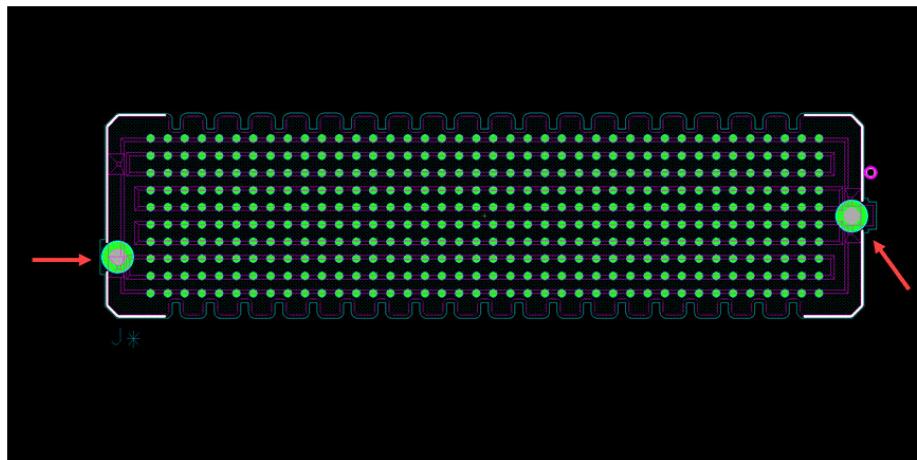
1. Click the *Package (3D)* view tab.
2. Select and drag the *Solid Model - Solid* node and drop it to the *Features - Body [Empty]* node. The *Features - Body [Empty]* node changes to *Feature - Body*. This tells Library Creator that the complete 3D shape is used to calculate the placement keepout.

Assigning Model Height

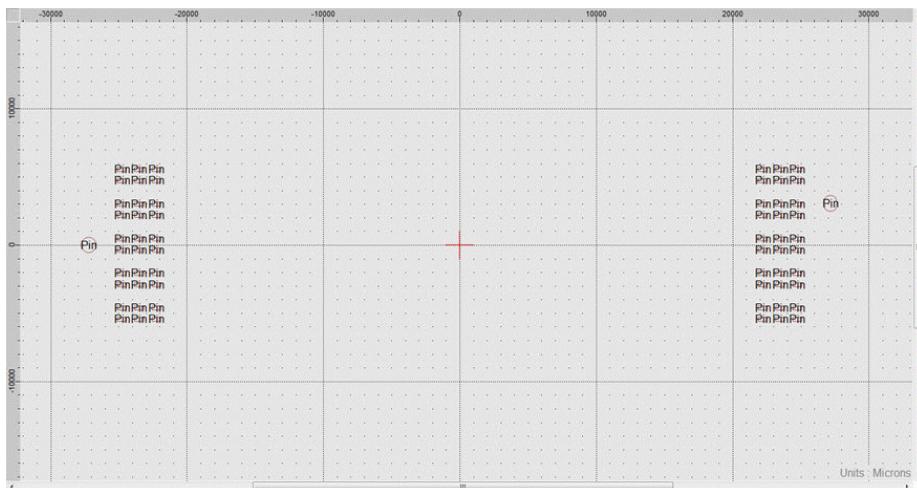
1. Right-click *ASP_134488-01* and select *Parameters - Edit* to assign the model height parameter.
2. In the *Parameter Editor* dialog, select the *Height* check box choose *Compute*. Library Creator examines the STEP model and calculates the height automatically.



3. Click *OK*.
In the footprint loaded in Allegro PCB Editor, notice that the lower left offset mounting hole is labeled as *ST2* and the one on the right is labeled as *ST1* as displayed in the following figure.

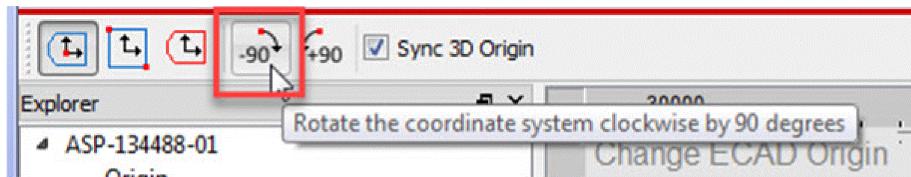


In the Library Creator 2D view, you can see that the connector orientation is 180 degrees different from the footprint loaded in Allegro PCB Editor.



Editing the Origin

1. In the *Explorer* pane, right-click *Footprint - ECAD Origin* and choose *Change ECAD Origin*.
2. Select *Sync 3D Origin* check box.
3. Double-click *Rotate Clockwise* to rotate the package as well as the STEP model.



4. Click *Accept*.

The two mounting holes are now in the same locations as those on the original footprint.

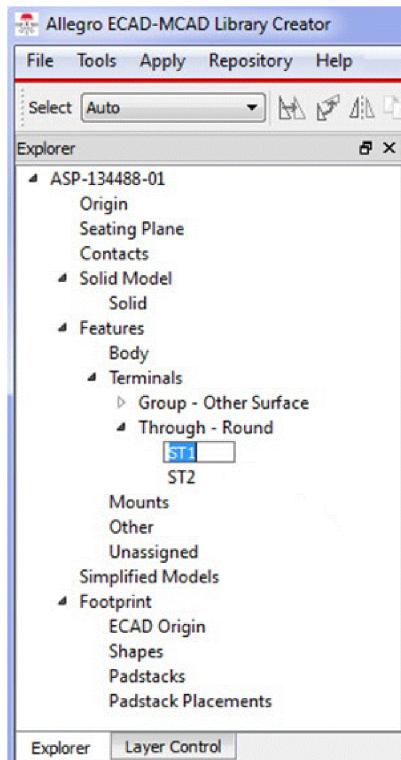


5. Click  to fit the footprint on the canvas.

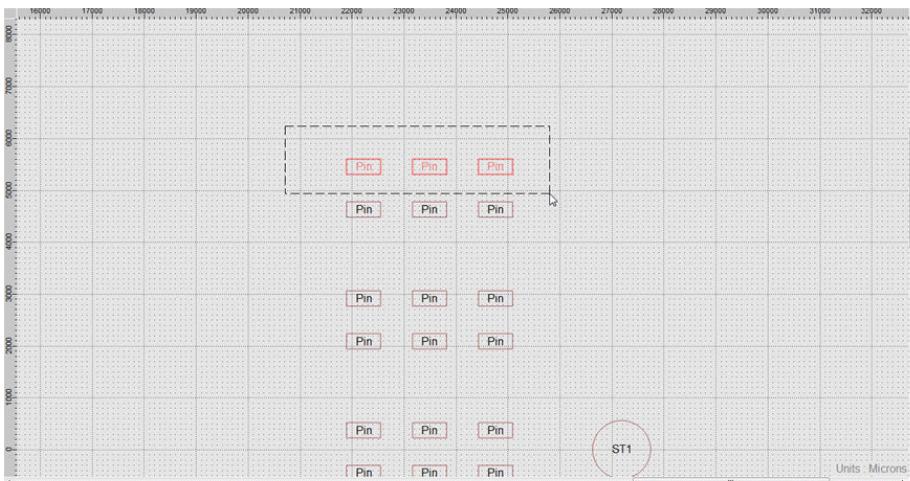
Assigning Pin Numbers

1. In the *Explorer* pane, expand *Features - Terminals - Through – Round*.
2. Double-click the pin of the right-side to rename it to ST1.
3. Double-click the pin of the left-side to rename it to ST2.

Allegro X ECAD-MCAD Library Creator Tutorial
Adding STEP Model to Existing Allegro PCB Editor Footprint--Editing the Origin



4. Select the contact areas on the top-right.

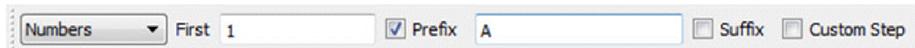


5. In the *Explorer* pane, expand *Features - Terminals - Group – Other Surface*.



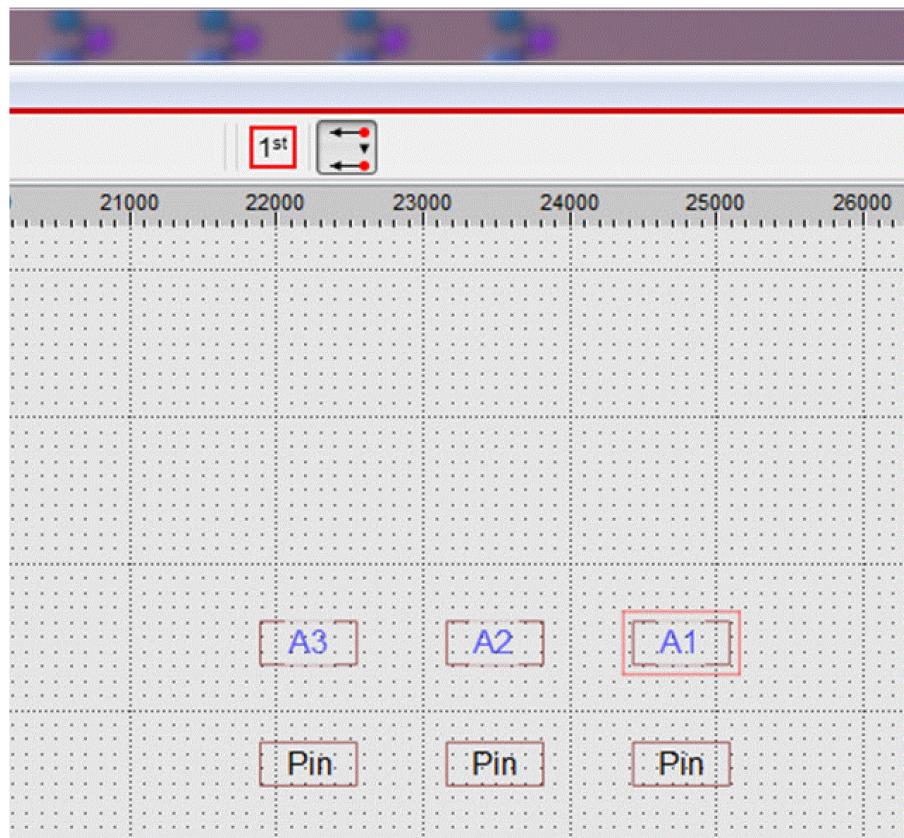
6. Click on the toolbar.

7. Select the *Prefix* check box and type ^A.



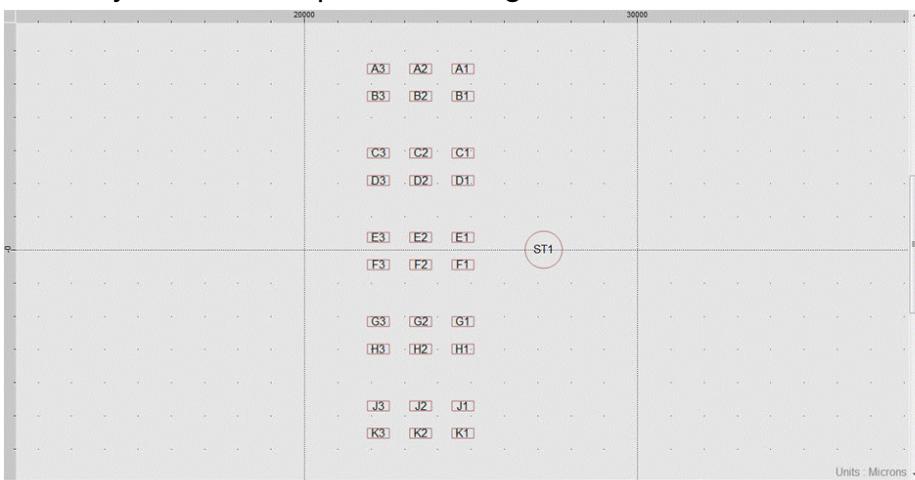
The top right pin should be pin A1.

8. If the top-right pin is numbered A3, click **1st** and select the top-right pin.



9. Click *Accept*.

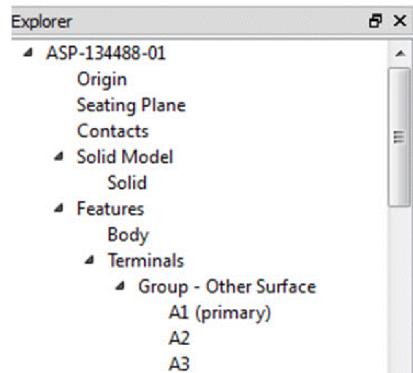
10. Similarly, number the pins on the right-side as shown in the following figure.



11. Similarly, number the pins on the right-side as shown in the following figure.



The pin numbers in the *Explorer* view indicate *Pin A1* as the primary pin.



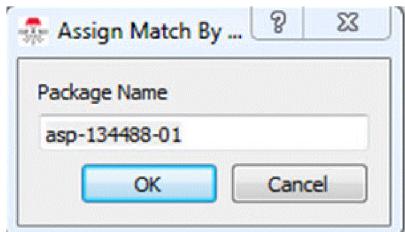
Saving the Footprint

It is important that you save a new version of the footprint before exporting it to PCB Editor.

For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

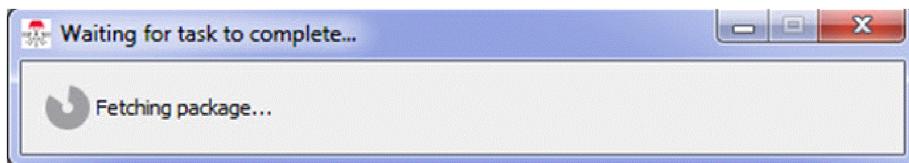
Merging Package with Footprint

1. Click the *Synchronize* tab.
2. Right-click the STEP Model *asp_134488-01* and choose *Match By Name*.
Assign Match By Name dialog box is displayed.



3. Click *OK*.

Waiting for the tasks to complete dialog is displayed.



4. Click the *Accept and Save* button in the *Match Merge* dialog.

The synchronization record is updated to indicate that the footprint and the package are now linked.

IPC-A					
Status	Footprint	Part	Package	Transform?	
1 Up to Date	asp-134488-01	*	ASP-134488-01	Y	-

5. Right-click *asp_134488-01* and choose *Push STEP to Allegro*.

A confirmation message is displayed.

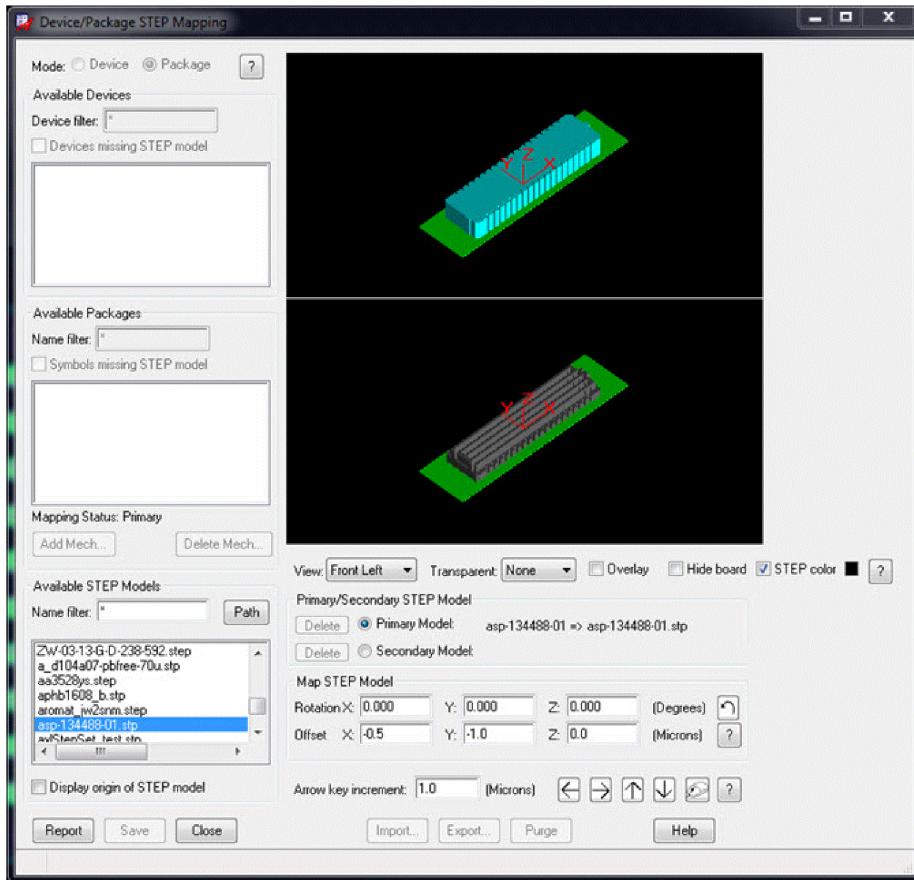
6. Click *Yes*.

The *STEP* model is pushed and mapped to the original footprint and the synchronize record is update to indicate an association between footprint, Package and *STEP* model.

IPC-A					
Status	Footprint	Part	Package	Transform?	Allegro/Step
1 Up to Date	asp-134488-01	*	ASP-134488-01	Y	asp-134488-01

To confirm that the original 2D centric footprint now has a 3D model associated with it, open the footprint in Allegro PCB Editor as mentioned in the [Checking STEP Model](#) section.

Allegro X ECAD-MCAD Library Creator Tutorial
Adding STEP Model to Existing Allegro PCB Editor Footprint--Saving the Footprint



Synchronizing Libraries

What You Will Learn

In this module, you will learn how existing footprints can be synchronized with the provided 3D packages in the library repository. Library synchronization is useful for

- finding and attaching a detailed 3D STEP model to an Allegro PCB Editor footprint or
- checking an existing Allegro PCB Editor footprint against a physical package model and a particular set of design rules or design technology.

Synchronizing Existing Footprints with Packages

In this section, you will synchronize existing footprints with package in the repository by importing a footprint, comparing packages with the imported footprint, and merging the package with the footprint.

- [Importing Footprints](#)
- [Searching Packages](#)
- [Comparing Packages and Footprint](#)
- [Merging Package with Footprint](#)

Importing Footprints

In this section, you will import all the four the footprints from

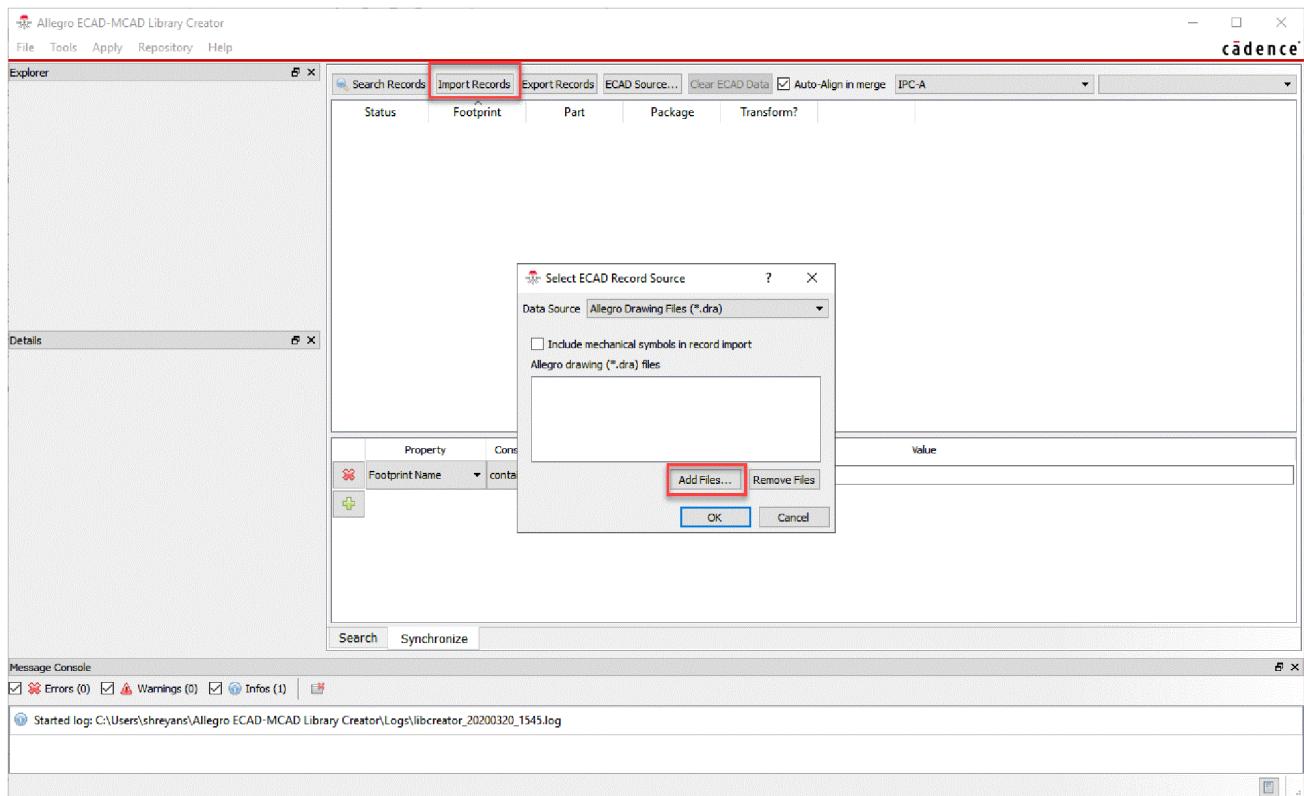
`<your_install_dir>/doc/lc_tut/tutorial_examples/Synchronization_Examples.`

1. Select the Synchronize tab at the bottom of the Library Creator window.
2. Click the Import Records button.

Select ECAD Record Source dialog is displayed.

Allegro X ECAD-MCAD Library Creator Tutorial

Synchronizing Libraries--What You Will Learn



3. Click the *Add Files* button and browse to the following directory:

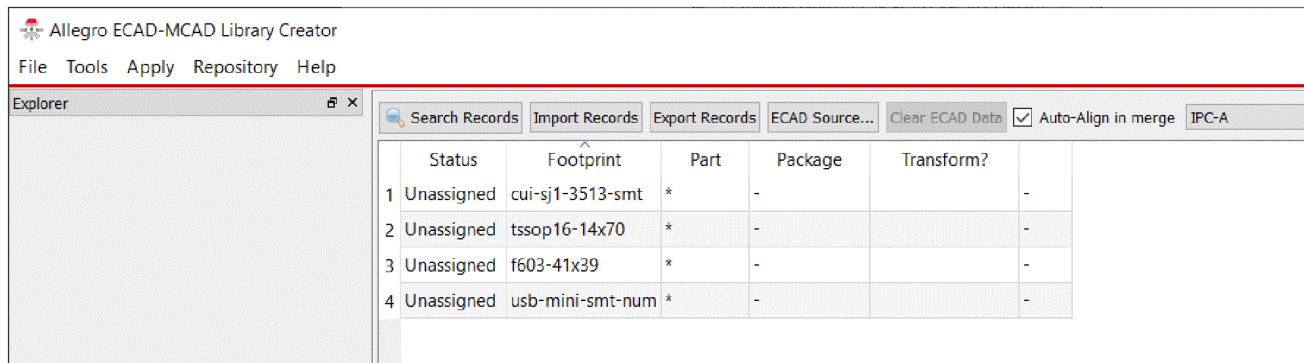
<your_install_dir>/doc/lc_tut/tutorial_examples/Synchronization_Examples.

4. Select all the .dra files in the folder and click *Open*.

The files are added to the *Select ECAD Record Source* dialog.

5. Click *OK*.

The footprints are added to Library Creator.



6. Right-click *tssop16-14x70* and choose *View Footprint* to test the communication between Library Creator and Allegro PCB Editor.

After a successful import of the footprint, it is displayed in the 2D and 3D views of Library

Creator. Successful import of the footprint into Library Creator also means that the communications lines between Allegro PCB Editor and Library Creator are working and that the paths are also set correctly.

7. Choose *File - Close Active*.

Searching Packages

1. Select the *IPC-B* from the drop-down list next to the *Auto-Align in merge* check box.
2. Right-click *tssop16-14x70* and choose *Auto Search*.

A search criteria panel with search constraints appears. The search constraints shown are automatically created by Library Creator based on the objects of the Allegro PCB Editor footprint loaded in.

The screenshot shows the Library Creator search interface. At the top, there's a toolbar with buttons for Search Records, Import Records, Export Records, ECAD Source..., Clear ECAD Data, and Auto-Align in merge (which is checked). A dropdown menu shows 'IPC-B'. Below the toolbar is a table of search results:

Status	Footprint	Part	Package	Time
1 Unassigned	cui-sj1-3513-smt	*	-	
2 Unassigned	tssop16-14x70	*	-	
3 Unassigned	usb-mini-smt-num	*	-	
4 Unassigned	f603-41x39	*	-	

To the right of the table is a search criteria panel with tabs for Search, Edit Columns, and Score All. The 'Search' tab is selected. It shows the search term 'tssop16-14x70/*' and a list of search constraints:

Property	Constraint	Value	Tolerance
Pitch	approx. eq	Value 0.650	Tolerance 0.050
Contact Extent Width	less than c	5.507	
Pin Count	approx. eq	Value 16	Tolerance 0.0
Contacts Minimum Center Distance	greater thi	1.603	
Mount Form	equals	Surface	

At the bottom of the search criteria panel, there are checkboxes for 'Only search demo packages' (unchecked) and 'Latest versions only' (checked).

3. Ensure that the *Only search demo packages* check box is deselected to ensure that the entire repository is searched.
4. Click the *Search* button.

Matching search results are displayed.

You can also refine the search by adding more search parameters in the lower section of the search criteria panel.

Comparing Packages and Footprint

1. Click the *Score All* button to score all the results.

Library Creator compares the packages against the footprint (*tssop16-14x70*) and scores the packages. Depending on the number of matching results, the scoring may take some time.

2. Click the *Score* column to sort the scores from highest to lowest.

The screenshot shows the Allegro X Library Creator interface. At the top, there are menu options: Search Records, Import Records, Export Records, ECAD Source..., Clear ECAD Data, Auto-Align in merge, and IPC-B. Below the menu is a toolbar with icons for search, edit columns, and score all. The main area has two panes. The left pane displays a table of packages with columns: Status, Footprint, Part, Package, and Total Height. The right pane shows a detailed view of the footprint 'tssop16-14x70' with a table of matching packages. The packages listed are: MO-153_AB, MO-152_AB, MO-137_AB, MO-150_AC, TI_RSA, and NSC_SQB1... Each package has a score, name, version, owner, creator, category, and pin count. Below the footprint view is a table of constraints with columns: Property, Constraint, Value, and Tolerance. The constraints listed are: Pitch (approx. eq, 0.650, 0.050), Contact Extent Width (less than c, 5.507), Pin Count (approx. eq, 16, 0.0), Contacts Minimum Center Distance (greater than, 1.603), and Mount Form (equals, Surface). At the bottom of the interface, there are checkboxes for 'Only search demo packages' and 'Latest versions only'.

Status	Footprint	Part	Package	Total Height
1 Unassigned	cui-sj1-3513-smt	*	-	
2 Unassigned	tssop16-14x70	*	-	
3 Unassigned	usb-mini-smt-num	*	-	
4 Unassigned	f603-41x39	*	-	

Score	Name	Version	Owner	Creator	Category	Pin Count
1	MO-153_AB	3	System	System	SOP	16
2	MO-152_AB	4	System	System	SOP	16
3	MO-137_AB	4	System	System	SOP	16
4	MO-150_AC	5	System	System	SOP	16
5	TI_RSA	2	System	System	QFN	16
6	NSC_SQB1...	2	System	System	QFN	16

Property	Constraint	Value	Tolerance
Pitch	approx. eq	0.650	0.050
Contact Extent Width	less than c	5.507	
Pin Count	approx. eq	16	0.0
Contacts Minimum Center Distance	greater than	1.603	
Mount Form	equals	Surface	

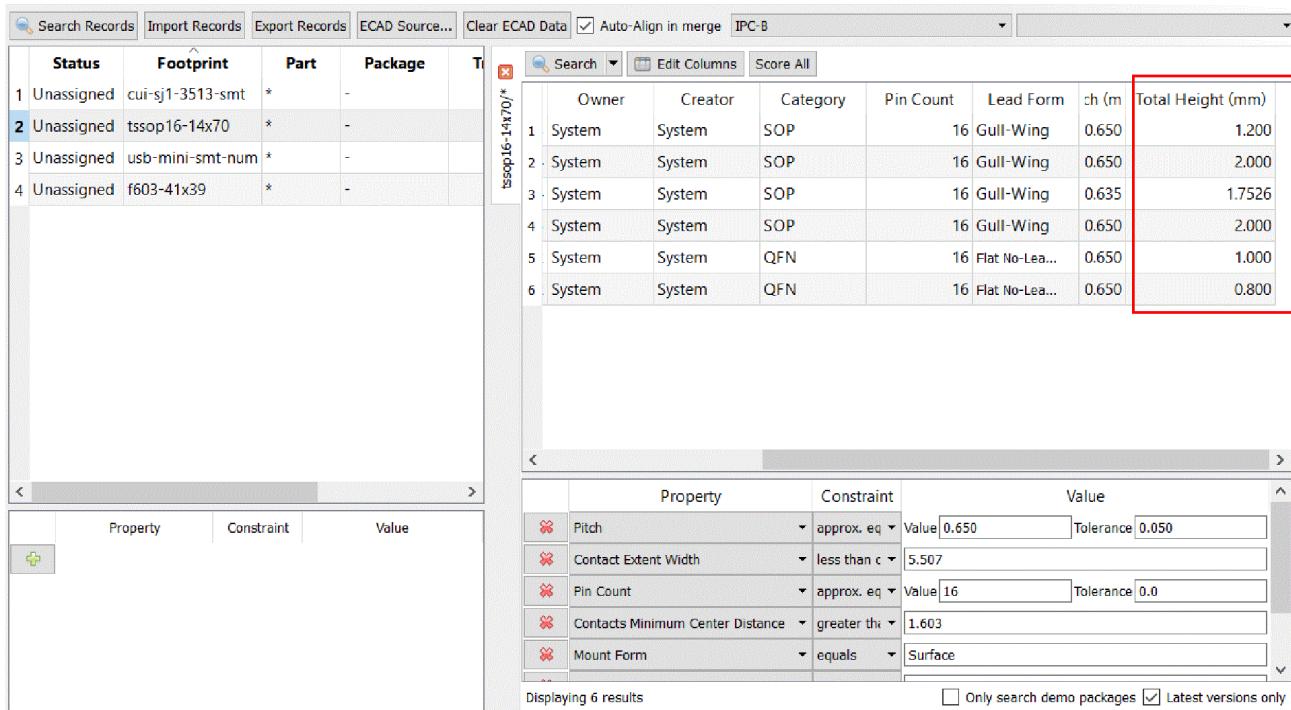
If you hover the mouse over the name of the package in the *Name* column, thumbnail view is displayed showing you the comparison between the package it found in the Repository and the footprint for the package receiving the lowest score.

Notice that even though the pin count is the same, the package types are different – hence the low score. As you can see, this is not a compatible package.

3. Move your cursor up the package names to see how the package contact points match up with the pads in the footprint.
The top two results have the highest score of 0.8 (80%). Comparing the contacts to the pads indicates that both are the same.
4. Scroll horizontally to bring the *Total Height* column into view.

Allegro X ECAD-MCAD Library Creator Tutorial

Synchronizing Libraries--What You Will Learn



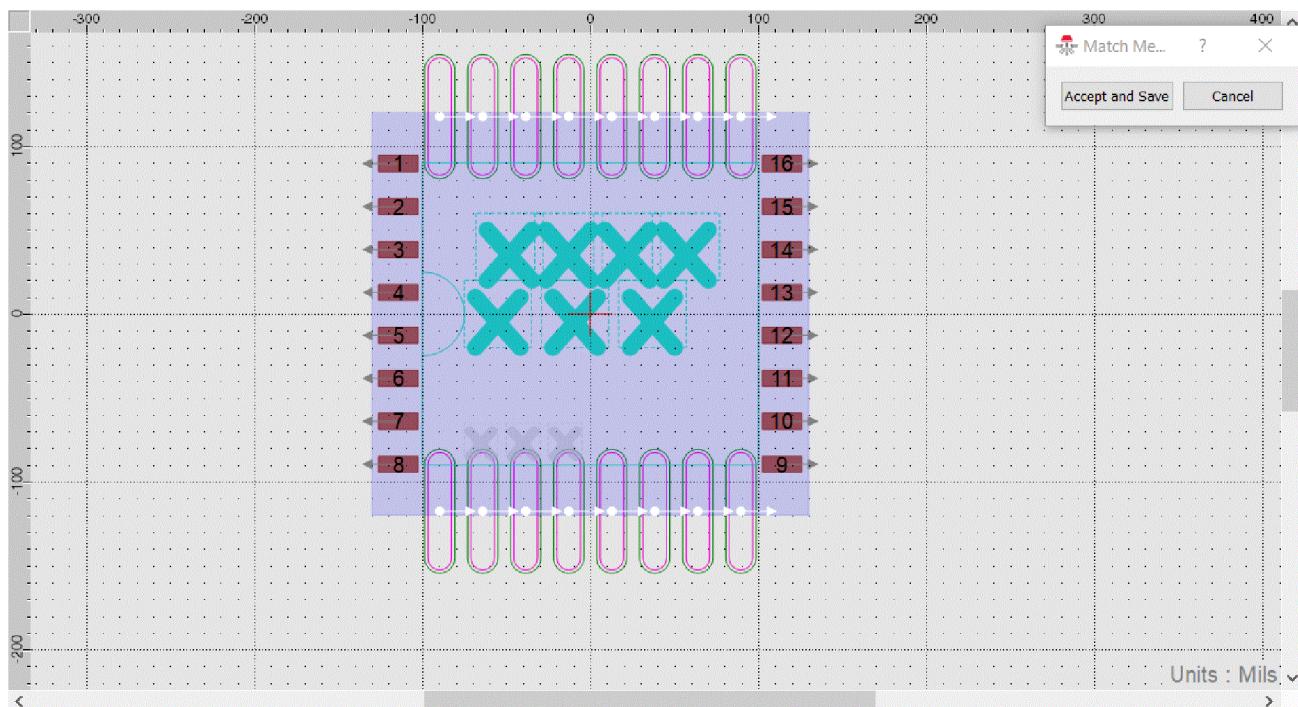
The screenshot shows the Allegro X ECAD-MCAD Library Creator interface. At the top, there is a toolbar with buttons for Search Records, Import Records, Export Records, ECAD Source..., Clear ECAD Data, Auto-Align in merge, and IPC-B. Below the toolbar is a search bar with the text "tssop16-14x70". The main area contains two tables. The left table lists footprints by status, footprint name, part number, package type, and revision. The right table lists packages by owner, creator, category, pin count, lead form, height (mm), and total height (mm). A red box highlights the "Total Height (mm)" column in the package table. Below these tables is a constraint editor with columns for Property, Constraint, and Value. The properties listed are Pitch, Contact Extent Width, Pin Count, Contacts Minimum Center Distance, and Mount Form. The values and constraints are as follows:

Property	Constraint	Value
Pitch	approx. eq	Value 0.650 Tolerance 0.050
Contact Extent Width	less than c	Value 5.507
Pin Count	approx. eq	Value 16 Tolerance 0.0
Contacts Minimum Center Distance	greater thi	Value 1.603
Mount Form	equals	Value Surface

Merging Package with Footprint

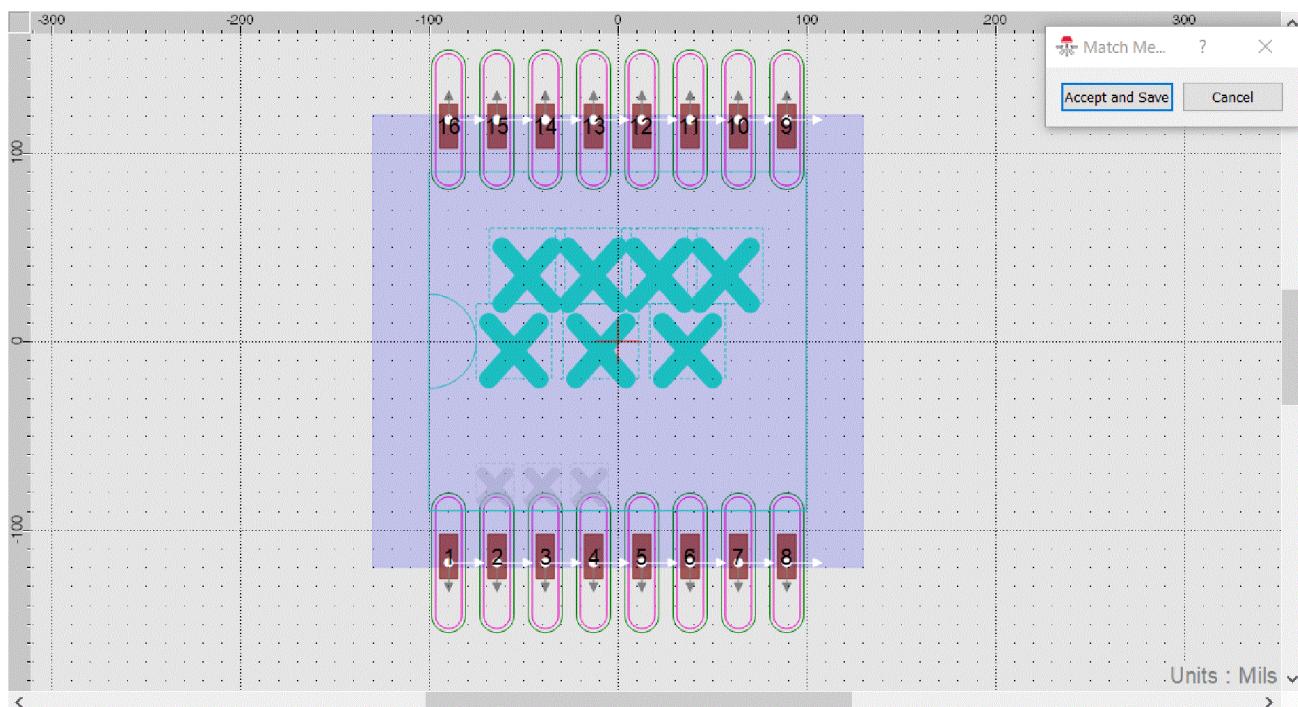
1. Right-click 2.0000 and choose *Merge*.

The package and footprint are merged and displayed in the 2D view. Notice that the pins do not line up with the footprint pads.



2. Click *Cancel* on the *Match Merge* dialog.
3. In the Synchronization tab, select *Auto-Align in merge* check box.
4. Right-click *2.0000* and choose *Merge*.

Notice how the results this time are much, much better.



- Click the *Accept and Save* button on the *Match Merge* dialog.

The package is associated with the footprint.

Property	Constraint	Value	Tolerance
Pitch	approx. eq	0.650	0.050
Contact Extent Width	less than	5.507	
Pin Count	approx. eq	16	0.0
Contacts Minimum Center Distance	greater than	1.603	
Mount Form	equals	Surface	

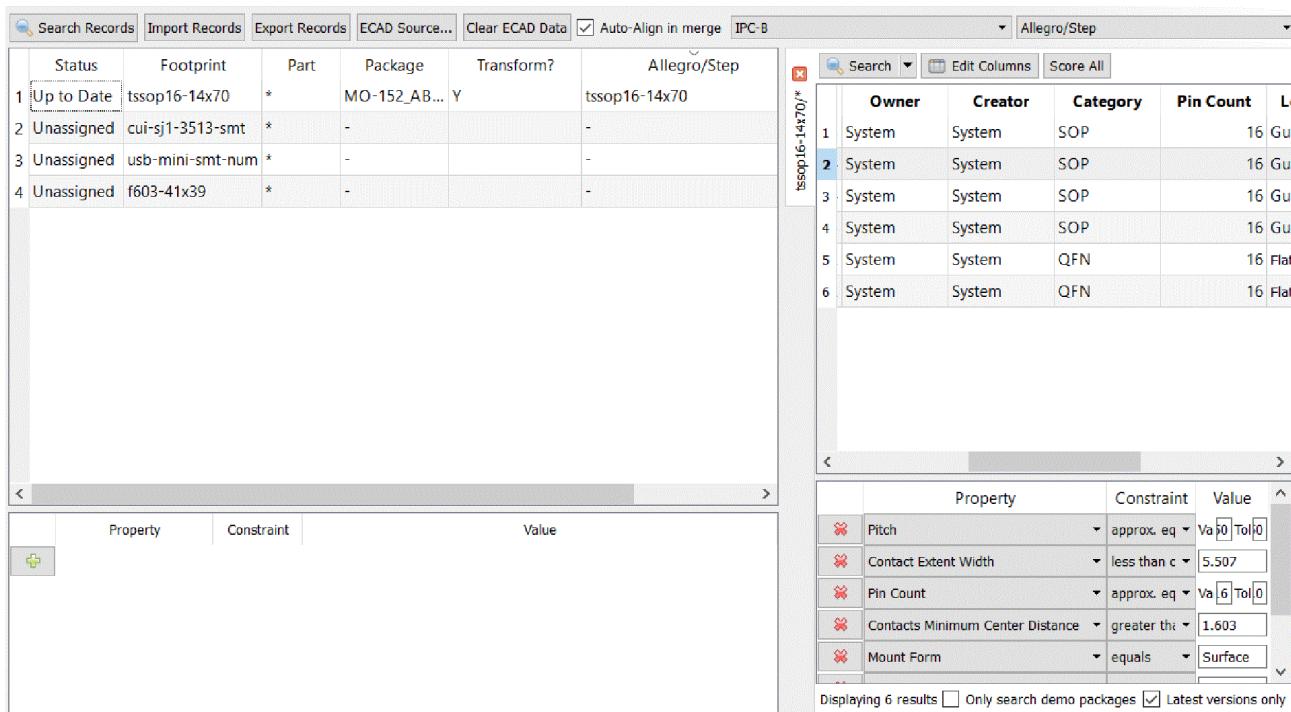
Next, you will push the STEP model to the original Allegro PCB Editor symbol.

- Right-click *tssop16-14x70* and choose *Push STEP to Allegro*.

A confirmation message is displayed.

- Click *Yes*.

The footprint is associated with both a *Package* and a *STEP* model.



You can view the revised footprint in Allegro PCB Editor in both the STEP Mapper as well as the 3D Viewer.

Checking Consistency of Footprints with Rules

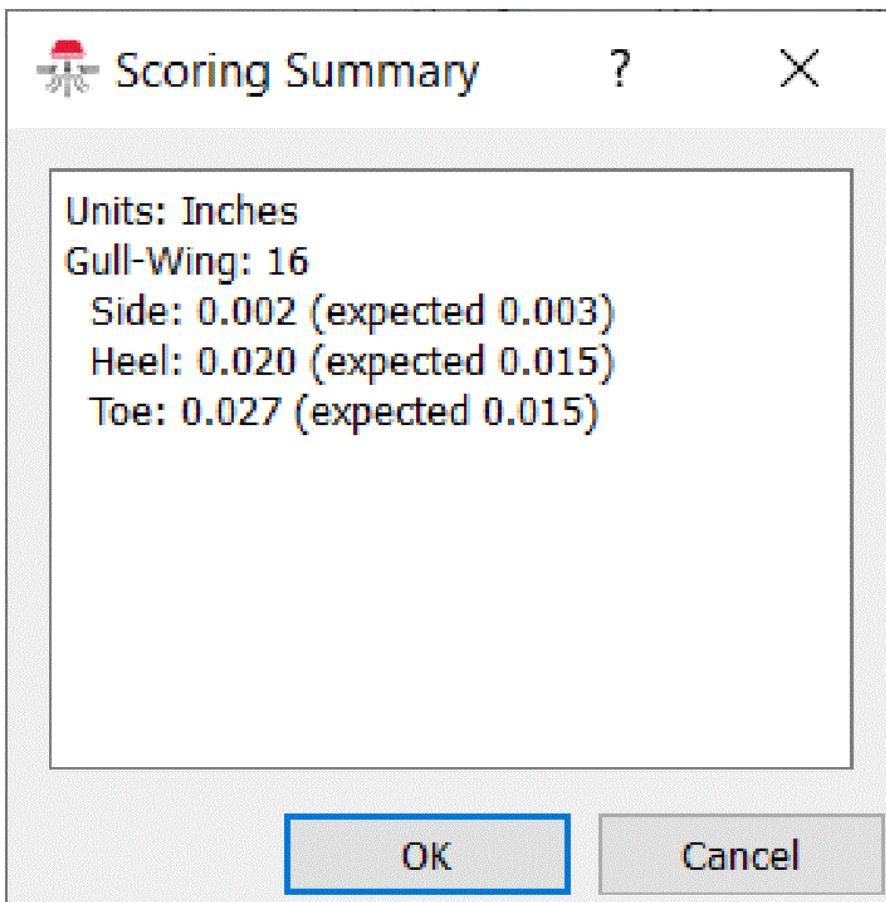
In this section, you will learn to check consistency of a footprint with a rule.

In [Comparing Packages and Footprint](#), the matching rule is set to *IPC B* and the package we used to merge with the footprint received a score of *0.8*.

To check consistency of the footprint *MO-152-AB* with rules:

1. Right-click the entry with Name *MO-152-AB* and choose *Score Summary*.

The *Scoring Summary* dialog appears displaying the expected *Toe*, *Heel* and *Side* offsets compared with the actual values for this package and footprint combination. Some of the results are quite larger than expected values.



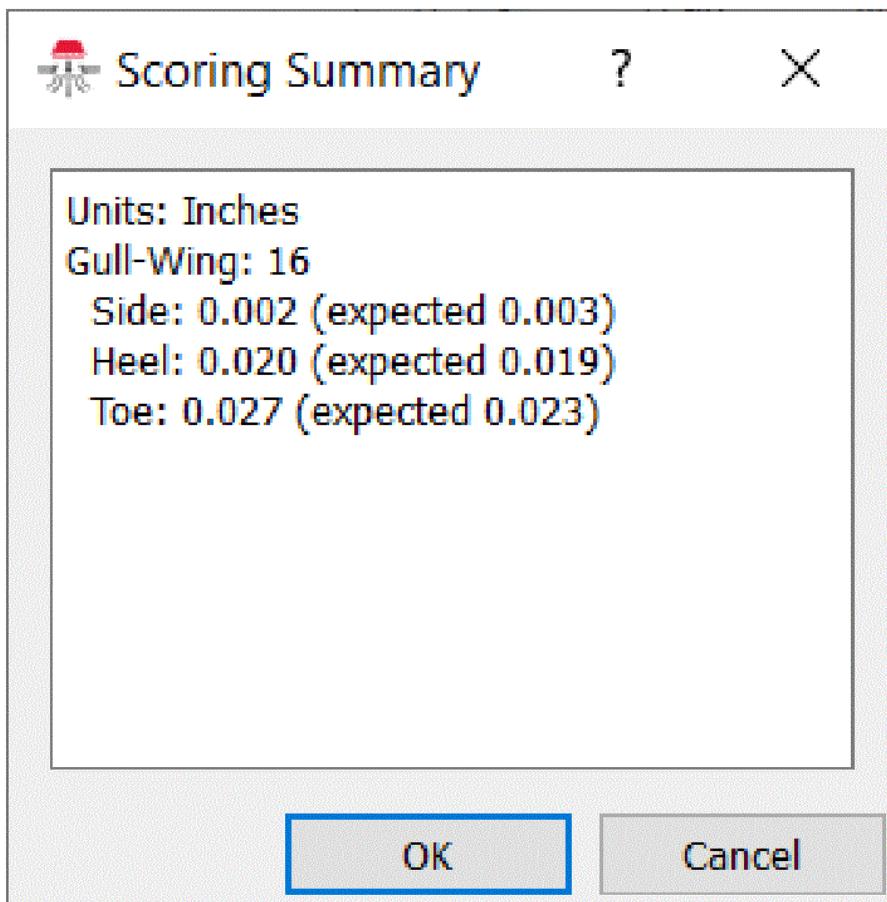
2. Click *OK* to close the *Scoring Summary* dialog.
3. Select the *IPC-A* from the drop-down list next to the *Auto-Align in merge* check box.
4. Click *Score All*.

The scores are much higher – from 0.8 to 0.94 (80% to 94%).

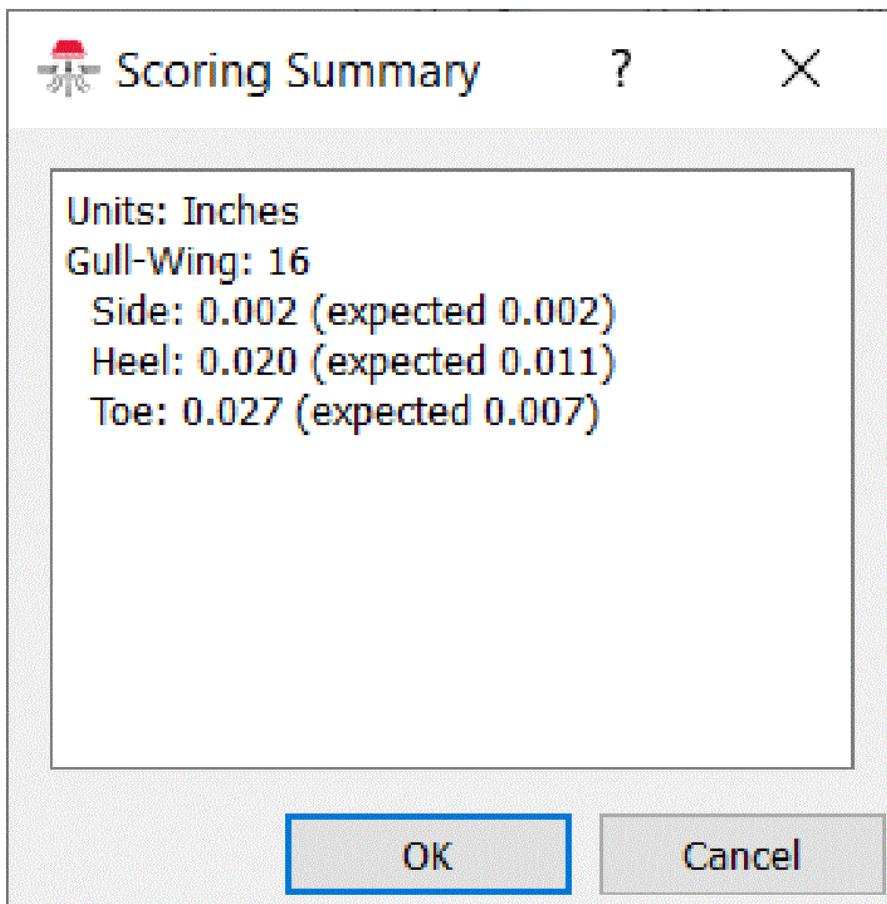
The screenshot shows the Allegro X ECAD-MCAD Library Creator interface. At the top, there is a toolbar with 'Search' and 'Edit Columns' buttons. Below the toolbar is a table titled 'tsop16-14x70/*' containing six rows of data. The columns are: Score, Name, Version, Owner, Creator, Category, and Pin Count. The first row has a highlighted 'Score' value of 0.94. Below the table is another table titled 'Property' with five rows of constraints. The columns are: Property, Constraint, Value, and Tolerance. The constraints listed are: Pitch (approx. eq, 0.650, 0.050), Contact Extent Width (less than c, 5.507), Pin Count (approx. eq, 16, 0.0), Contacts Minimum Center Distance (greater th, 1.603), and Mount Form (equals, Surface). At the bottom of the interface, there is a message 'Displaying 6 results' and two checkboxes: 'Only search demo packages' and 'Latest versions only'.

- Right-click the entry with Name *MO-152-AB* and choose *Score Summary*.

The *Scoring Summary* dialog appears. Notice that the actual values are closer to the expected results.



6. Click *OK* to close *Scoring Summary*.
7. Select the *IPC-C* from the drop-down list next to the *Auto-Align in merge* check box.
8. Click *Score All*.
The resulting score is down to *0.56* and check the *Score Summary*. The actual values are much higher than the expected tight results of the *IPC-C* rule.



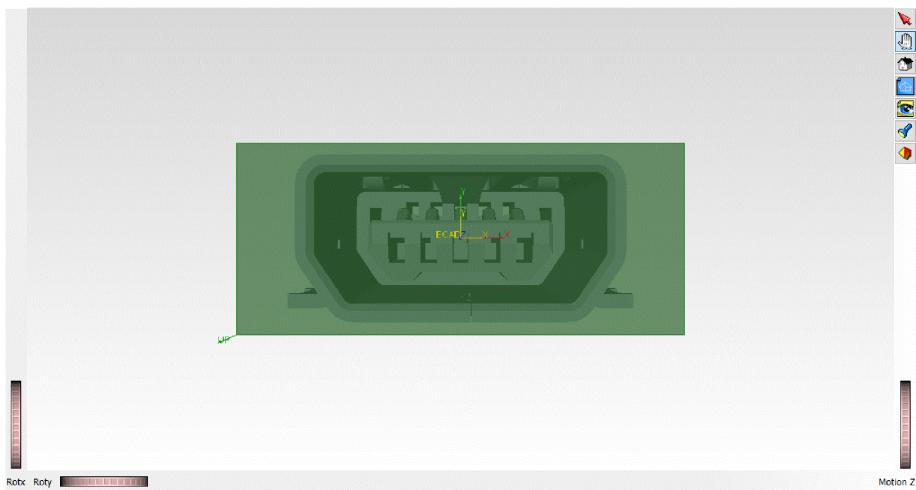
Adding Packages to Repository

In this section, you will import a vendor provided STEP model, apply minimal featurization information so that it can be added to the repository. You will also add and merge the STEP Model with an existing footprint.

Importing the STEP Model

In this section, you will import the step model *Molex_565790576.stp* from
<your_install_dir>/doc/lc_tut/tutorial_examples/STEP_models.

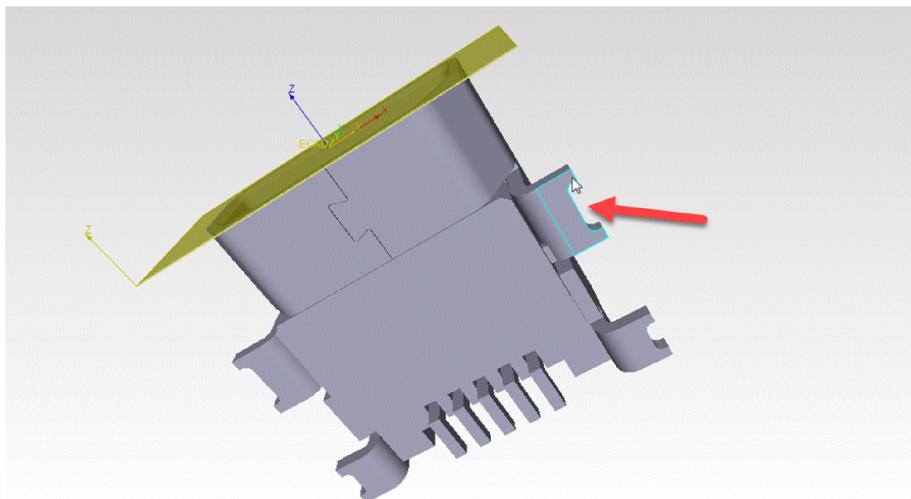
For more information about importing STEP Model, follow the steps mentioned in the [Importing STEP Models](#) section.



Editing Seating Pane

In this section, you will edit and move the seating pane to the bottom of the lead as shown in the following figure.

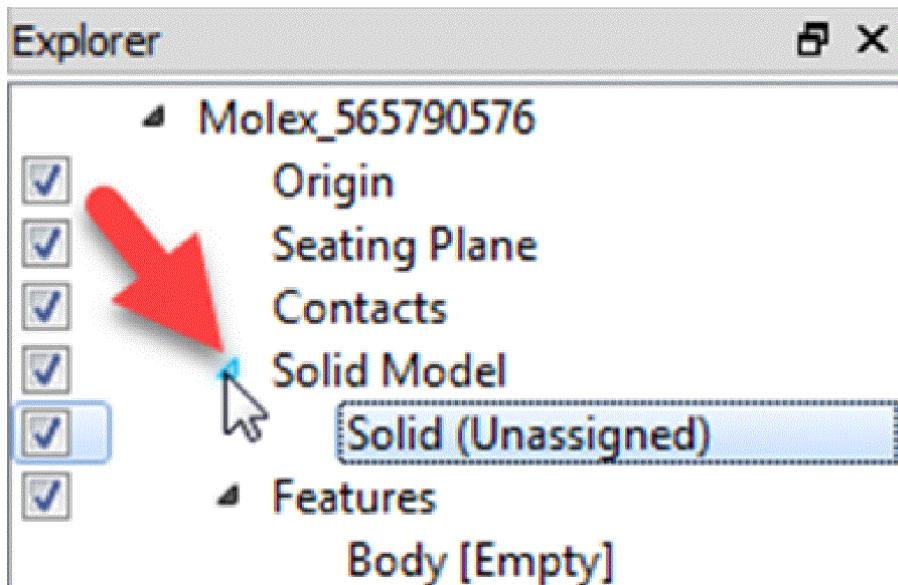
For more information about editing a seating pane, follow the steps mentioned in the [Editing Seating Pane](#) section.



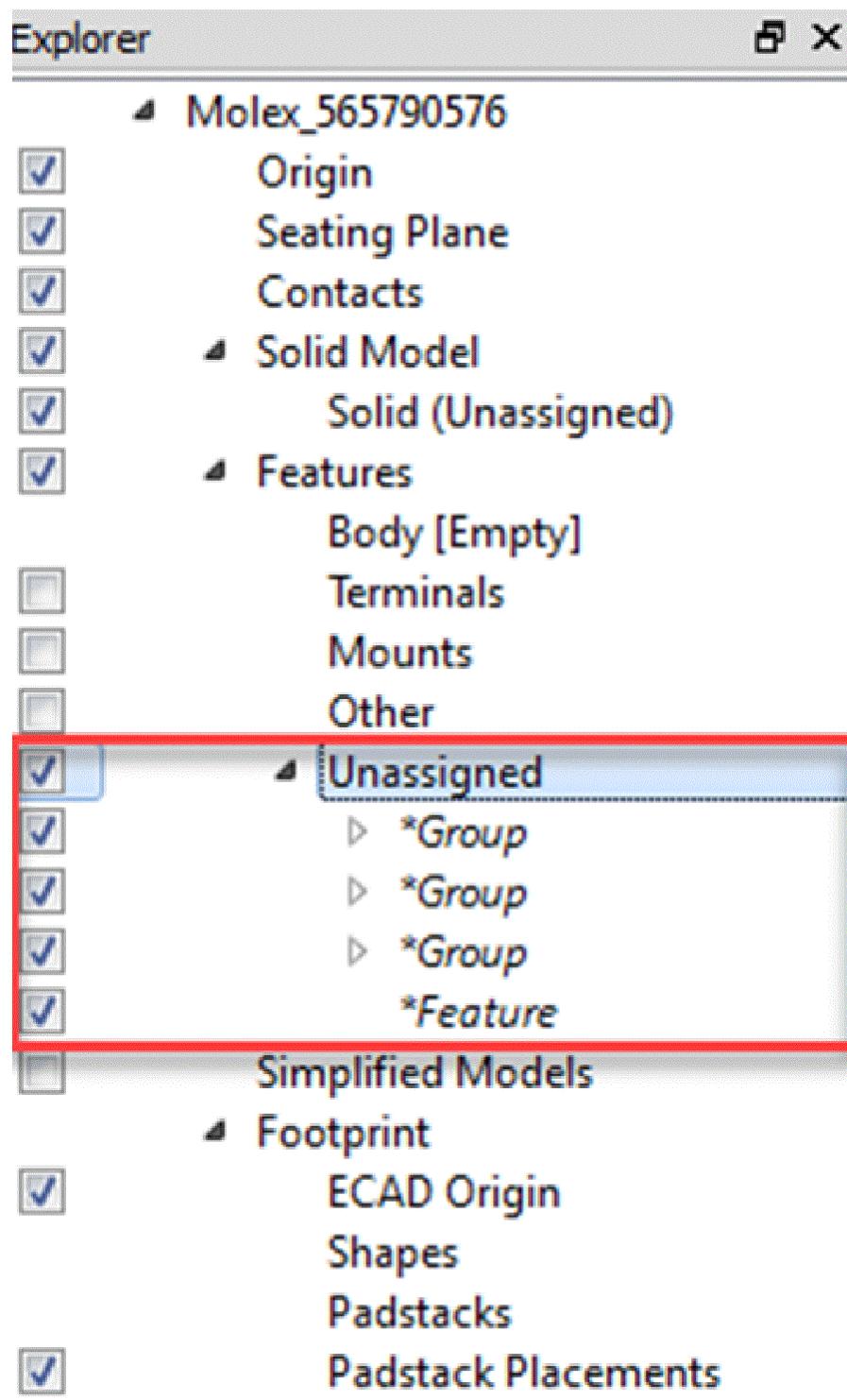
Adding Contact Features

In this section, you will add *Touching Features* to the unassigned solid model. To assign the contact features:

1. In the *Explorer* panel, expand *Solid Model*.



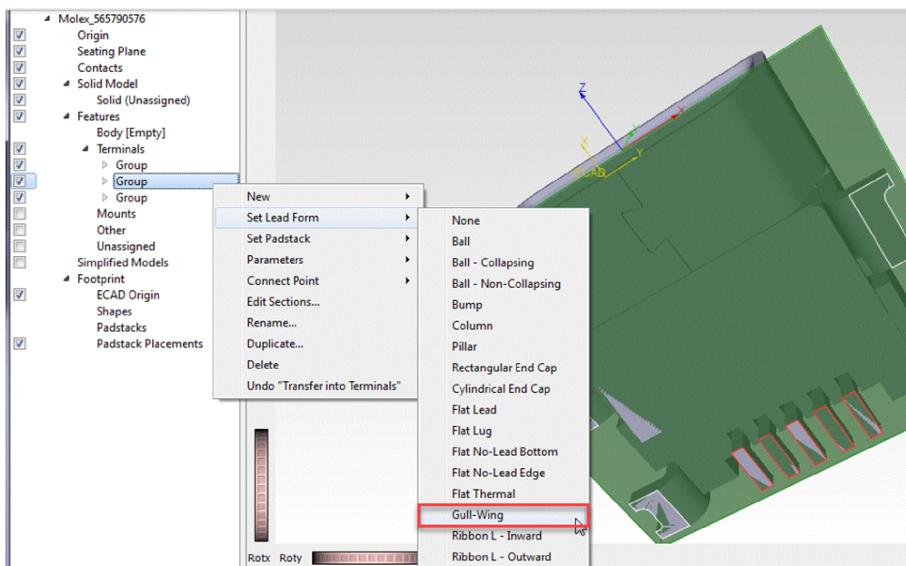
2. Right-click *Solid Model - Solid (Unassigned)* and choose *Contact Features - Training/Touching Features*.
3. Expand *Features - Unassigned*.



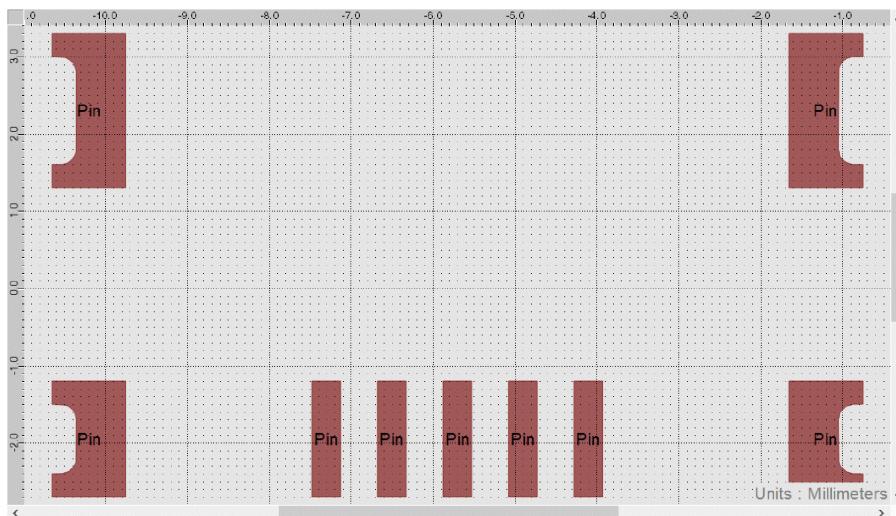
4. Right-click *Unassigned - Feature* and choose *Delete*.
5. Select and drag all the three *Groups* under *Features - Unassigned* and drop them to the

Features - Terminals node.

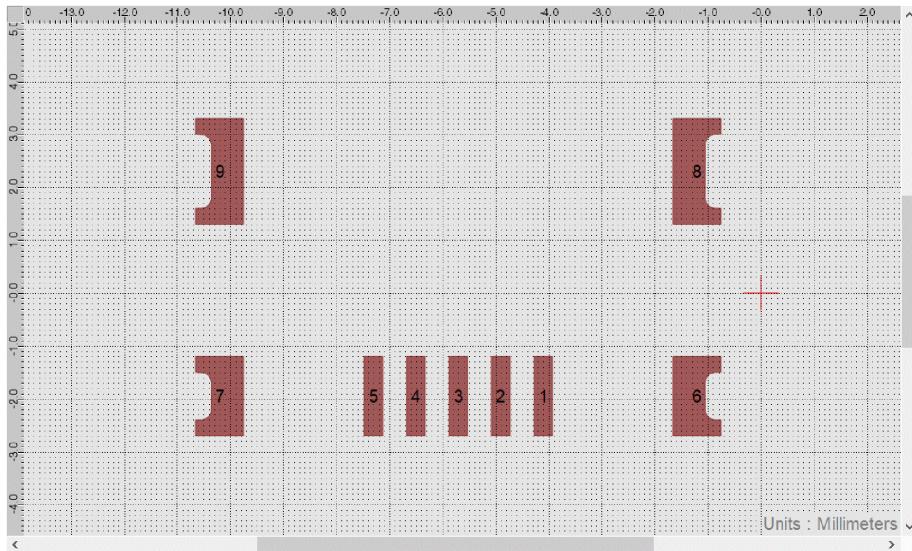
6. Locate and select the *Group* under *Features - Terminals* that corresponds to the five signal pins.
7. Right-click the selection and choose *Set Lead Form - Gull-Wing*.



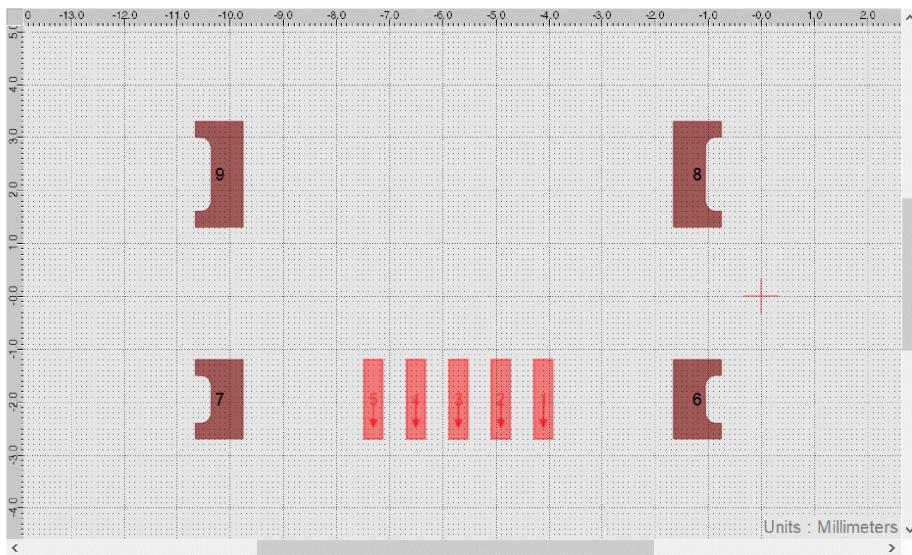
8. Set the other two *Groups* to *Side Lead – Concave*.
9. Click the *Footprint (2D)* tab.
10. Rotate the *Footprint (2D)* view so that the view is as shown in the following figure.



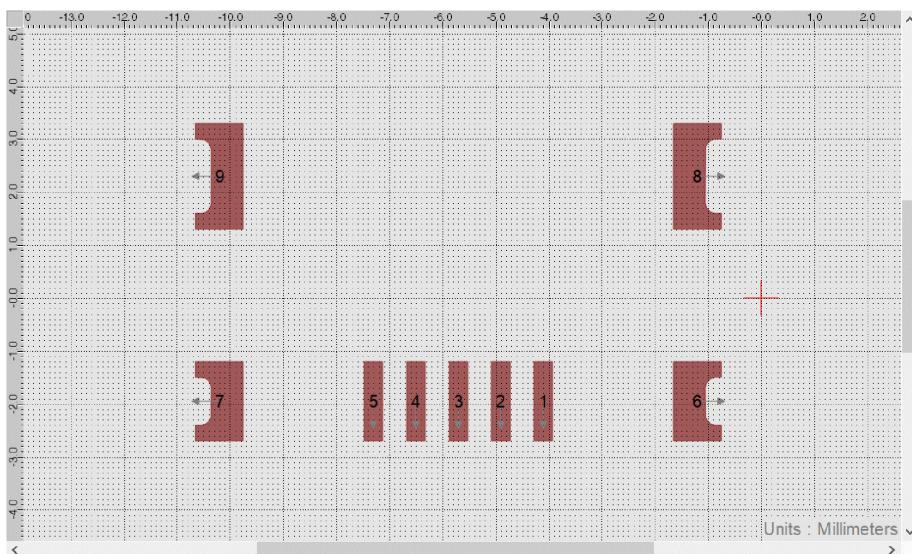
11. Change the pin numbers using so that they match the pin numbers as shown in the following figure.



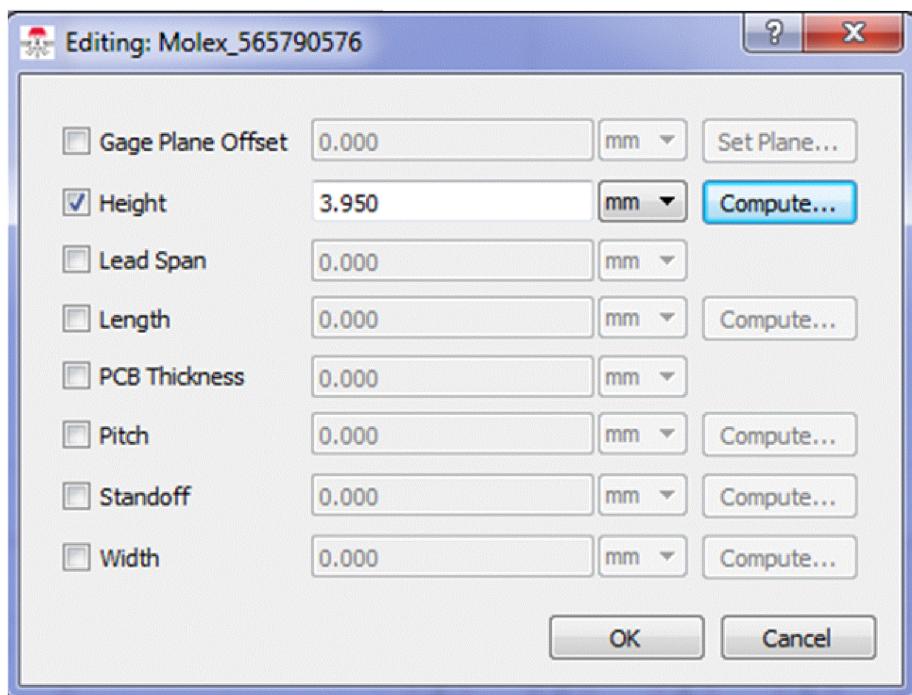
12. In the *Explorer* pane, right-click the five pins under *Terminals - Group – Gull-Wing* and choose *Orientation - South* to set the orientation of the pins.



13. Similarly, set the orientation of pins 6 and 8 to *East*, and of pins 7 and 9 to *West*.
The results for the above orientation is as shown in the following figure.



14. In the *Explorer* pane, select and drag the *Solid Model - Solid (Unassigned)* and drop to the *Features - Body (Empty)* node.
15. Right-click *Molex_565790576* and choose *Parameters - Edit*.
Editing dialog is displayed.
16. Select the *Height* check box and click the *Compute* button.
The result is as shown in the following figure.



17. Click *OK*.

Saving the Footprint

In this section, you will save this version of the *Molex_565790576* STEP Model to make it part of the Library Creator repository.

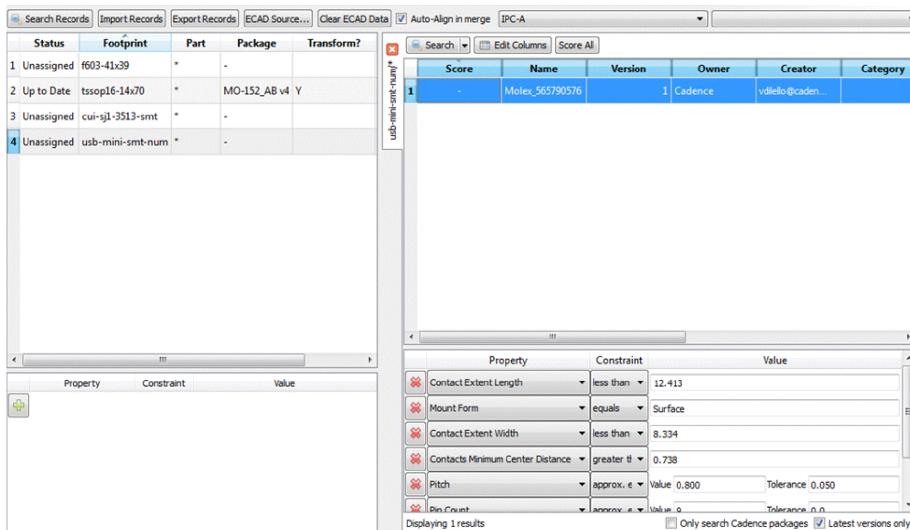
For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

Synchronizing Non-Standard Footprints

In this section, you will associate two non-standard footprints corresponding to connectors. You will create an association and check for consistency between the footprint and the connector model.

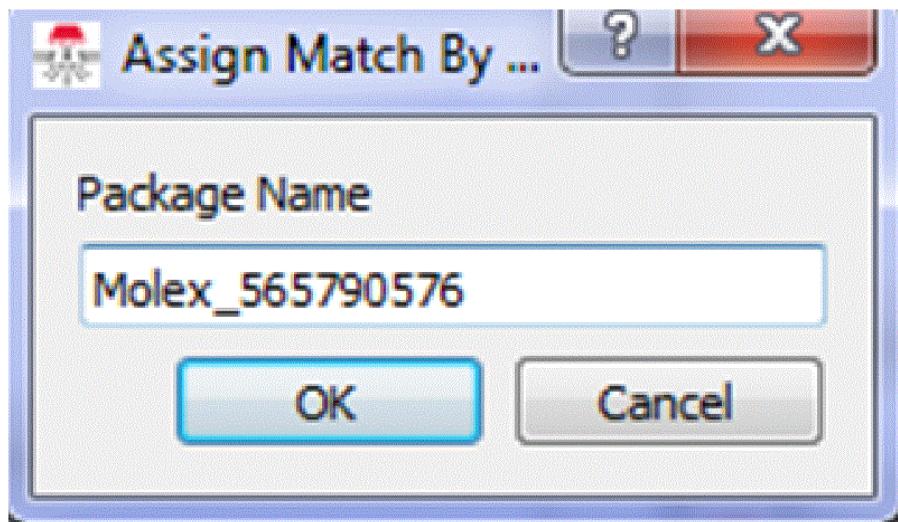
1. Select the Synchronize tab at the bottom of the Library Creator window.
2. Right-click *usb-mini-smt-num* and choose *Auto Search*.
3. Click the *Search* button.

Matching search results are displayed. In this example, only one result is displayed.

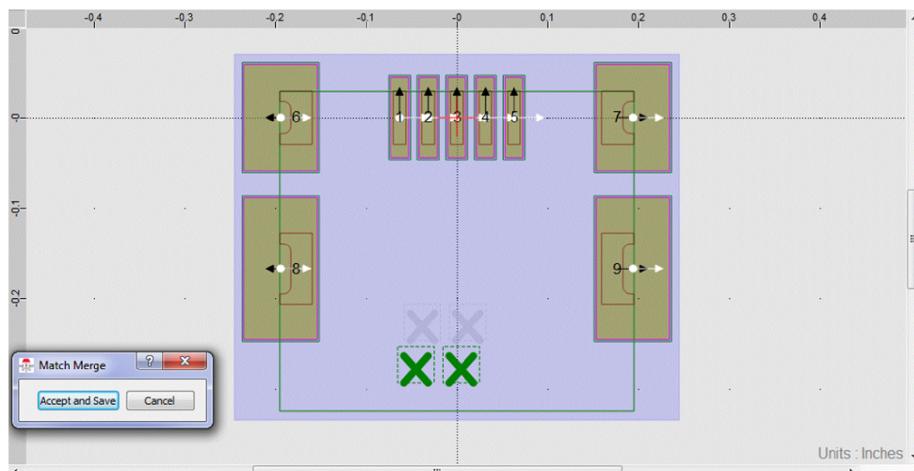


In such cases, if you know only one match is available, you can use the *Match By Name* option.

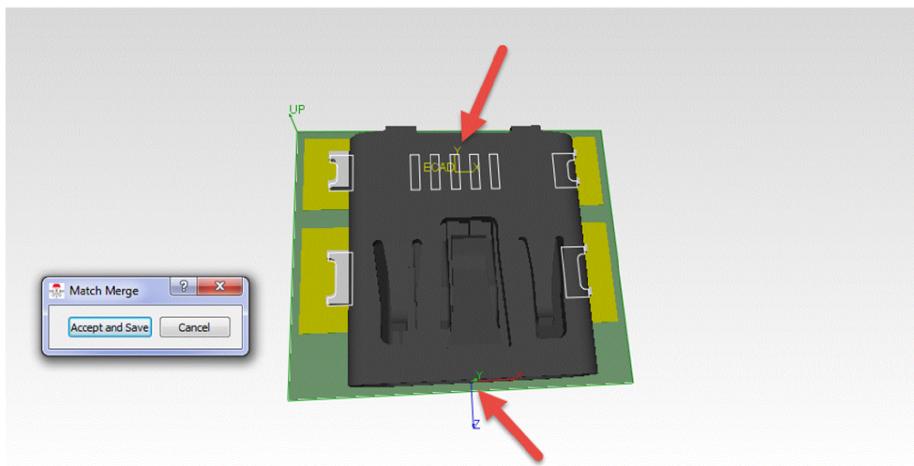
4. Right-click *usb-mini-smt-num* and choose *Match By Name*.
Assign Match By Name dialog is displayed.
5. Enter *Molex_565790576* in the *Package Name* field and click *OK*.



The canvas displays the correct association between the package and the footprint as shown in the following figure.



Inspect the results in both the 2D and 3D views to verify if this is a good match. The only difference is that the two starting points are at different locations. The footprint starting point is in the center of the middle pad while in the 3D model, the starting point is at the front center of the body.

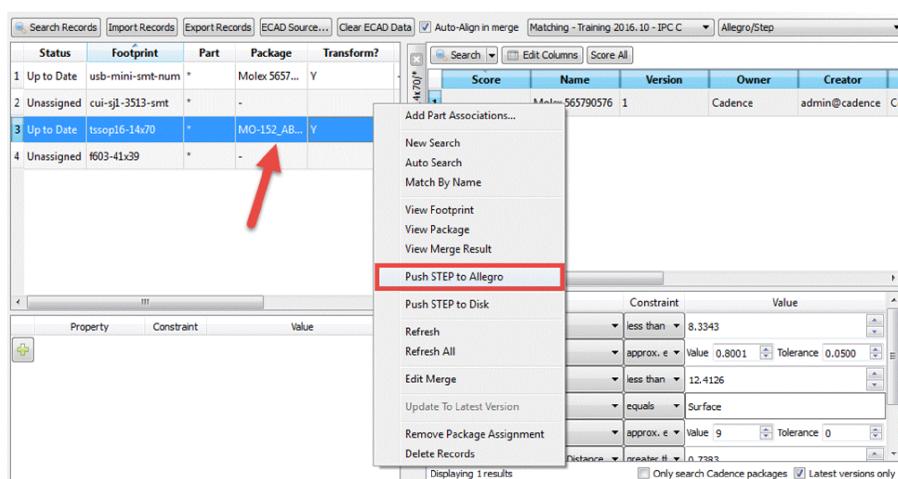


6. Click *Accept and Save*.

The package is added to the entry.

7. Right-click the entry and select *Push STEP to Allegro*.

A confirmation message is displayed.



8. Click *Yes*.

You can verify the STEP model in Allegro PCB Editor footprint by viewing the STEP Mapper and the 3D Viewer.

Summary

In this module, you created a series of persistent relationships between in the Library Creator repository, between the footprint and the package stored in the repository.

Exercise

Perform the steps mentioned in the [Synchronizing Existing Footprints with Packages](#) using the f603-41x39 footprint and merge the footprint with the package for which the values of Side, Heel, and Toe are closer to the expected results in the Scoring Summary.

Creating Multi-Grid BGA Packages

What You Will Learn

In this module, you will learn to create a BGA package that includes many different pin pitches. This type of BGA package can be created by loading a template and then adding pins using a CSV file.

Accessing the Package Templates

In this section, you will access the Ball Grid Array (BGA) template.

For more information about accessing a template, follow the steps mentioned in the [Accessing the Package Templates](#) section and click the BGA template.

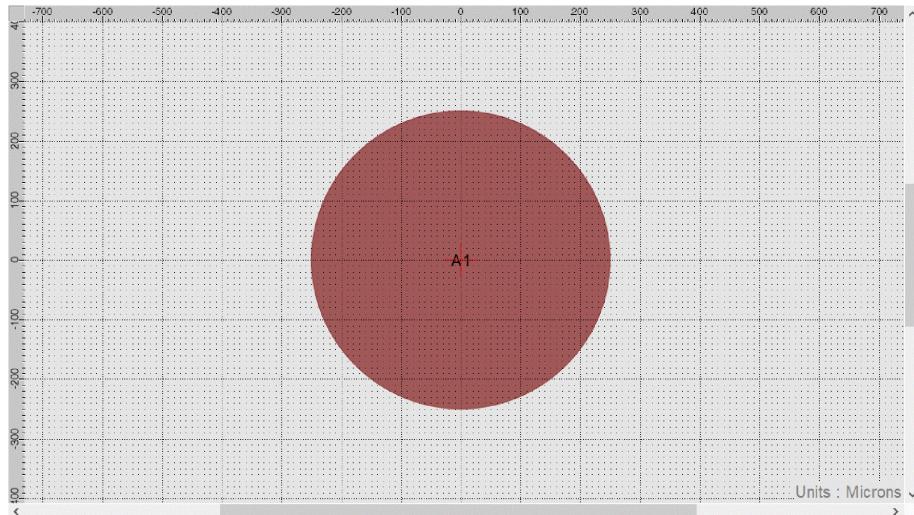
Creating a New Instance of the BGA Template

1. Double-click the *Value* column of the *name* field in the parametric table of the *Templates (Viewing)* dialog.
The field becomes editable.
2. Enter `demo_test_3`.
3. Select *Microns* from the units drop-down list.
4. Change all the parametric values as displayed in the following figure.

	Value
name	demo_test_3
lead_form	Ball
units	Microns
n_x	1
n_y	1
pitch_x	400
pitch_y	400
package_height	700
standoff	350
body_x	37500
body_y	37500
terminal_diameter	500

5. Choose New Instance and close the *Templates (Viewing)* dialog.
6. Click Footprint (2D) to view.

You can see that a pin A1 is displayed on the canvas.

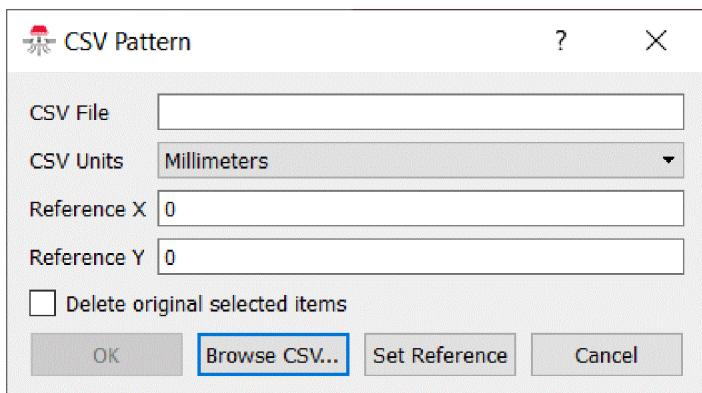


Adding Pins

In this section, you will add pins using the *bga_1434_37_5_off_grid.csv* file available at <your_install_dir>/doc/lc_tut/tutorial_examples.

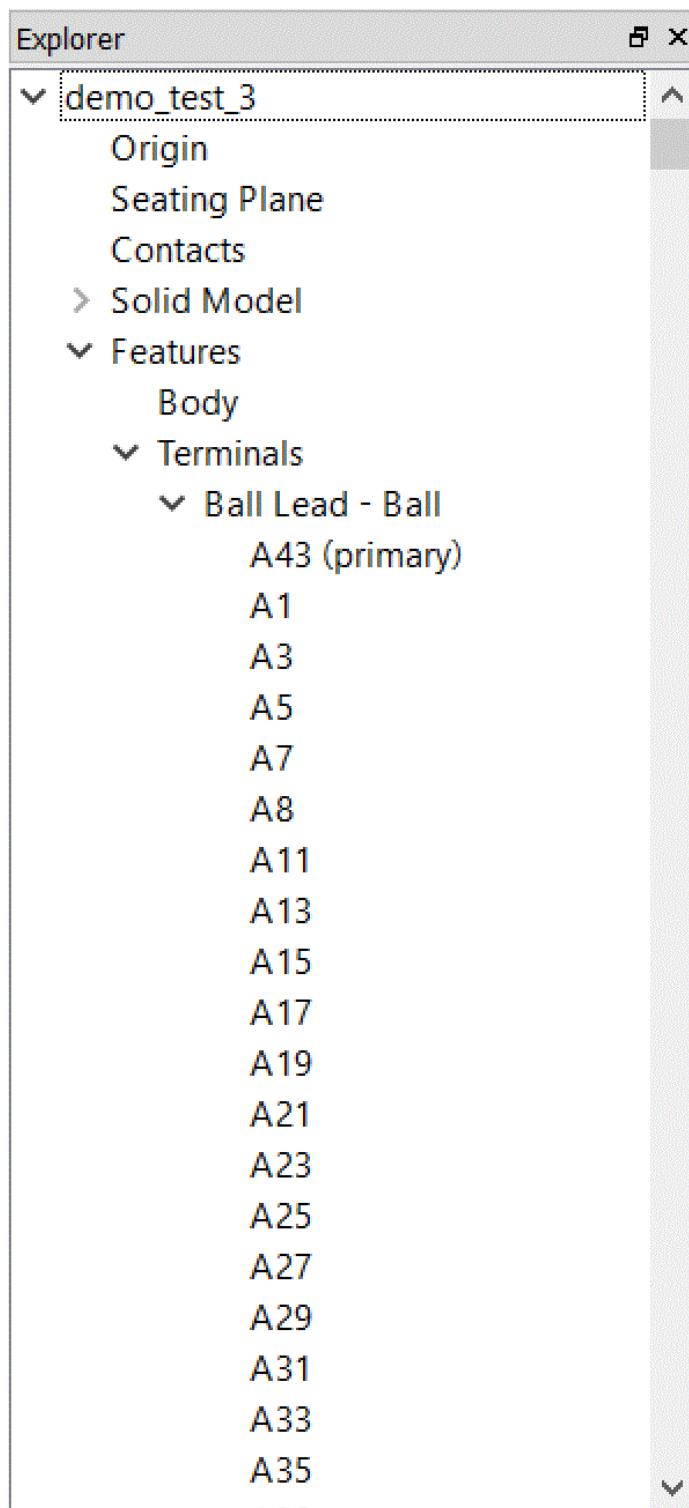
1. Select the pin A1.
2. Right-click the selection and choose *Pattern Copy - CSV Pattern*.

CSV Patter dialog box is displayed.



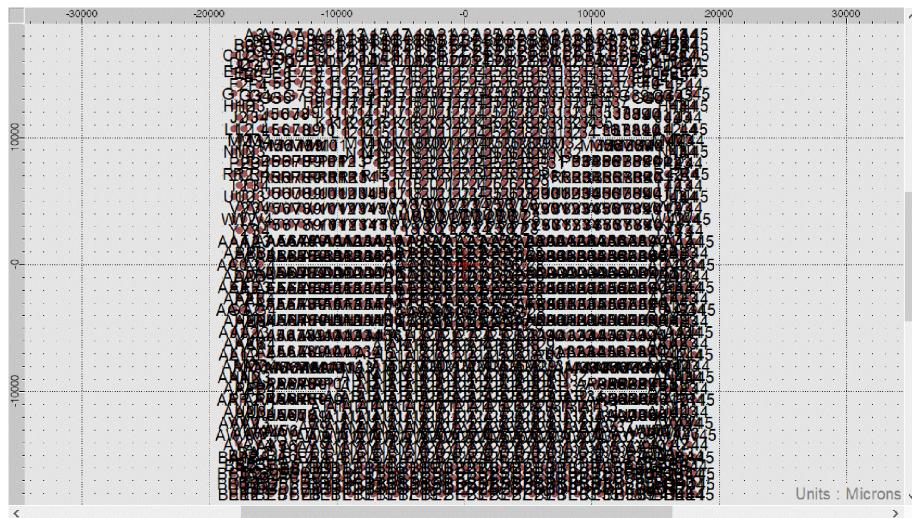
3. Select *Same as package* from the *CSV Units* drop-down list.
4. Click *Browse CSV*, locate and select the *bga_1434_37_5_off_grid.csv* file from `<your_install_dir>/doc/lc_tut/tutorial_examples`.
5. Click *OK*.

After the processing is completed, the *Explorer* pane is updated with the added pins under the *Features > Terminals > Ball Lead – Ball* node.

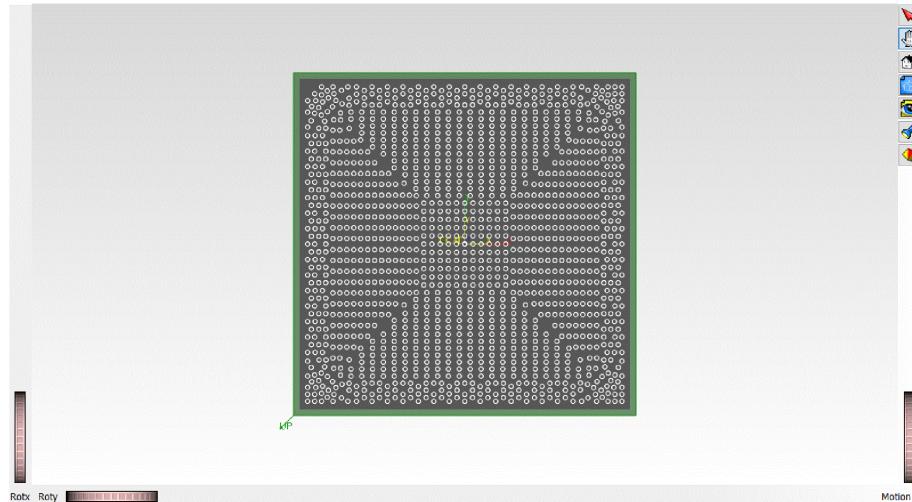




6. Click  to fit the footprint on the canvas.



7. Click *Package (3D)* to see that the added pins are also visible in 3D.



Saving the Footprint

It is important that you save a new version of the footprint before exporting it to PCB Editor.

For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

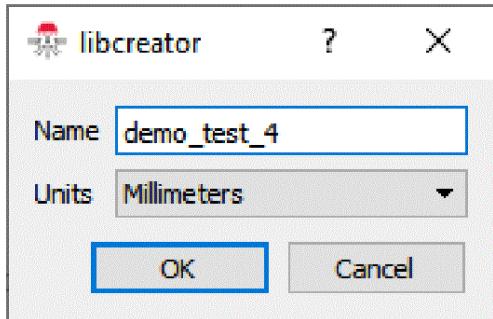
Creating Non-Standard Packages

What You Will Learn

In this module, you will learn to create a non-standard *LFB18(0603)_SG Series* package.

Creating Packages

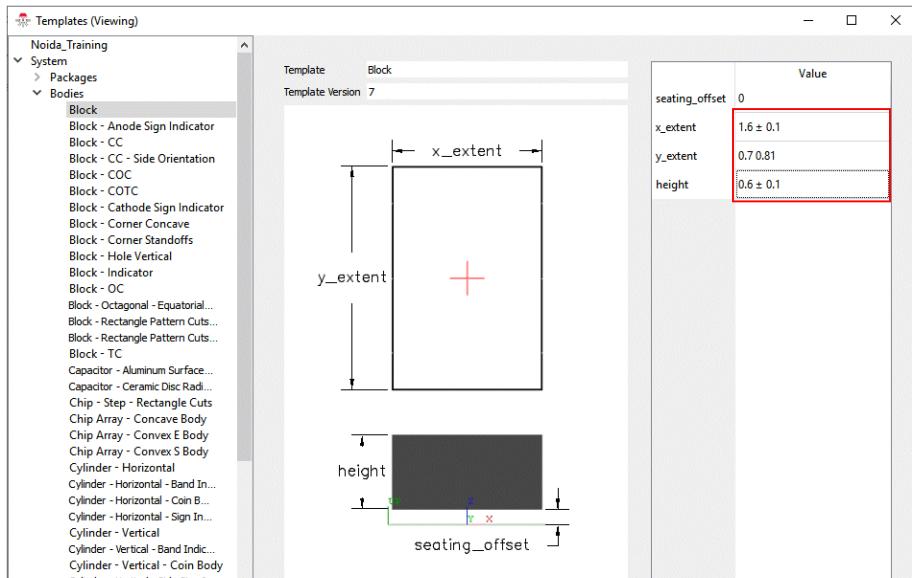
1. Choose File - New Package and enter `demo_test_4` in the Name field of
2. Click OK.



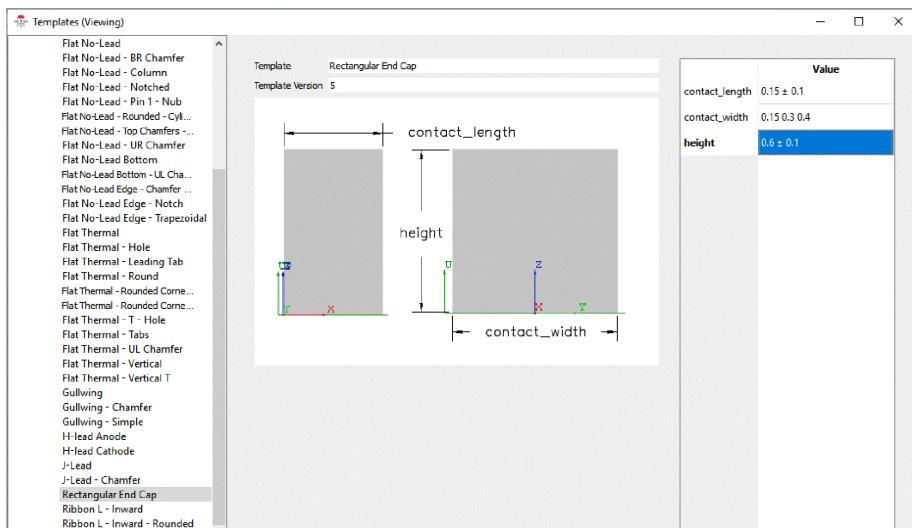
A new package is created and displayed in *Explorer* as well as on the Canvas.

Adding Pins to Package

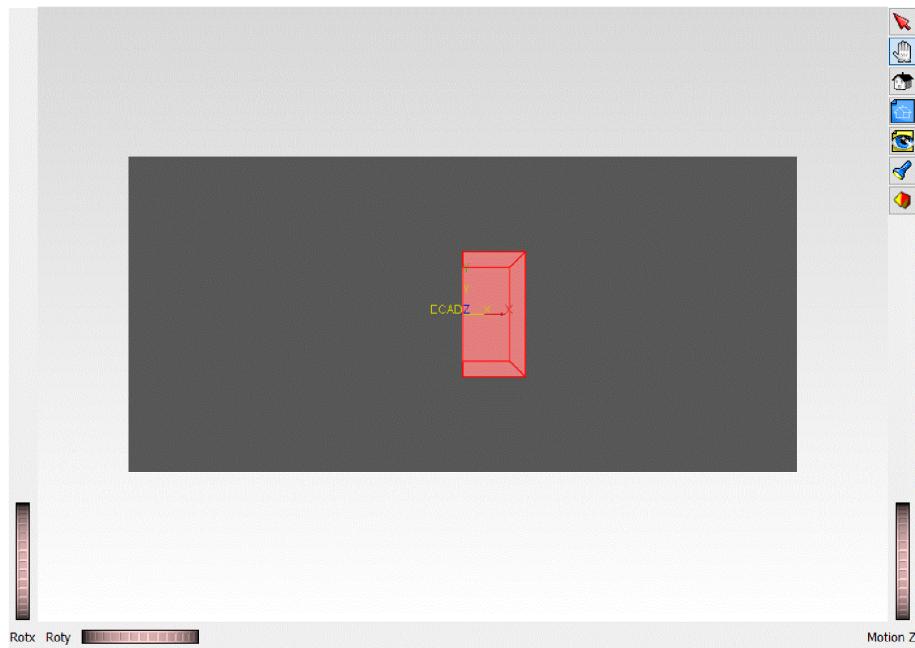
1. Choose *Tools - Templates*.
2. Choose *System - Bodies - Block* in the *Templates (Viewing)* dialog to add a body to this new package.
3. Change the parametric values as shown in the following figure and click *Add Instance*.



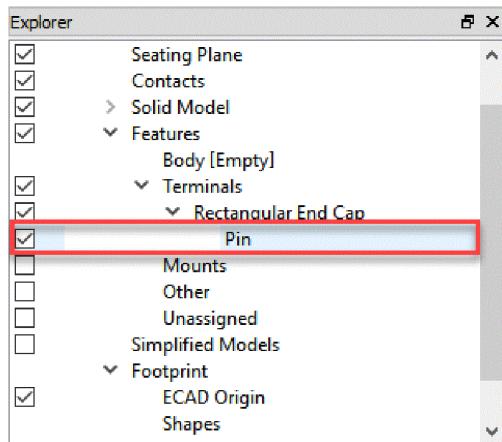
4. Choose *System - Terminals - SMT - Rectangular End Cap* in the *Templates (Viewing)* dialog.
5. Click *Rectangular End Cap* and change the parametric values as shown in the following figure.



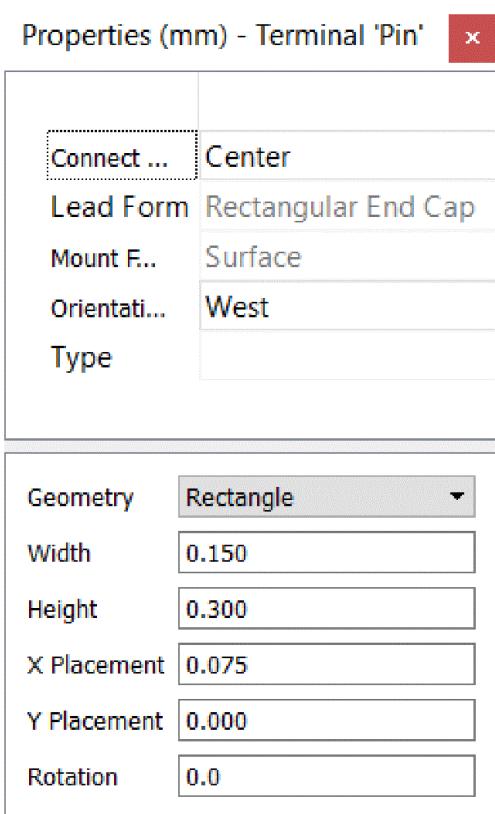
6. Click *Add Instance* and close the *Templates (Viewing)* dialog.
 In the *Package (3D)* tab, you can now see that the body and one contact are visible.



7. Expand *Features - Terminals - Rectangular End Cap*
The newly added pin is displayed in the *Explorer* pane.

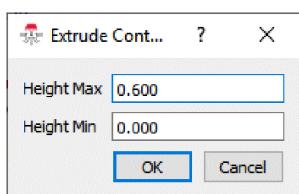


8. Select *Pin* and see the details in the *Properties* panel.

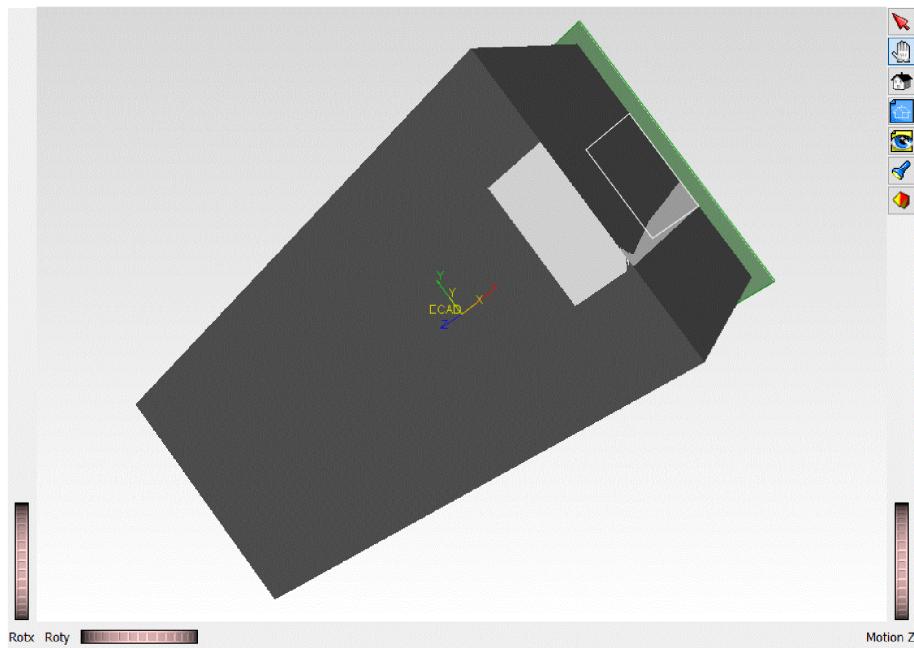


The X and Y placement for the added pin is 0.075 0.000 respectively which means that the starting point for this pin is in the center of the pin. This can be verified in the *Package (3D)* pane as we can see the package datum point (0,0) on the left side of the pin (red, yellow and green arrows) as shown in the following figure.

9. Change the value of *X Placement* to 0.725 and click anywhere in the *Properties* panel for the new value to be accepted.
10. Right-click *Pin* in the *Explorer* pane and choose *Extrude Contact*.
11. Click *OK*.



12. Click *demo_test_4* in the *Explorer* pane for the extrusion command to take effect. Reposition the 3D view to see the contact now properly located at the right end of the body as shown below.



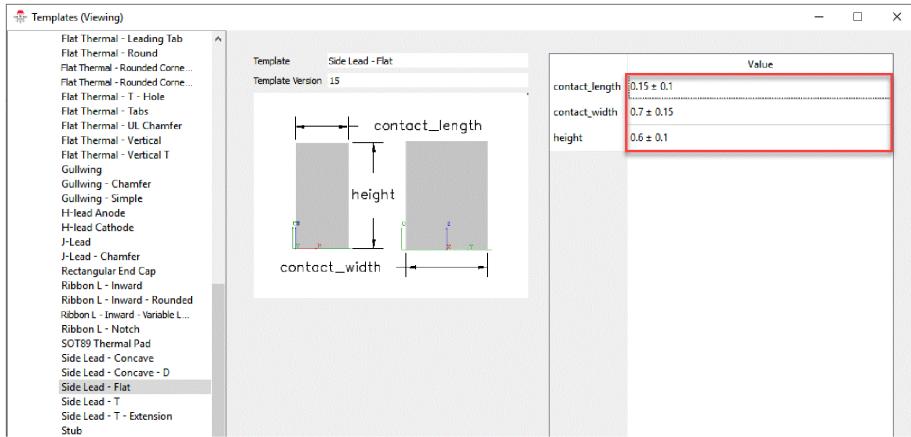
13. Repeat the *Step 6* to *Step 14* to add another left side pin by keeping the value of *X Placement* to -0.725 .



Adding Side Contacts

1. Choose *Tools - Templates*.

2. Choose *System - Terminals - SMT - Side Lead – Flat* to add side contacts to this new package.
3. Change the parametric values as shown in the following figure and click *Add Instance*.

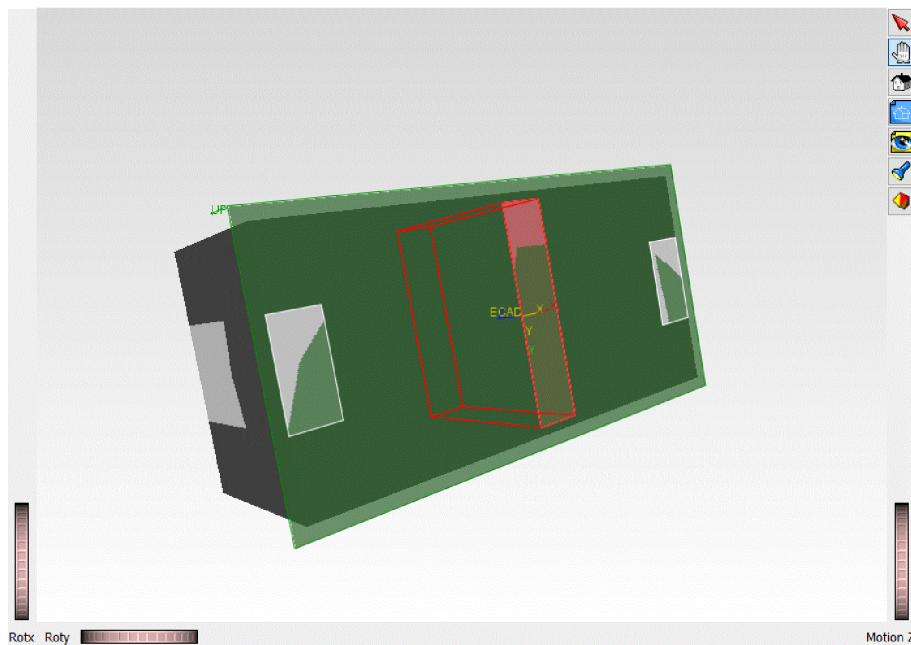


4. Expand *Features - Terminals*

The newly added pin is displayed under *Side Lead - Flat*.

5. Select the latest added pin.

Library Creator has positioned the pin at X = 0.075 and Y = 0.000. This time, you will use a different method to locate the pin properly on the body. You will also change the pin orientation.



6. In the *Properties* panel:

a. change the value of *Rotation* to 90.

b. change the value of *X Placement* to 0.000.

c. click anywhere in the *Properties* panel for the new values to be accepted.

Now that you have the center of the newly added pin at the 0,0-starting point of the body, you can easily move it into its correct position.

d. Change the value of *Y Placement* to 0.325.

7. Right-click the pin in Explorer and choose *Extrude Contact* to update the 3D view for the new pin position.

8. Click *OK* in the *Extrude Contact* dialog box.

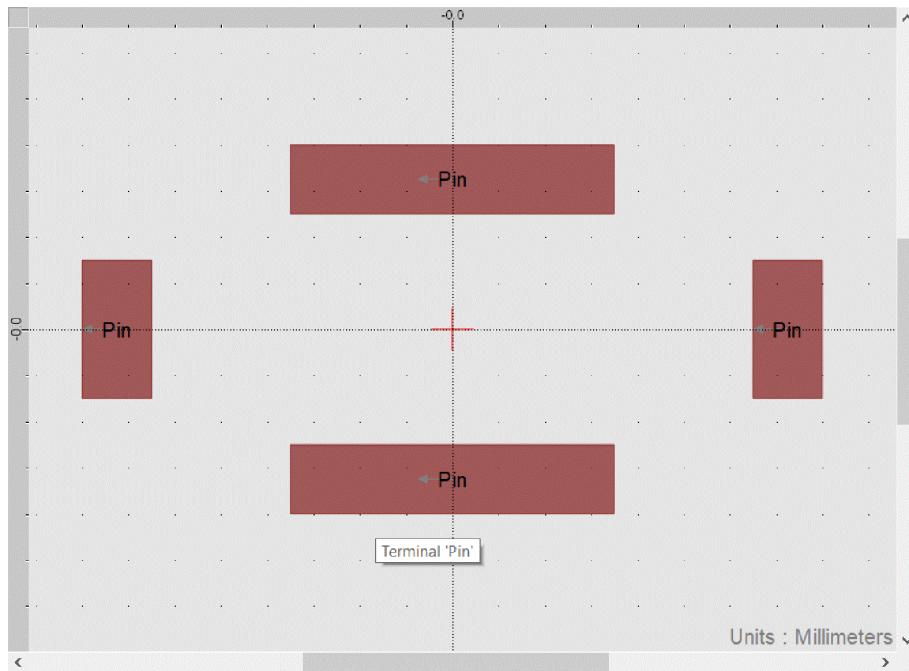
9. Click anywhere in the *Explorer* pane.

The 3D view is updated.

10. Repeat from *Step 2* onwards to add the other side lead by keeping the value of *Y Placement* to -0.325.



11. Click on the *Footprint (2D)* pane view and change the pin orientations.



Assigning Pin Numbers

In this section, you will assign pin numbers to the four pins.

For more information about assigning pin numbers, follow the steps mentioned in the [Assigning Pin Numbers](#) section.

Saving the Footprint

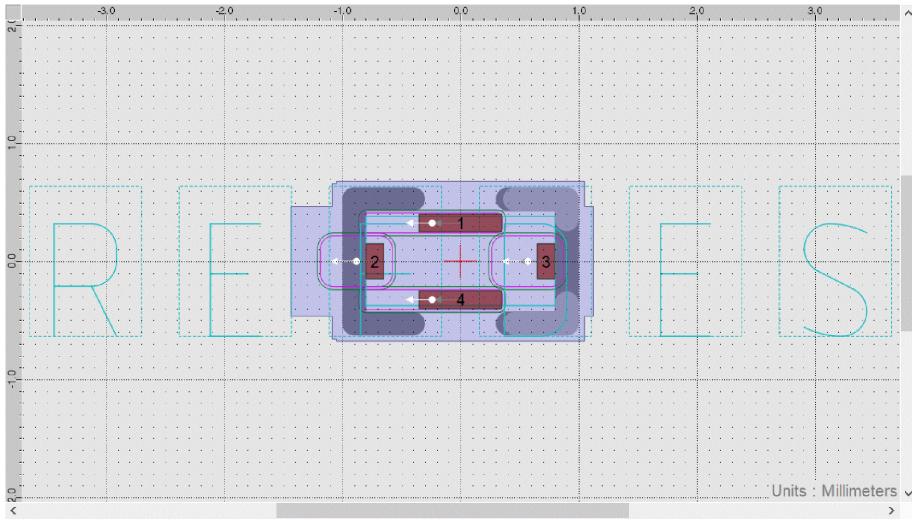
It is important that you save a new version of the footprint before exporting it to PCB Editor.

For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

Applying Rules

In this section, you will apply the Training/IPC-B (Design Technology) rule to create the footprint.

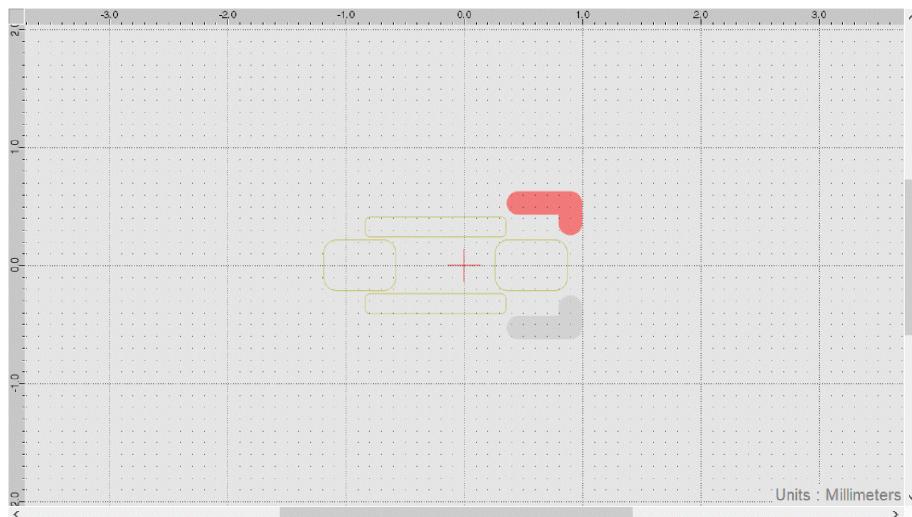
For more information about applying rules, refer to the [Applying Rules to Template](#) section. The footprint matching the IPC-B rule is created.



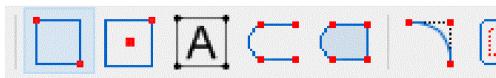
Adding Orientation Mark

When assembling the component to the board to match the correct pins of the component to the pads on the board, an orientation mark is required. Since this is a very small component, an orientation mark is required to be added outside of the part footprint near pin 4. To add an orientation mark:

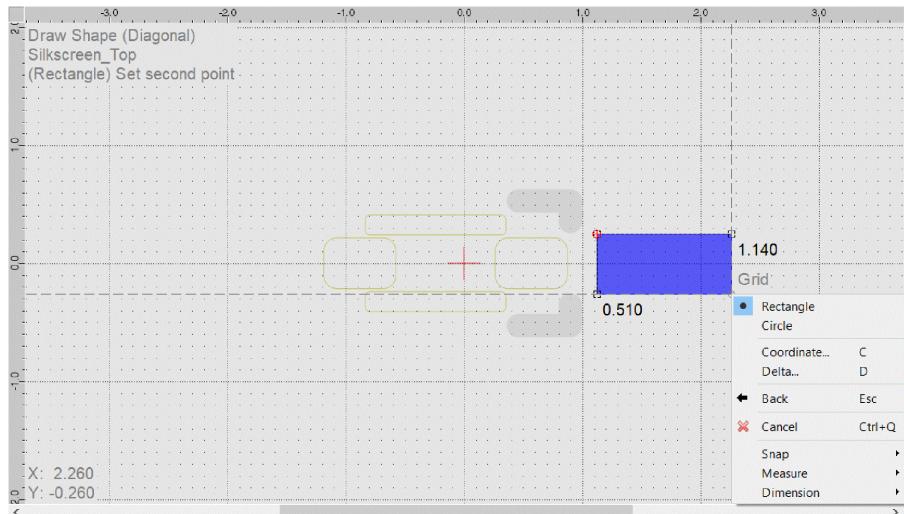
1. In the *Layer Control* panel, right-click and choose *Hide - All* to turn off all the layers.
2. Select the *Silkscreen_Top* and *Etch_Top* check boxes in the *Layer Control* panel.
3. Click anyone of the white silkscreen objects on the 2D canvas.



4. Click the *Add a new standard shape object* icon on the toolbar.

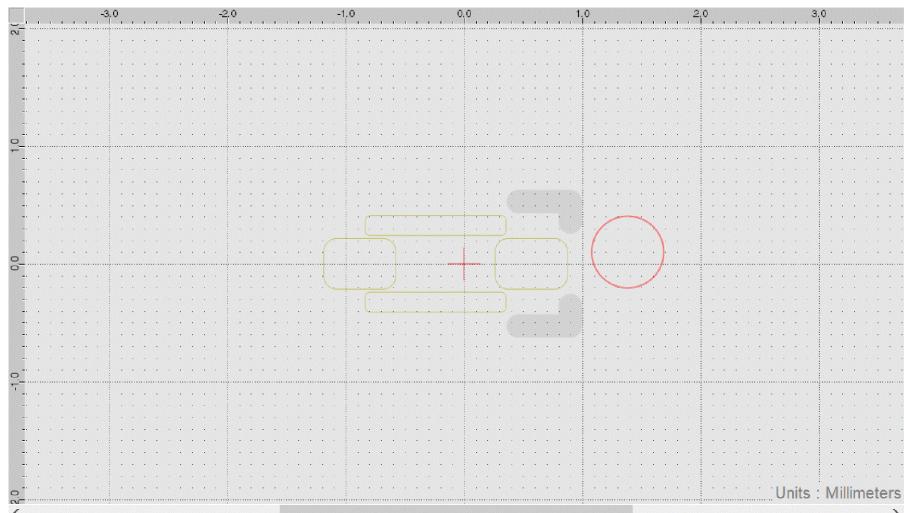


5. Click once on the right of Pin 4 and move the mouse to create a shape.



6. Right-click to choose *Circle*.

7. Click to complete creating a circle.

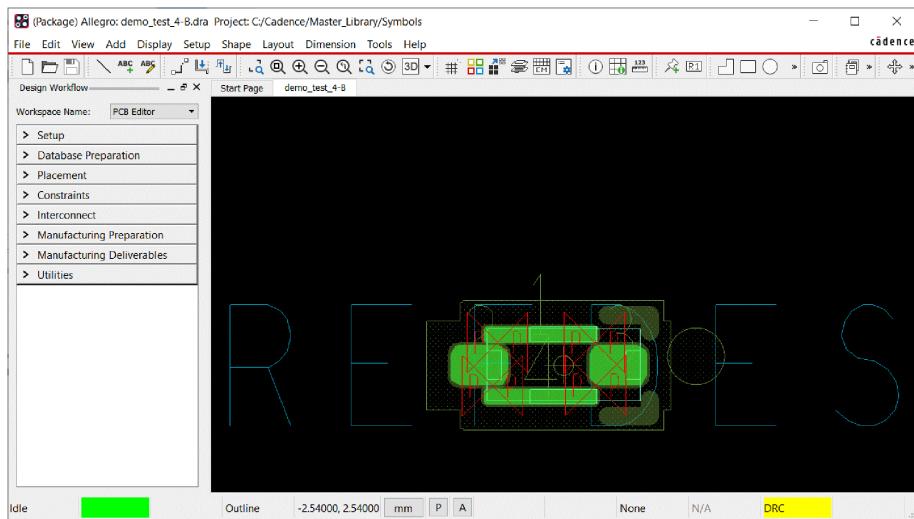


8. In the *Layer Control* panel, right-click to choose *Show - All* to turn on all the layers again.

Exporting Footprint to Allegro PCB Editor

Export this footprint to Allegro PCB Editor. Notice the orientation mark on the silkscreen layer as shown in the following figure.

For more information about exporting the footprint to Allegro PCB Editor, follow the steps mentioned in the [Exporting the Footprint to Allegro PCB Editor](#) section.



Creating Footprints from Scratch

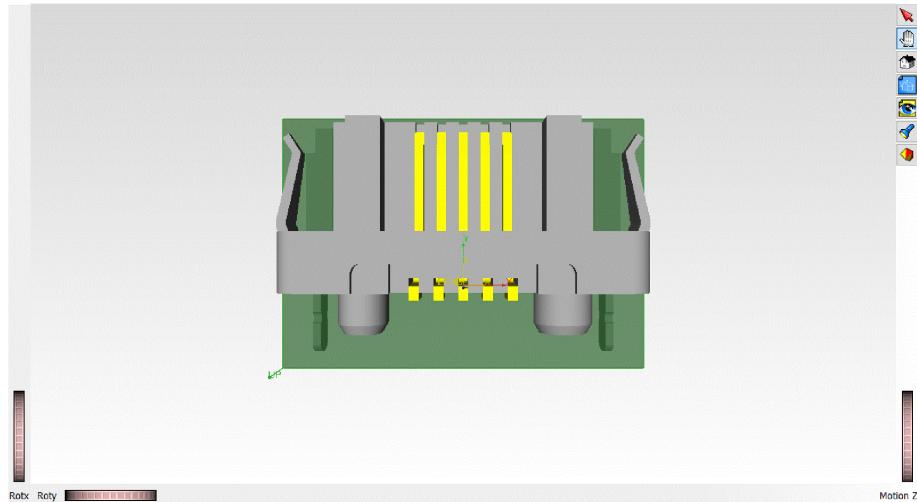
What You Will Learn

In this module, you will create the necessary footprint STEP model manually.

Importing a Step Model

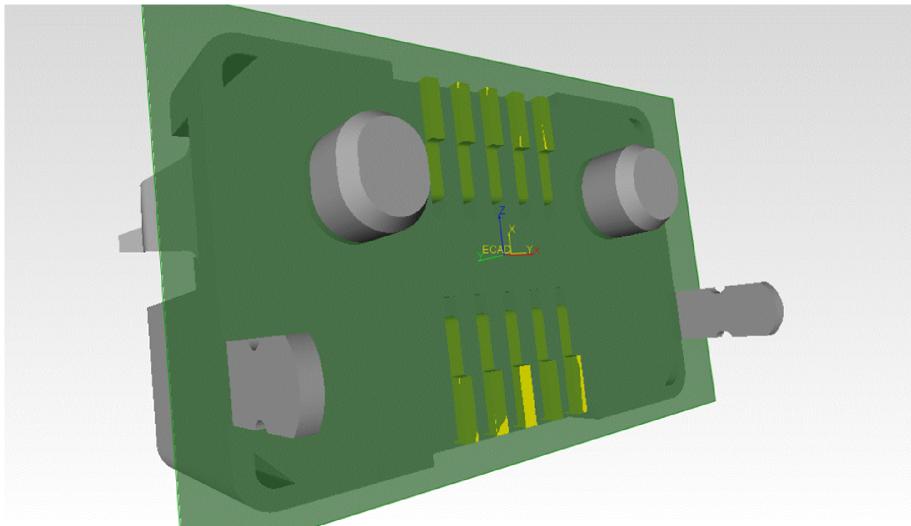
In this section, you will import the step model *LSHM-105-02.5-L-DV-A-S-TR.stp* from `<your_install_dir>/doc/lc_tut/tutorial_examples/STEP_models`.

For more information about importing a STEP model, follow the steps mentioned in the [Importing STEP Models](#) section.



Editing Seating Plane

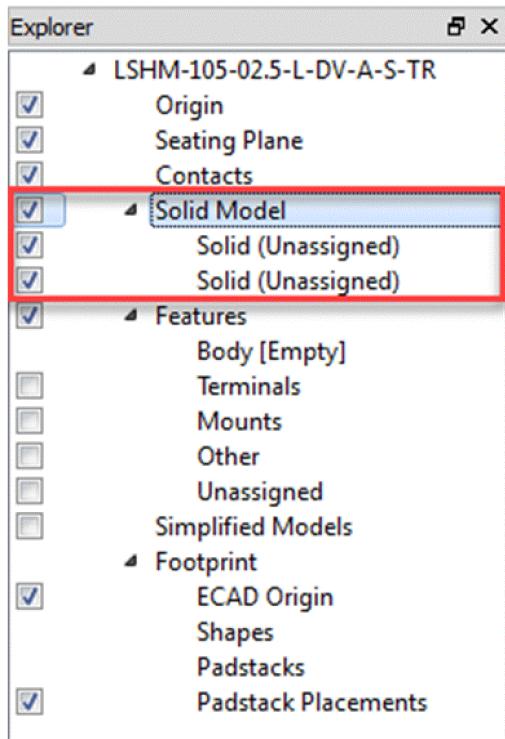
In this section, you will change the Seating Plane so that it is referenced to the bottom of the gold leads as shown in the following figure.



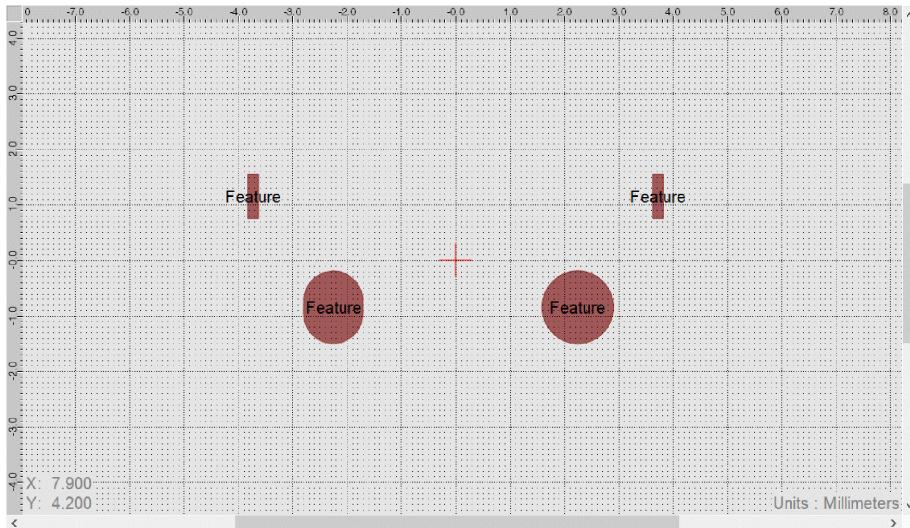
For more information editing Seating Pane, follow the steps mentioned in the [Editing Seating Pane](#) section.

Creating Holes

1. In the *Explorer* plane, expand the *Solid Model* node to see that there are two Solids – one for the body and one for the pins.



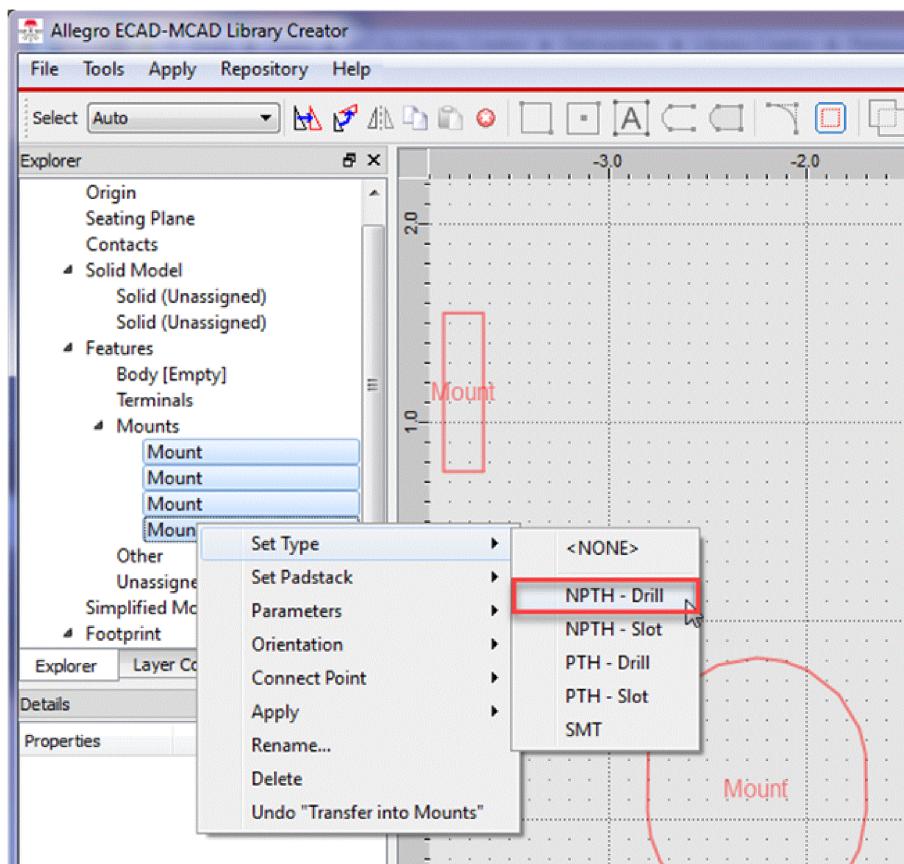
2. Right-click the *Solid Model - Solid* entity that is the body and choose *Contact Features - Training/THT Features* to locate and create the holes for this component.
3. Click *Footprint (2D)* view
4. Right-click *ECAD Origin* and choose *Change ECAD Origin*.
5. Right-click anywhere on the canvas and choose *Rotate Counterclockwise*.
6. Click *Accept*.



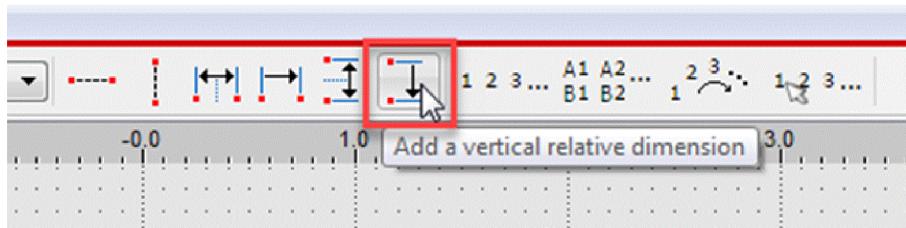
7. Select and drag the *Unassigned - Group* and the *Feature* nodes, and drop to the *Mounts* node.

For this tutorial, assume that all four pins are plastic and therefore the holes can be unplated.

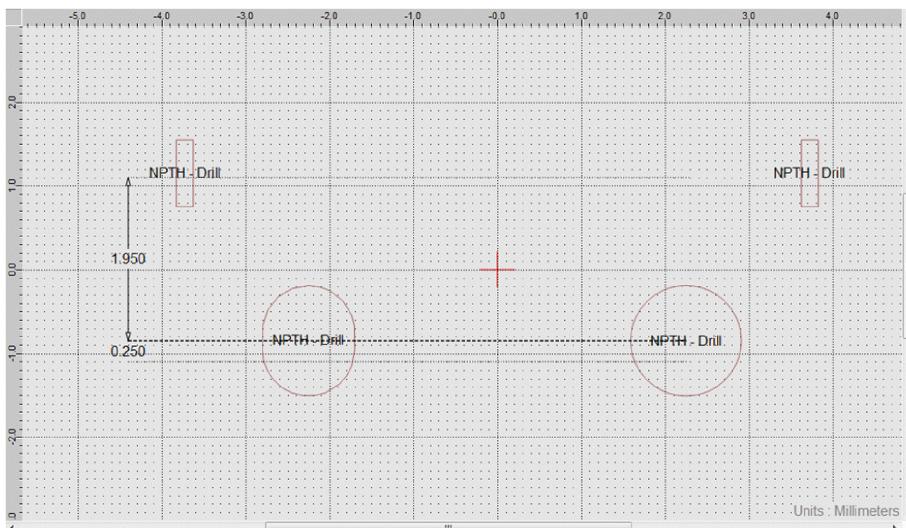
8. Right-click all four items under the *Mounts* node and choose *Set Type - NTPH – Drill*.



9. Select the *Add a vertical relative dimension* icon to add reference dimensions for easy vertical location points.



10. Using the center point of the bottom right hand corner circular Mount hole, enter a dimension from the center of the circle up *1.95 mm* and another one of *0.25 mm*.



11. In the *Explorer* pane, right-click *Terminals* and choose *New - Padstack Placements - Import*.
12. Select the padstack *SM_300X1500* from the drop-down list.

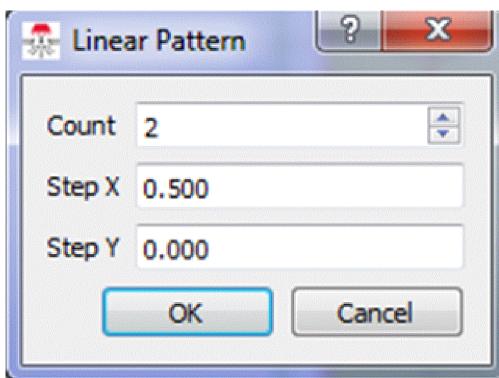
⚠ This padstack *SM_300X1500* must be added the folder specified in the Padstacks section of the *Allegro Settings* dialog.



1. Click *OK*.

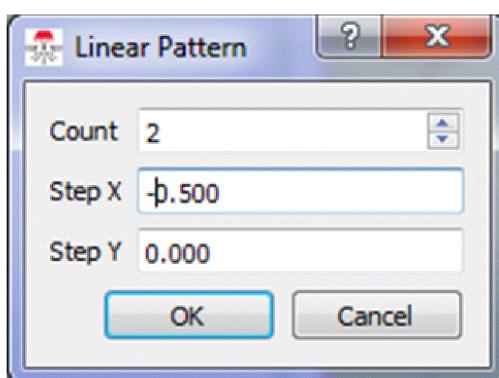
The newly imported padstack is attached to the cursor.

2. Drop the padstack on the center of the footprint.
Another padstack is attached to the cursor.
3. Right-click and choose *Accept*.
4. Right-click the newly added pin and choose *Pattern Copy - Linear Pattern*.
The *Linear Pattern* dialog appears.
5. Change the value of *Count* to *2*.
6. Change the value of *Step X* to *0.500*.
7. Click *OK*.

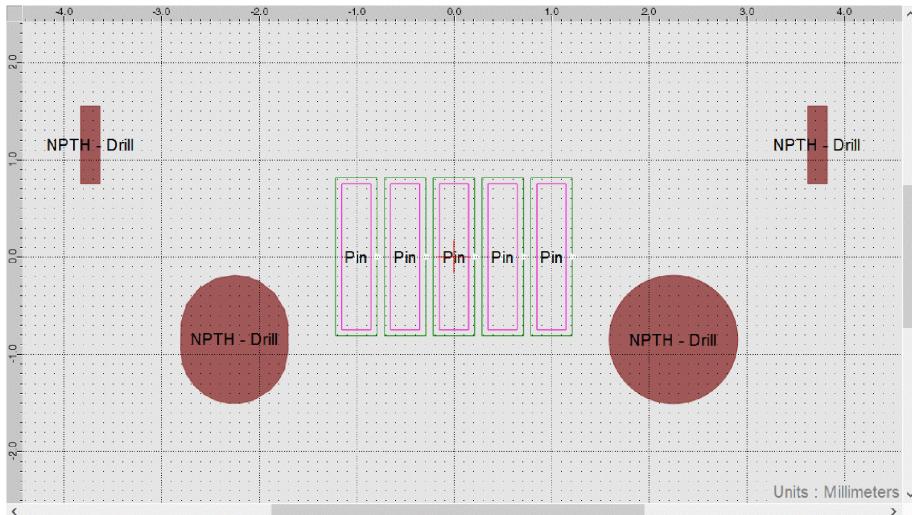


This adds two additional pins to the right of the first pin that was added.

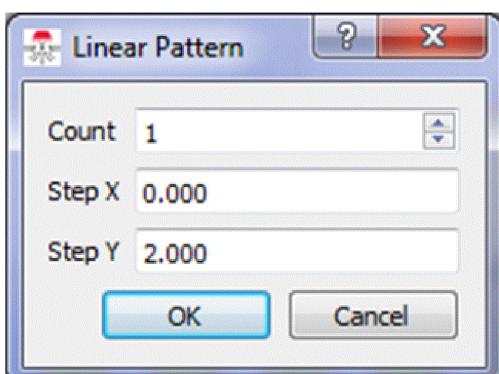
8. Right-click the newly added pin and choose *Pattern Copy - Linear Pattern*.
The *Linear Pattern* dialog appears.
9. Change the value of *Count* to *2*.
10. Change the value of *Step X* to *-0.500*.
11. Click *OK*.



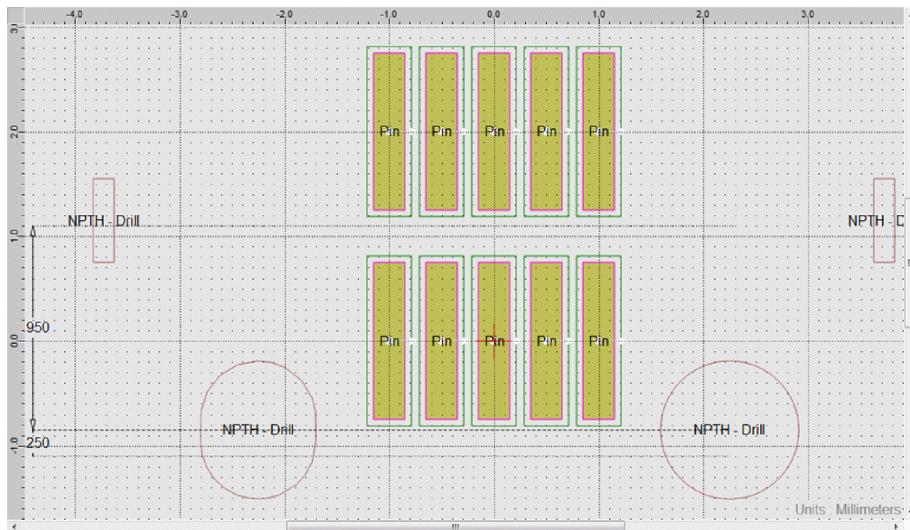
This adds two additional pins to the left of the first pin that was added.



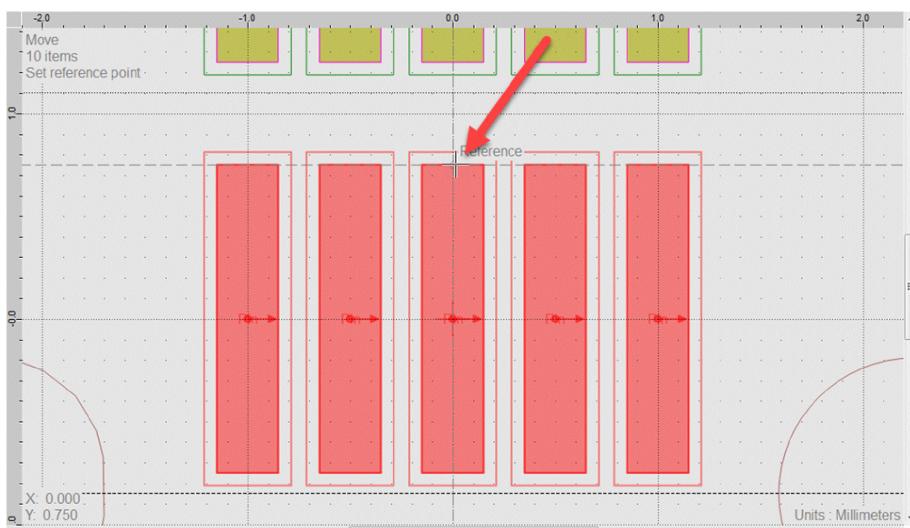
12. Select all five pins.
13. Right-click the selection and choose *Pattern Copy - Linear Pattern*.
14. In the *Linear Pattern* dialog, change the value of:
 - a. *Count* to 1.
 - b. *Step X* to 0.000.
 - c. *Step Y* to 2.000.
 - d. Click *OK*.



There add five pins on the canvas as shown in the following figure.

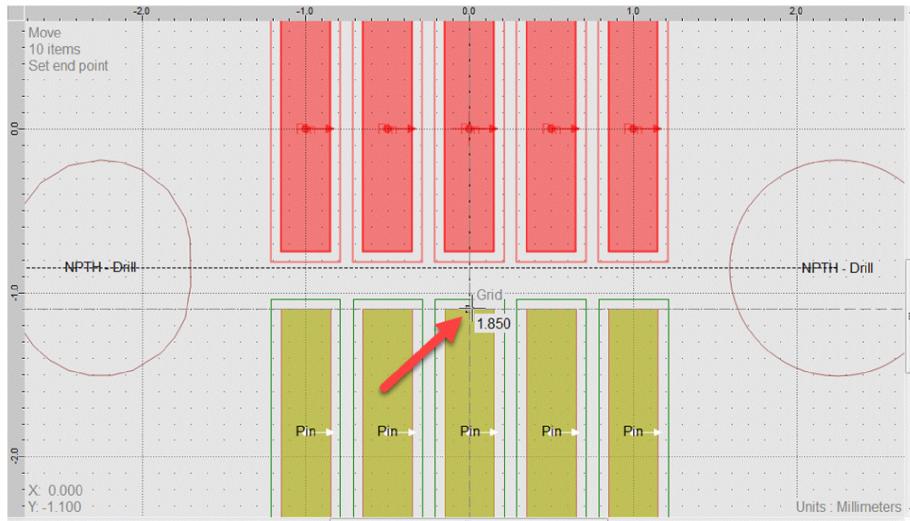


15. Select the lower five pins.
16. Right-click the selection and choose *Move*.
17. Move the cursor so that it locates the mid-point of the top edge of the center pin and click to secure the location.

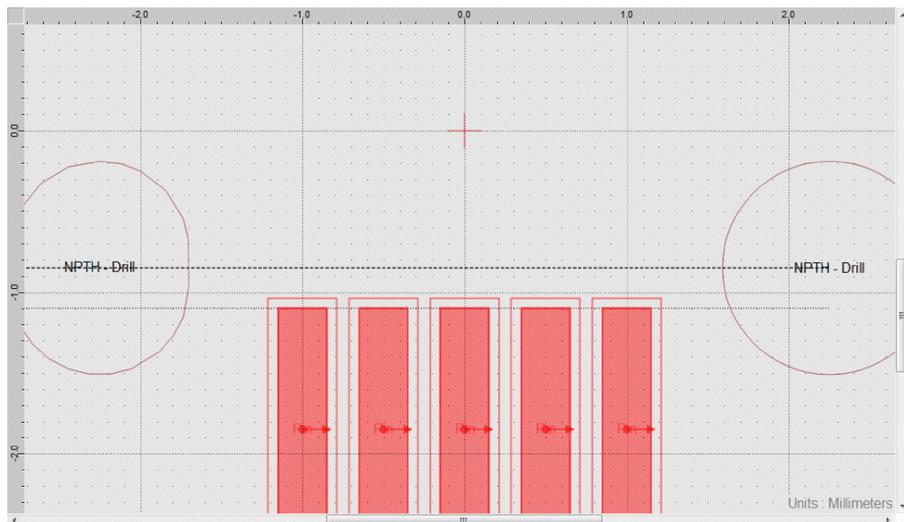


All five pins will now be on the cursor and move with the cursor.

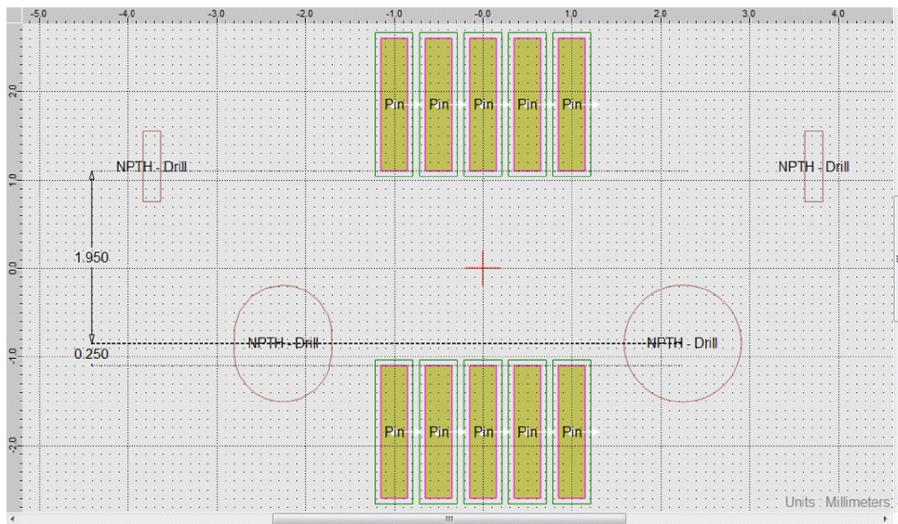
18. Move the cursor and locate the center of the lower of the two reference dimension lines that were added previously.



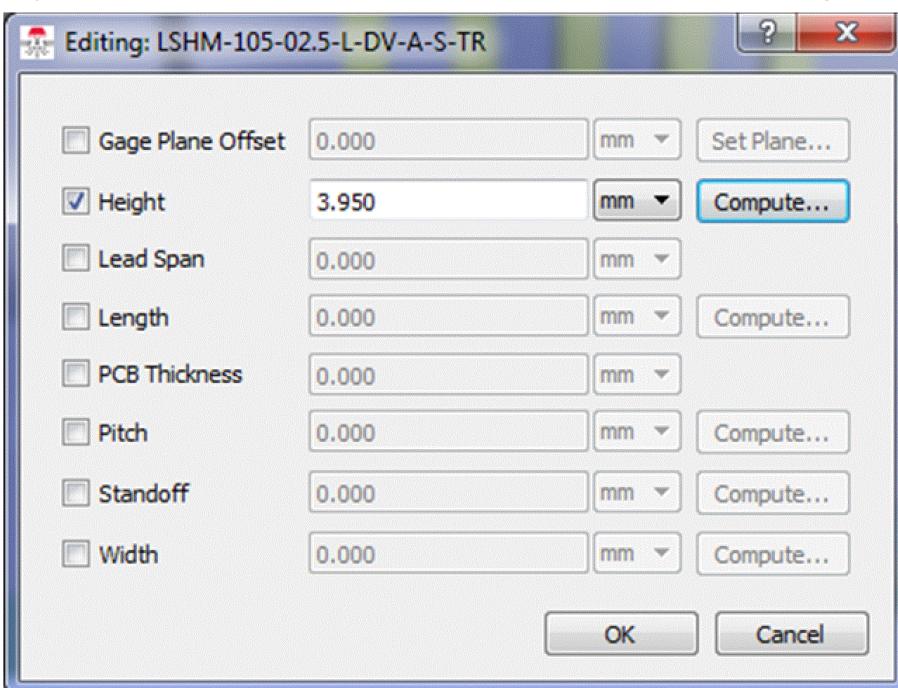
Five pins are now centered horizontally and the upper edge are located along the reference dimension as shown in the following figure.



19. Repeat the same steps for the upper five pins and move the bottom edge of the pins to the upper reference dimension added previously.



20. In the *Explorer* pane, select *Features - Terminals - Group*.
21. Right-click the selection and choose *Set Leaf Form - Flat Lead*.
22. Select *Package (3D)* view.
23. In the *Explorer* pane, select and drag *Solid Model - Solid (Unassigned)* that is the body and drop it to the *Features - Body (Empty)* node.
24. Right-click *LSHM-105-02.5-L-DV-A-S-TR* in the *Explorer* pane and choose *Parameters - Edit*.



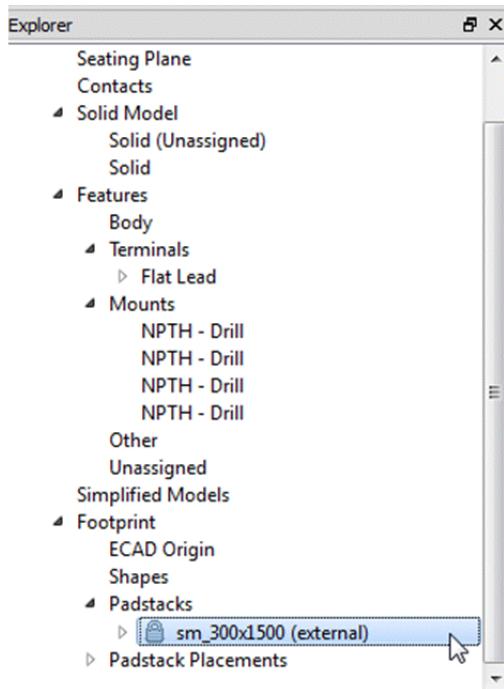
25. Select the *Height* check box and choose compute.

26. Click *OK*.

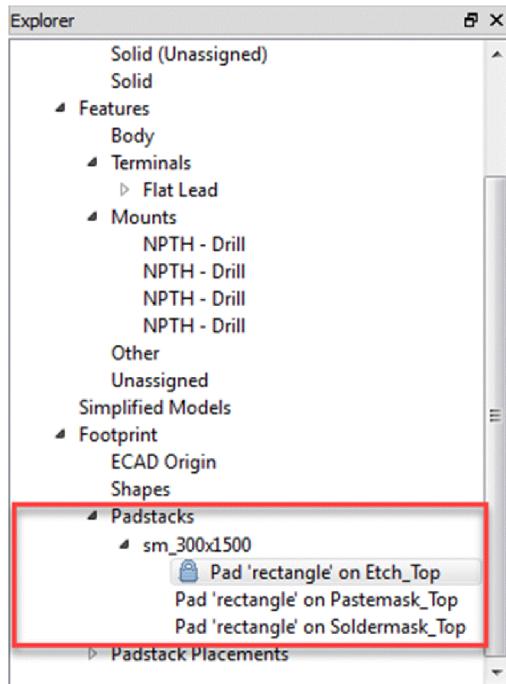
Saving the Footprint

For more information about saving the footprint, follow the steps mentioned in the [Saving the Footprint](#) section.

The imported padstack *SM_300X1500* in the *Explorer* pane is locked and displayed as (*external*). This means that it was from another padstack library if you apply any rules, this padstack will not be updated.



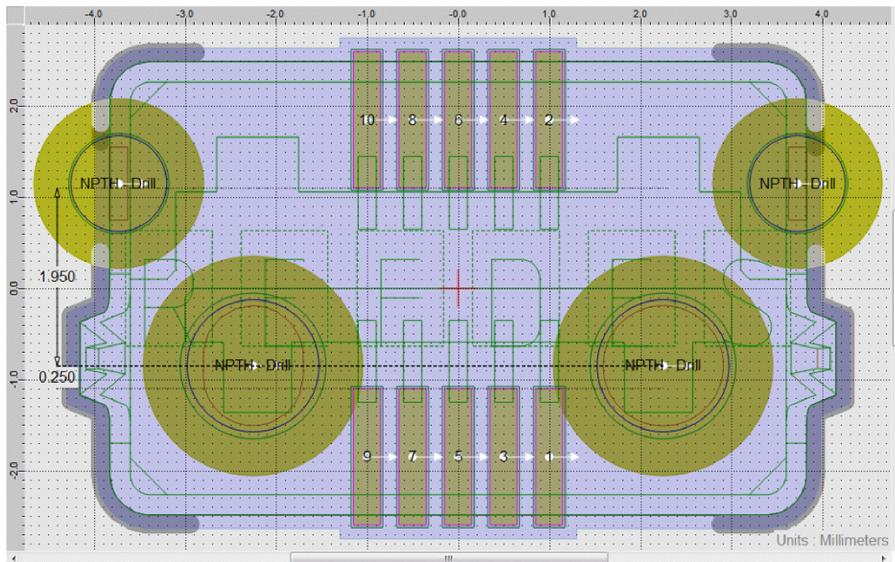
You can unlock import padstacks from the State right-click menu. The options include *Transient*, *Persistent*, *Locked*, and *External*. You can change the state to any option that suits their requirement, for this tutorial, we will select *Transient* and lock only the *Etch_Top* pad.



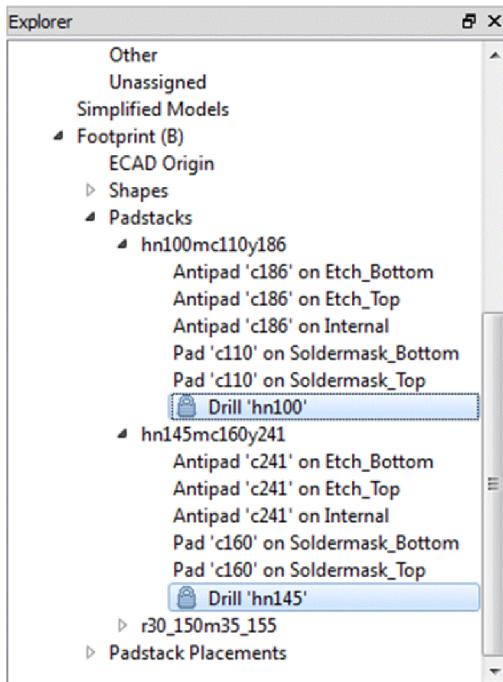
Applying Rules to Package

To apply rules:

1. Apply the *Training/IPC-B* rule to complete the footprint.



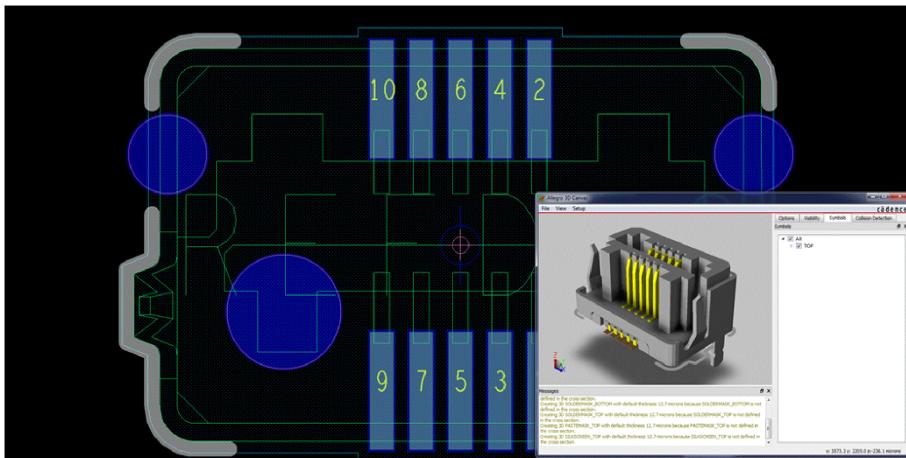
2. Update and lock the Drill sizes for the two sets of unplated holes. The upper holes will be 1.00 mm and the lower holes will be 1.45 mm.



3. Re-apply the *Training/IPC-B* rule so that the padstacks will update according to the newly locked drill sizes.

Exporting Footprint to Allegro PCB Editor

To export the footprint to Allegro PCB Editor, refer to the steps mentioned in the [Exporting the Footprint to Allegro PCB Editor](#) section.



Summary

This concludes this lab. In this module, you successfully utilized a manual method to create complex footprints. Library Creator gives users many options – auto or manual – to create footprints.

Changing Package Colors

Library Creator packages included in the repository or created from templates, have separate default colors for the body (blackish gray) and the pins (silver). The default colors can be customized and you can also specify a custom color for the Primary Pin.

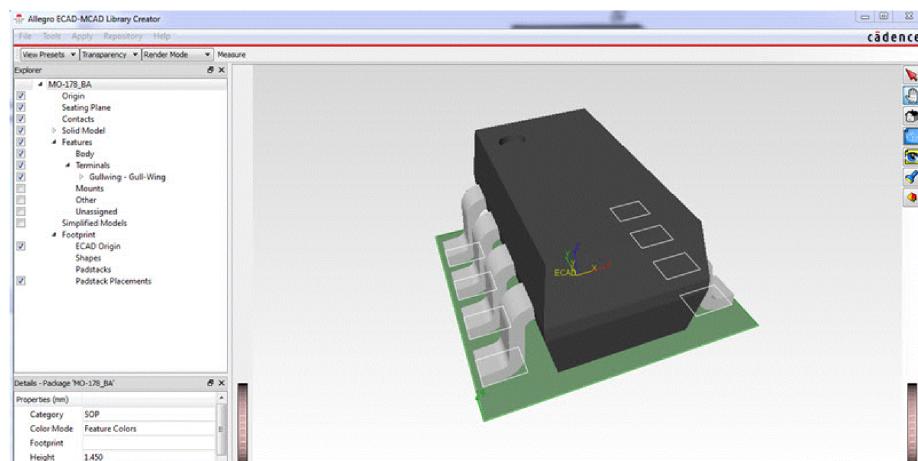
What You Will Learn

In this module, you will learn how to specify a customer color for a Primary Pin of a package.

Loading Packages

In this section, you will search and load the *MO-178_BA* package. The first step in creating a footprint from a package is searching and loading a package from the repository.

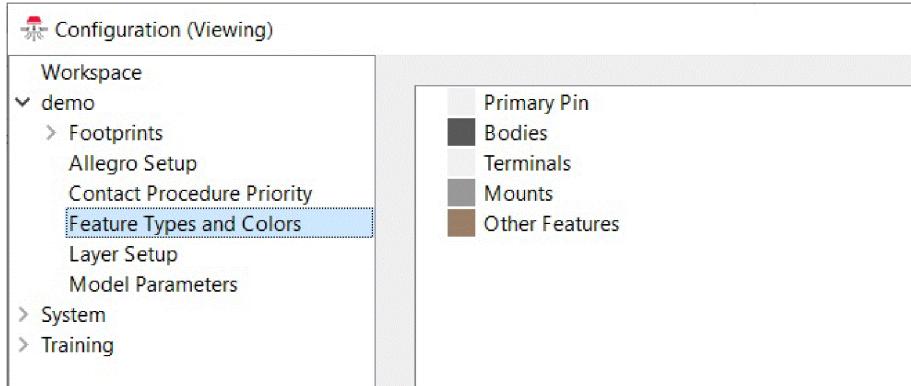
For more information about searching and loading a package, follow the steps mentioned in the [Loading Package](#) section.



Changing Primary Pin Color

1. Choose *Tools - Configuration*.
2. Click *Feature Type and Colors* under *demo*.

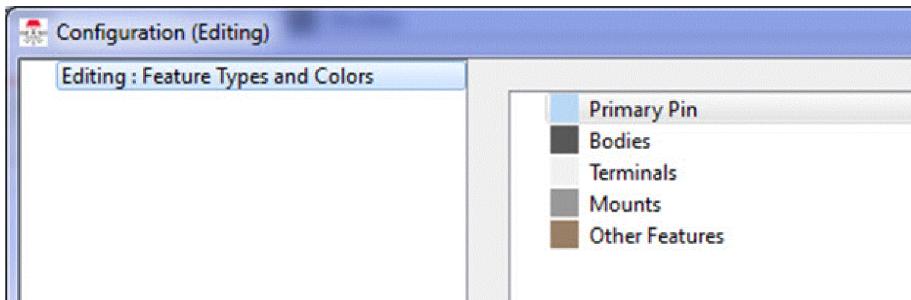
The default colors are displayed in the right pane.



⚠ Any changes made to *Feature Type and Colors* will be available to all users of this account in Library Creator because these are listed under *demo* account. Users having access to the *Configuration* area can edit the rules.

3. Right-click *Feature Types and Colors* and choose *Edit*.

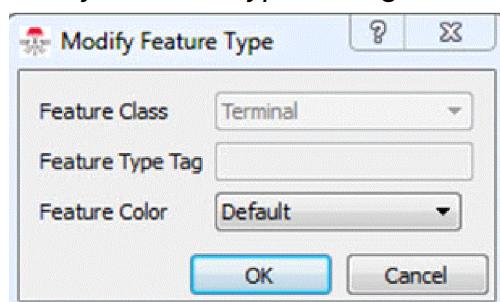
The *Configuration (Editing)* dialog is displayed.



Next, you will change the color of Primary Pin.

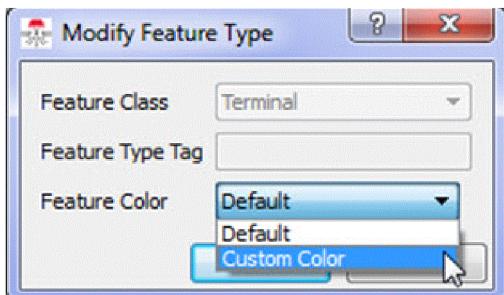
4. Right-click *Primary Pin* and choose *Modify*.

Modify Feature Type dialog box is displayed.

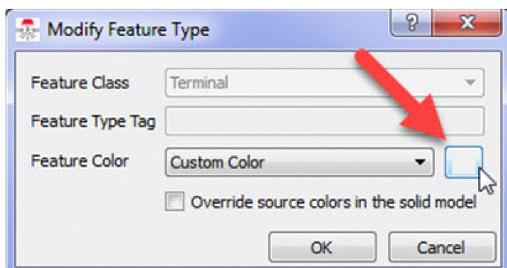


5. Select *Custom Color* from the *Feature Color* drop-down list.

The color box is displayed.

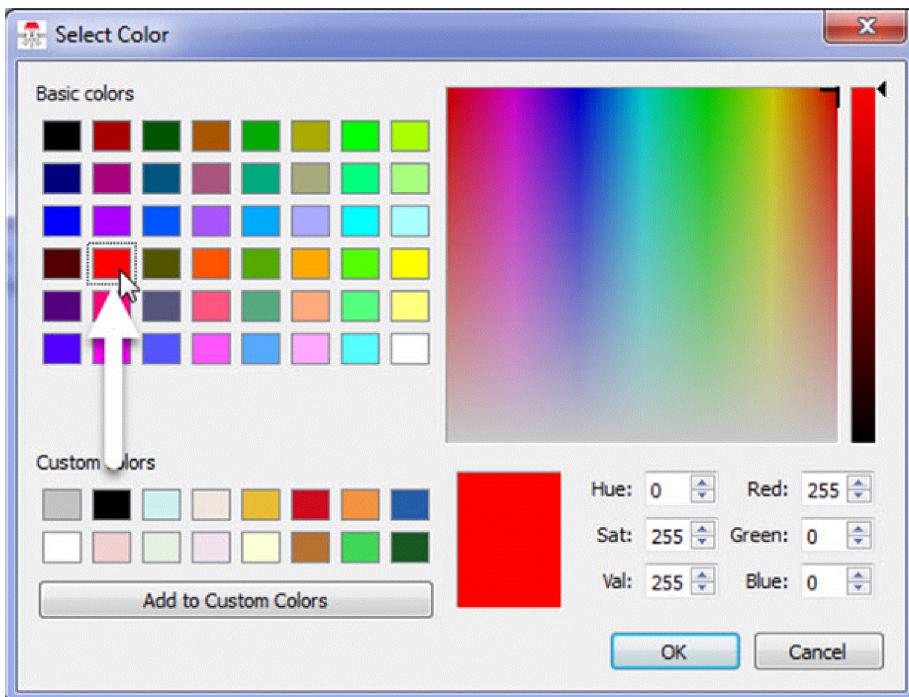


6. Click the color box.



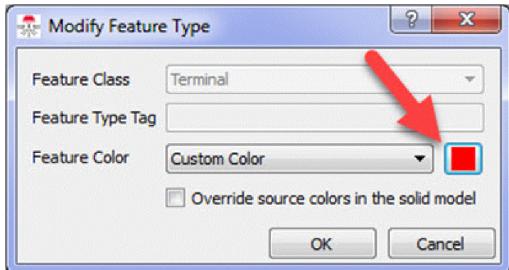
The Select Color dialog is displayed.

7. Select the color that has the *Red*, *Green*, and *Blue* values as 255, 0, and 0 respectively.

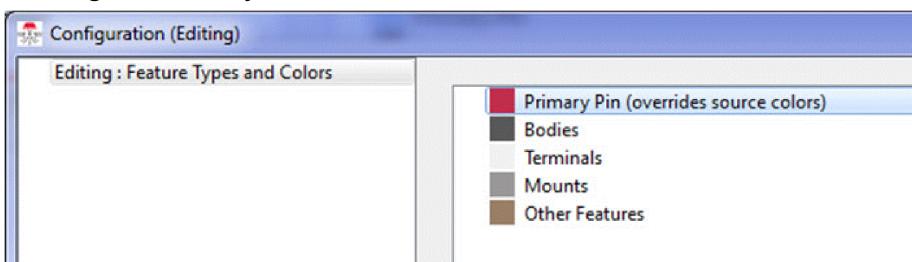


8. Click *OK*.

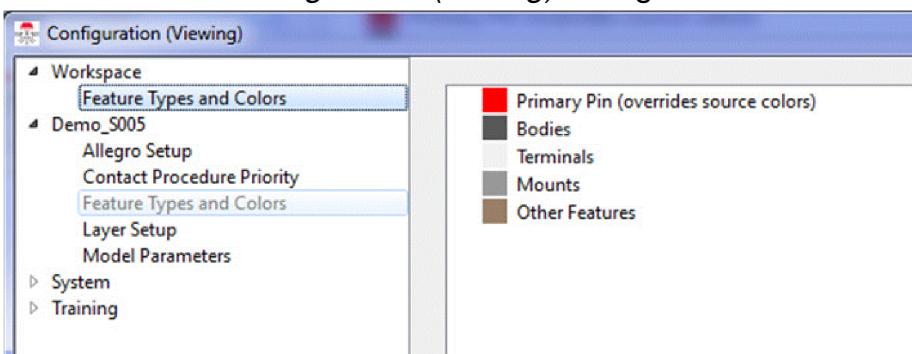
The selected color is displayed in the *Modify Feature Type* dialog box.



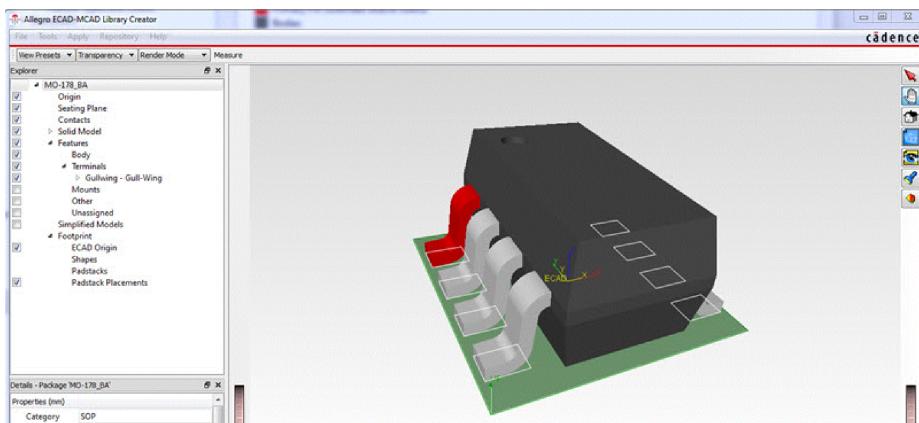
9. Select the *Override source colors in the solid model* check box to ensure that the chosen custom color is the color of the primary pin.
10. Click *OK*.
The *Configuration (Editing)* dialog shows the new color for the primary pin and indicates that the chosen color will override *Source Colors*. This means that the primary pin will always be red regardless if you choose *Default* or *Custom Color* for the model.



11. Click *OK* in the *Configuration (Editing)* dialog.

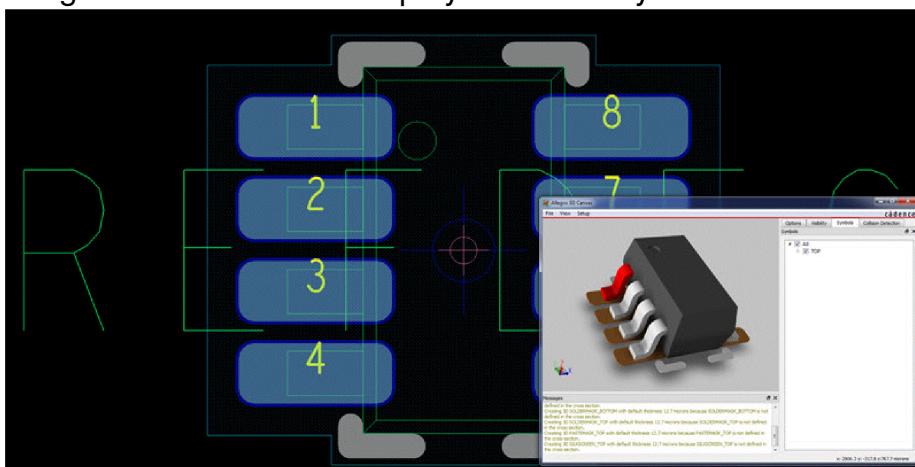


In the *Package (3D)* view, Pin 1, the primary pin is now red. With this change, the primary pin will be red for any existing packages loaded in from the repository or for any new packages created from Templates.



⚠ The above change in the primary pin color is only visible to the current logged in account and displayed in the *Workspace* copy. To make this change available for other users as well, right-click *Workspace* and choose *Publish Workspace*.

The color change to the primary pin is locked in with the *Package*. If a rule is applied and the footprint, padstacks and STEP models are exported to Allegro PCB Editor, the STEP model in Allegro PCB Editor also displays the Primary Pin as red.

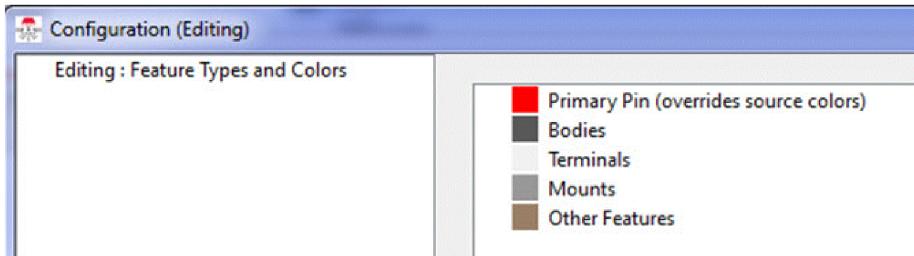


Changing Body Color

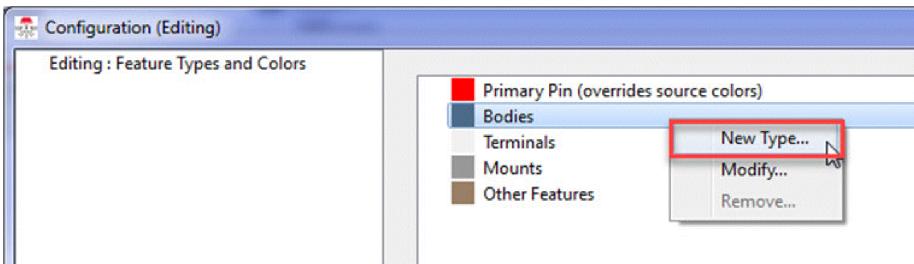
Library Creator also allows you to add sub-types of *Feature Types and Colors*. In this section, you will add and assign colors to body sub-types Ceramic, Plastic, and Metal.

1. Choose *Tools - Configuration*.
2. Right-click *Feature Types and Colors* and choose *Edit* under *Workspace* in the *Configuration (Viewing)* dialog.

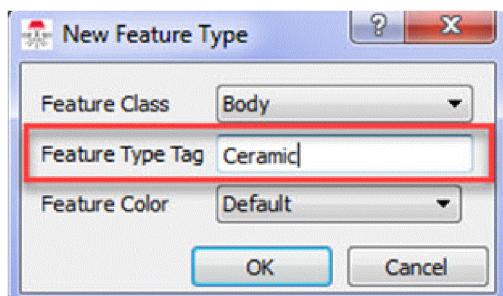
The *Configuration (Editing)* dialog is displayed.



3. Right-click *Bodies* and choose *New Type* to add new sub-types.
The *New Feature Type* dialog box is displayed.

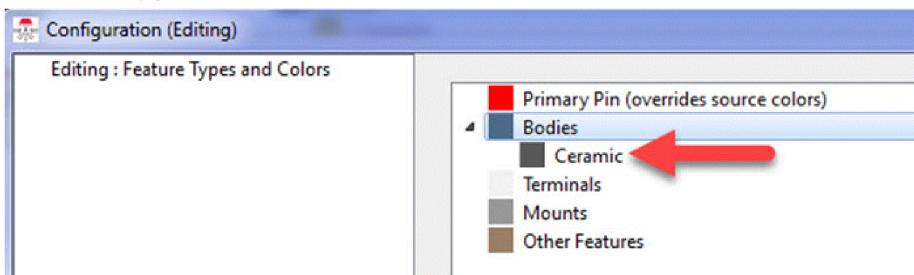


4. Type *Ceramic* in the *Feature Type Tag* field.



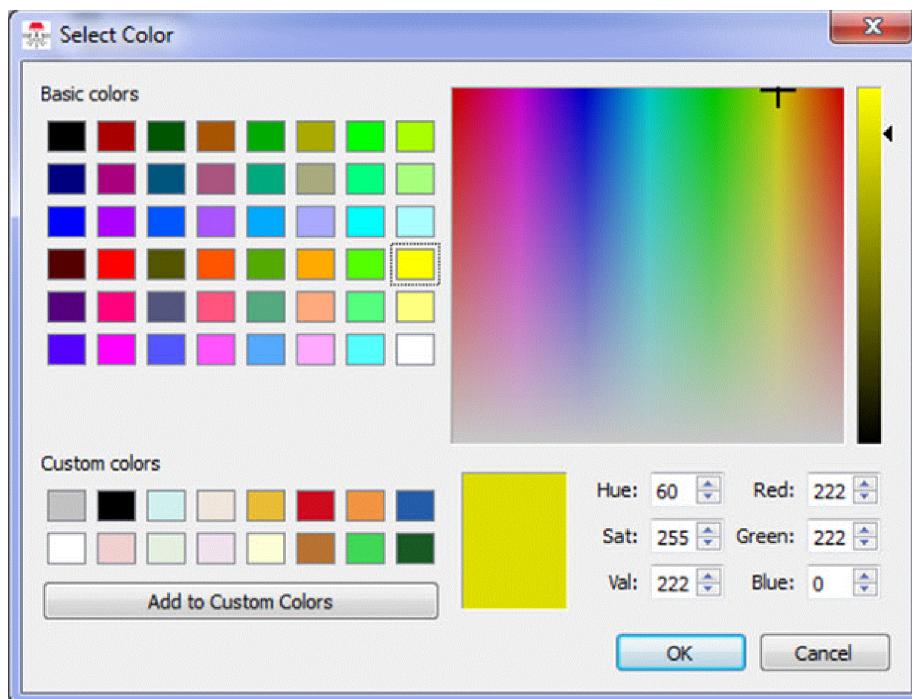
5. Click *OK*.

The sub-type *Ceramic* is now listed under *Bodies*.

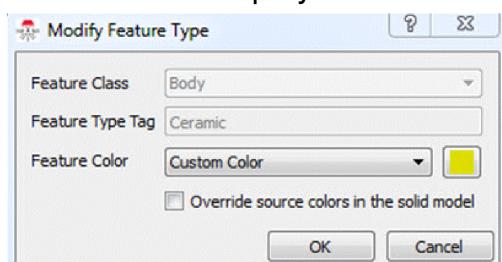


6. Double-click *Ceramic*.
7. Select *Custom Color* from the *Feature Color* drop-down list.
The color box is displayed.
8. Click the color box.

9. Type 222, 222, 0 in the *Red*, *Green*, and *Blue* fields respectively.
10. Click *Add to Custom Colors*.
11. Click *OK* in the *Select Color* dialog.

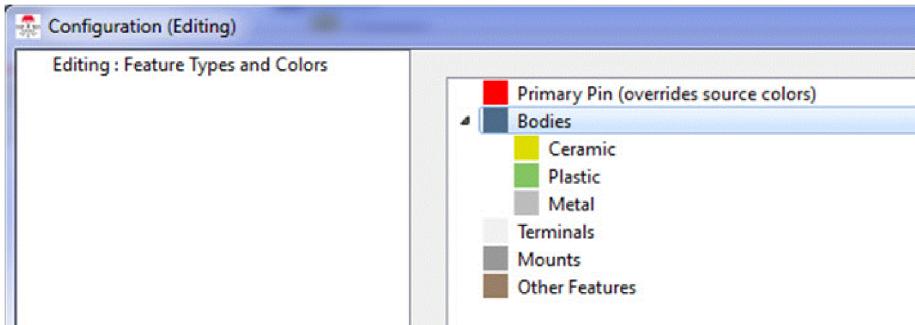


12. Click *OK* in the *Modify Feature Type* dialog.
The color box displays the new color.

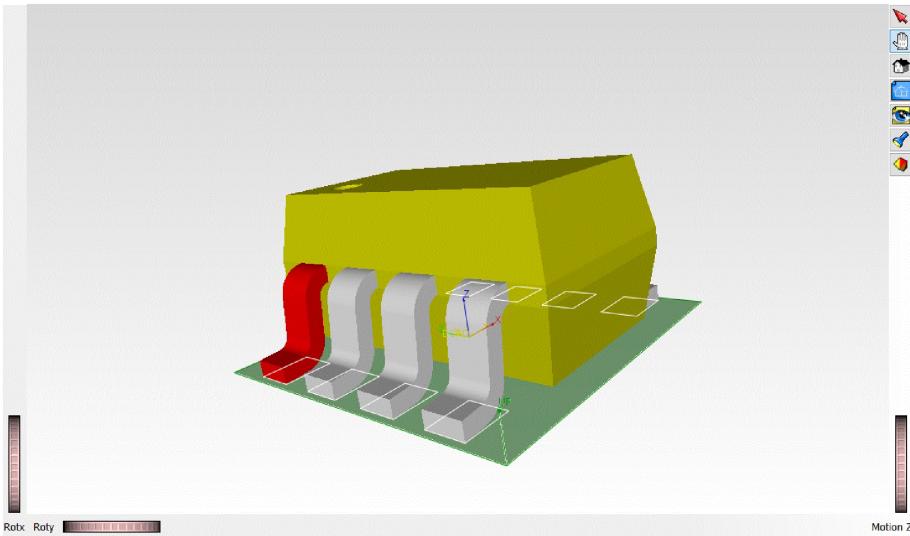


The *Override source colors in the solid model* is not selected. This means that the body color will only change to the new specified color when *Custom Colors* is chosen and *Ceramic* bodies are specified. The body color when using *Default* will remain the color specified by the vendor.

13. Using the step 1 to step 12, add two body sub-types – *Plastic* and *Metal*.

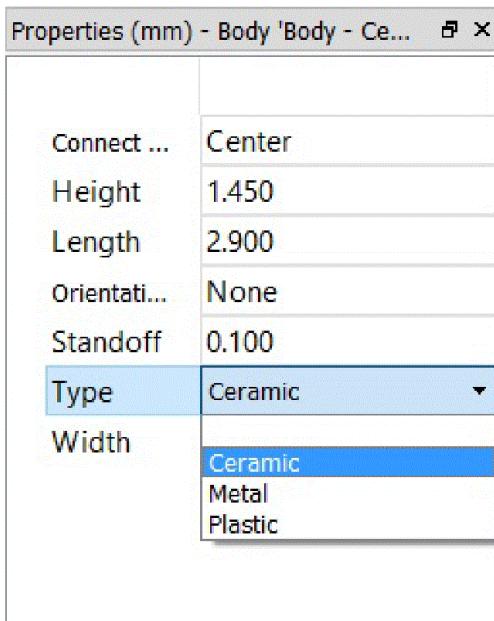


14. Click *OK* in the *Configuration (Editing)* dialog.
15. In the *Explorer* pane, right-click *Features - Body* and choose *Set Type - Ceramic* to check the package changing color specified for *Ceramic* in 3D view.

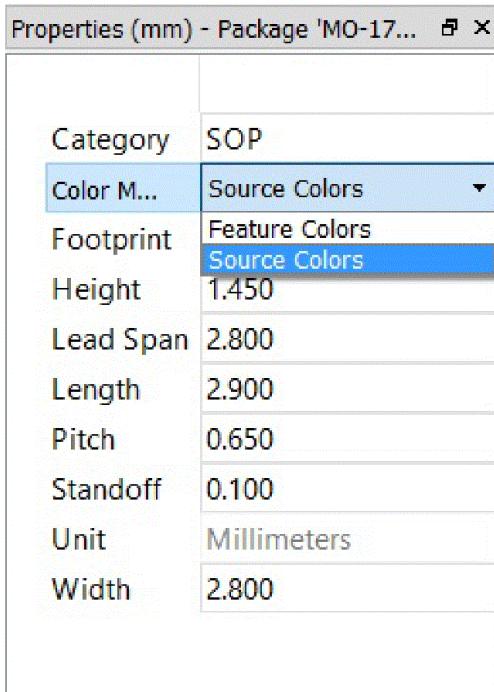


Similarly, you can check the colors for the other two body types.

Alternatively, Body type can also be set in the *Properties* panel. Select *Body* in the *Explorer* pane and double-click the value of the *Type* field to choose the Body types.

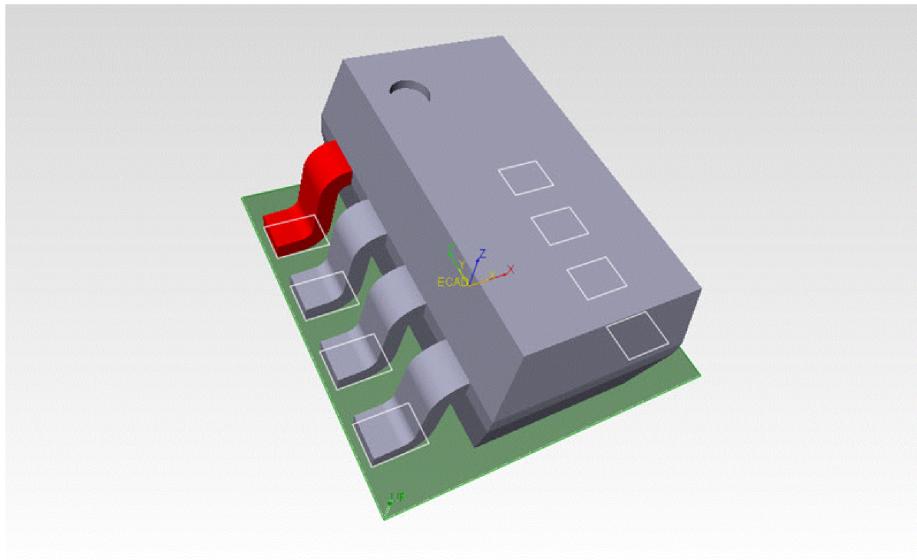


16. Right-click *Body* - Set Type and choose <NONE>.
17. Select *MO-178_BA* in *Explorer*.
18. Double-click *Color Mode* in the *Properties* panel and choose *Source Colors*.



19. Click anywhere on the canvas for the selection to be accepted.
The colors change to gray for all the parts of the package except for Pin 1 (the Primary pin)

because you selected *Override source colors in the solid model*.



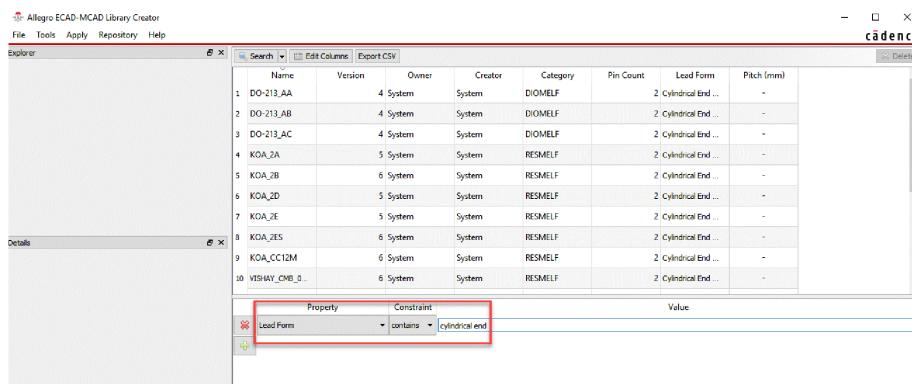
Creating a New Library Using Bulk Export

What You Will Learn

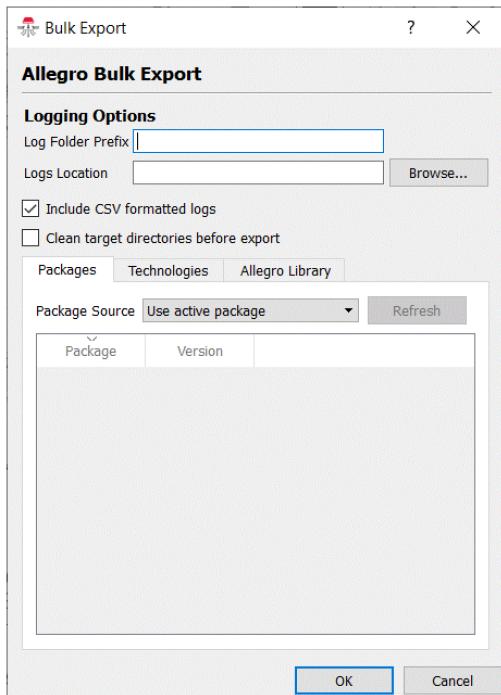
In this module, you will learn how to create a new library using existing packages and a different rule set. You will use a sub-set of the packages in the repository, but, you can perform the same exercise with all the packages in the repository, thereby creating new footprints, padstack and models for the entire library.

To create a new library using bulk export:

1. Select *Lead Form* from the *Property* drop-down list and *contains* from the *Constraint* drop-down list in the *Search tab*.
2. Type *cylindrical end* in *Value* and press *Enter*.

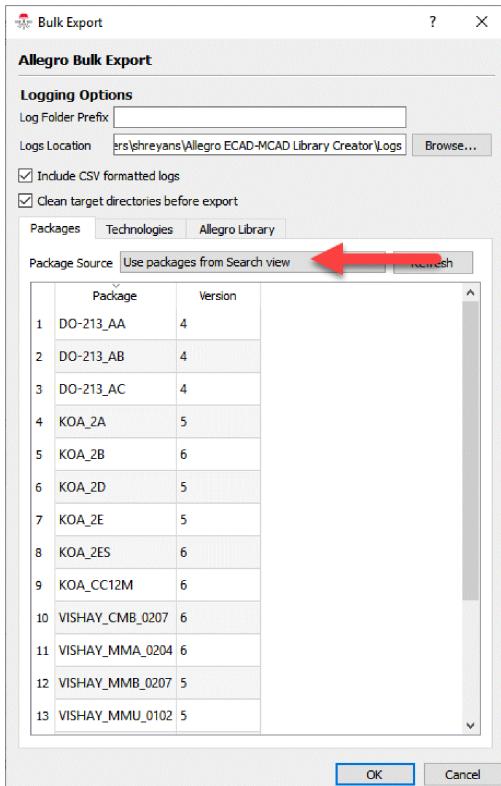


3. Choose Tools - Allegro Bulk Export.
Bulk Export dialog is displayed.

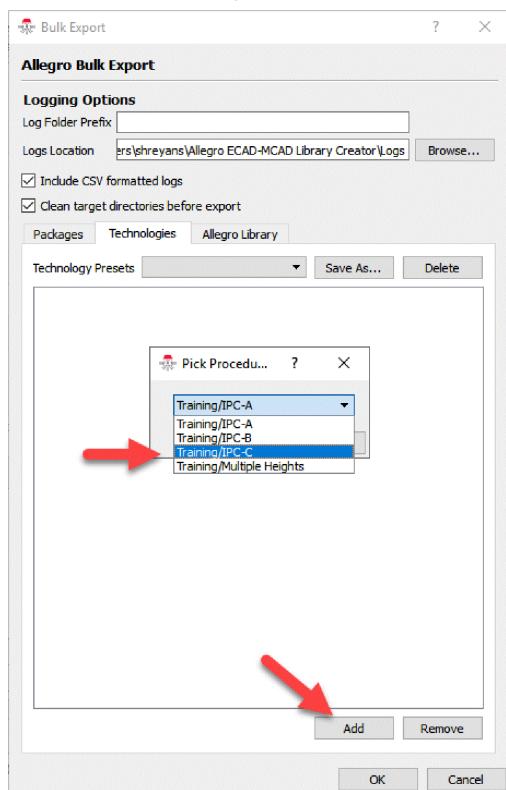


4. In the *Packages* tab, select *User packages from Search view* from the *Package Source* drop-down list.

Packages from the Search results are displayed.



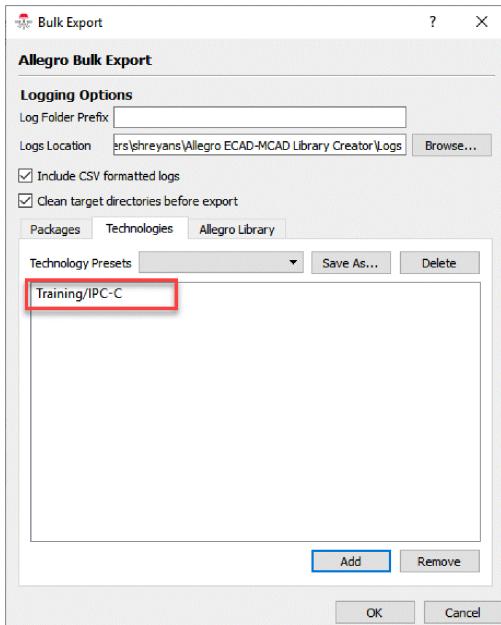
5. Select a valid folder in the *Logs Location* field.
 6. Select the *Include CSV formatted logs* and *Clean target directories before export* check boxes.
 7. Click the *Technologies* tab.
 8. Click *Add*.
- Pick Procedure* dialog box is displayed.
9. Choose *Training/IPC-C* from the drop-down list.



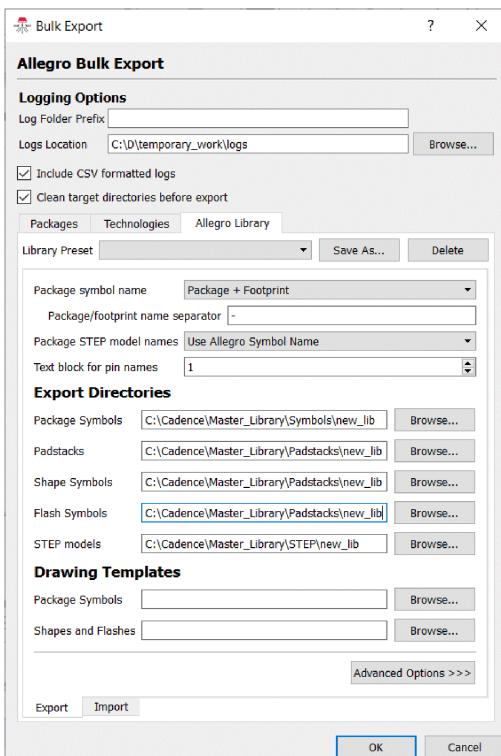
10. Click *OK*.
- The selected rule is now listed in the *Technologies* tab.

Allegro X ECAD-MCAD Library Creator Tutorial

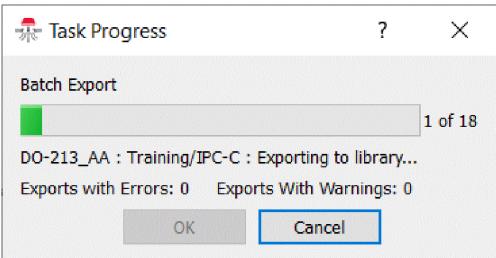
Creating a New Library Using Bulk Export--What You Will Learn



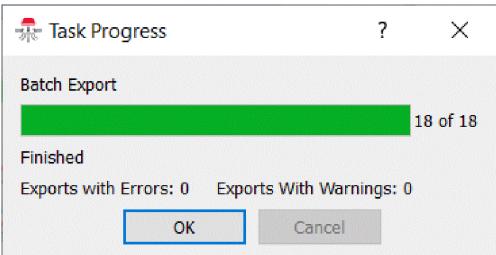
11. Click the *Allegro Library* tab.
12. Create new folders in the Master Library folders.
13. Under the *Export Directories* section, set the path in the *Padstacks*, *Shape Symbols*, *Flash Symbols*, and *STEP models* fields to the newly created folders.
Creating new folders allows you to differentiate the different types of libraries.



14. Click *Save As* in the *Library Preset* field.
Enter Name dialog box is displayed.
15. Type `Test_IPC_C` in the *Enter Name* dialog box.
16. Click *OK*.
Library Presets can be used to save different export folders for different rules.
17. Click *OK* in the *Bulk Export* dialog.
A confirmation message is displayed.
18. Click *Yes*.
The *Task Progress* dialog is displayed indicating the packages that are processed.



19. Click *OK* on the *Task Progress* dialog after all the packages are completed.



The newly created padstacks, footprints, and STEP models are stored in export paths selected in *Step 13*.

Summary

In this module, you used the Bulk Export feature of Library Creator to export a set of padstacks and create a new library.

Editing Existing Rules

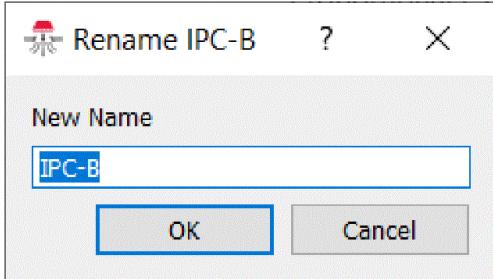
What You Will Learn

In this module, you will learn how to edit an existing rule in Library Creator by changing the values of *Variables* and *Padstack* rules. You will also learn to test and publish the newly edited rule.

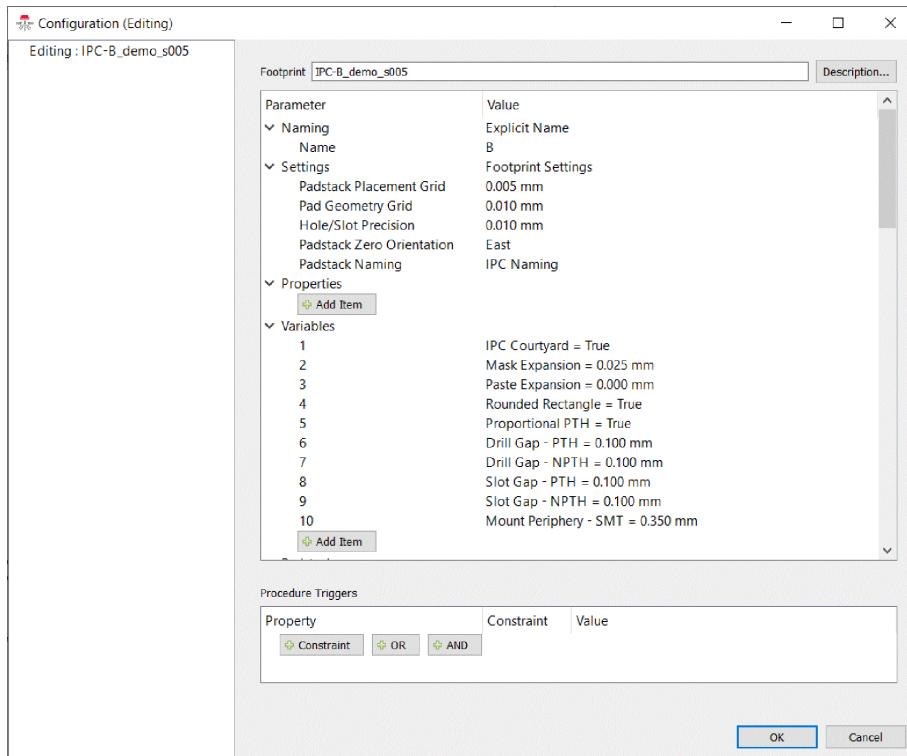
Creating Copy of Rules

To create a copy an existing rule:

1. Choose *Tools - Configuration*.
Configuration (Viewing) dialog is displayed.
2. Expand *Training - Footprints*.
3. Right-click *IPC-B* and choose *Copy*.
Rename IPC-B dialog is displayed.



4. Type `IPC-B_demo_s005` in the *New Name* filed and click *OK*.
Configuration (Editing) dialog appears with the details of the *IPC-B* rules in the *Footprint* section.

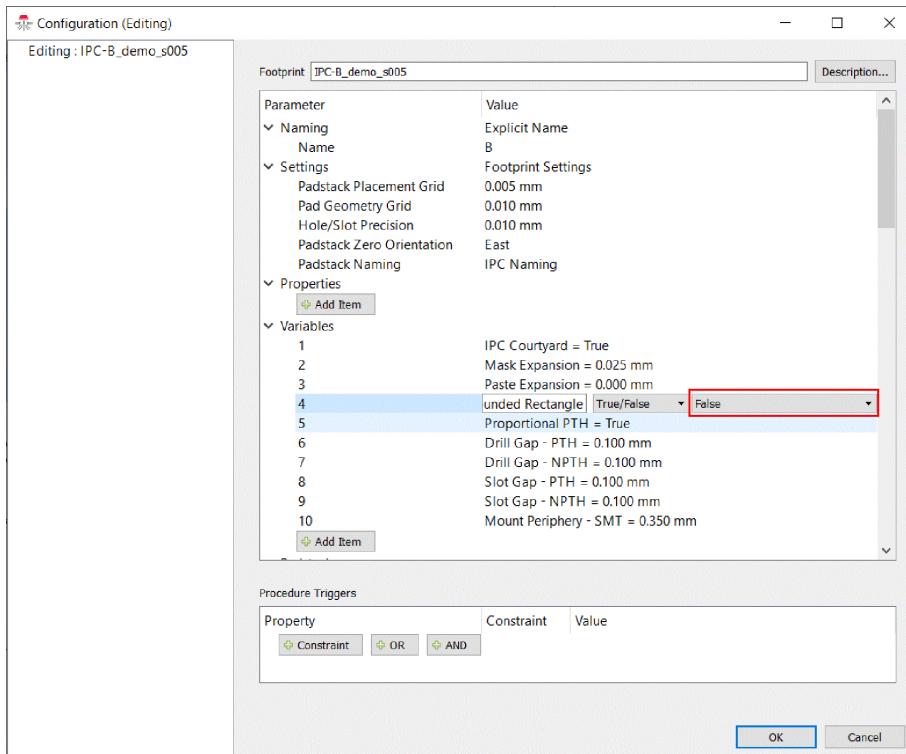


For more information about the different sections of this dialog, refer to the [Configuration](#) section of the Library Creator User Guide.

Editing Variable Rules

In this section, you will change the *Variable – Rounded Rectangle* from *True* to *False*.

1. Click the *Rounded Rectangle* field under the *Variables* section.
The field becomes editable.
2. Select *False* from the drop-down list.



3. Click *OK* to close the *Configuration (Editing)* dialog.

The rule you just edited is now saved under *Workspace - Footprints* section in the *Configuration (Viewing)* dialog.



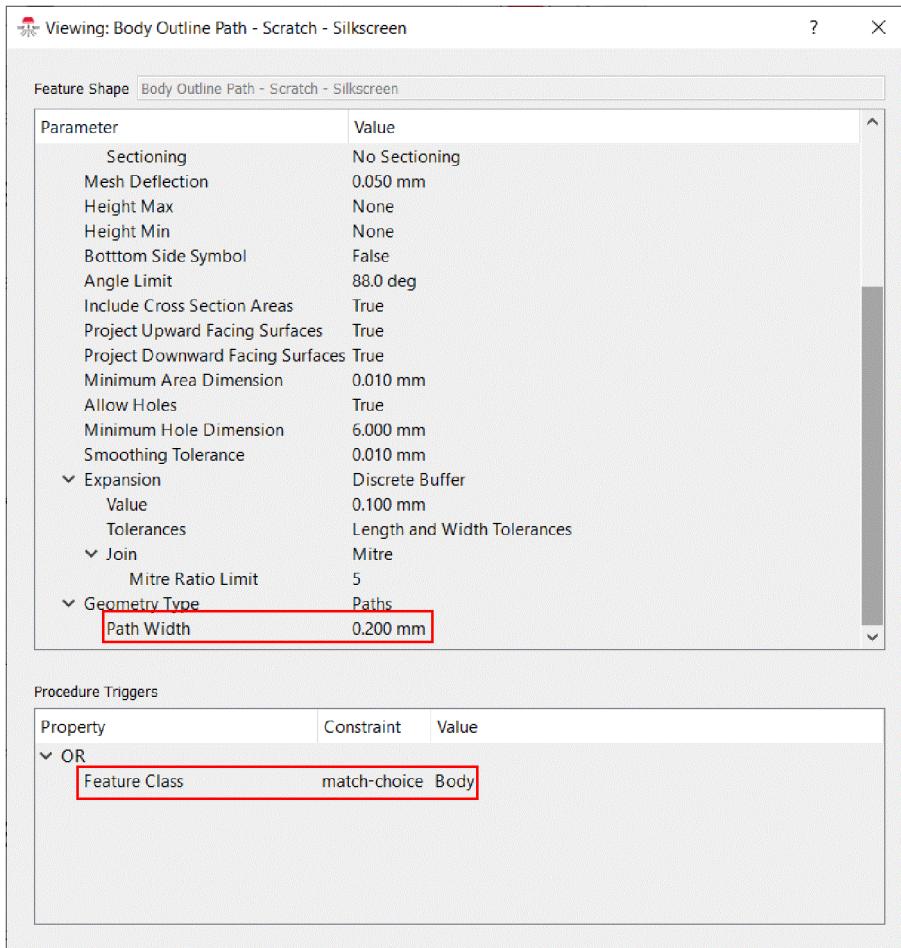
Newly saved rules are only available to the current logged in user and can be made available to all the users when published.

Editing Geometry Procedures

In this section, we will edit the rule no. 18, 19, and 20 in the Geometry Procedures section of the *Configuration (Editing)* dialog.

To edit the rules under the *Geometry Procedures* section:

1. Right-click the rule no. 18 under the *Geometry Procedures* section.
Viewing dialog appears displaying the details of the rule 18.



Rules only execute if it matches the criteria contained in the *Procedure Triggers* section is met. In the above example, *Body* must be specified as value of the *Feature Class* in the *Procedure Triggers* section. If the Package being worked on has no *Body*, then no silkscreen outline will be created.

Notice that the *Pad Width* rule is set to create a *0.200 mm* wide silkscreen outline. If this width does not meet your requirements or specifications, it can be edited to meet your needs.

2. Close the *Viewing* dialog.
3. Right-click rule no. 19 in the *Configuration (Editing)* dialog and choose *Delete*. Deleting this rule will create a silkscreen outline that goes around the entire perimeter of the package as this rule extracts only the corner points.
4. Click *OK* to close the *Configuration (Editing)* dialog.
The rule is saved under *Workspace - Footprints* section in the *Configuration (Viewing)* dialog.

Testing Edited Rules

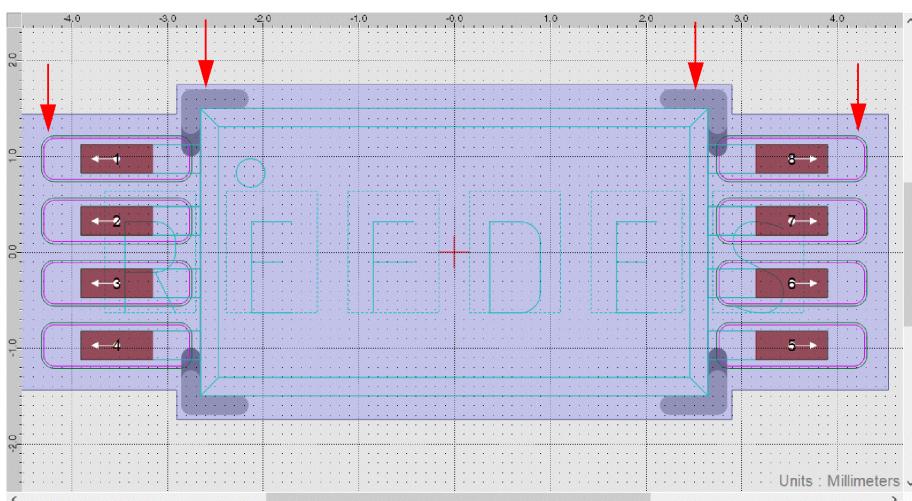
To test the edited rule:

1. Choose *Pin Count* in the *Property* drop-down, *equals* in *Constraint* drop-down.
2. Type `8` in *Value* and then press *Enter*.
3. Double-click the *MO-150_AA* package from the search results to load it.
4. Click *Footprint (2D)*.
5. Add the new rule *IPC-B_demo_s005* to the *Apply* menu.

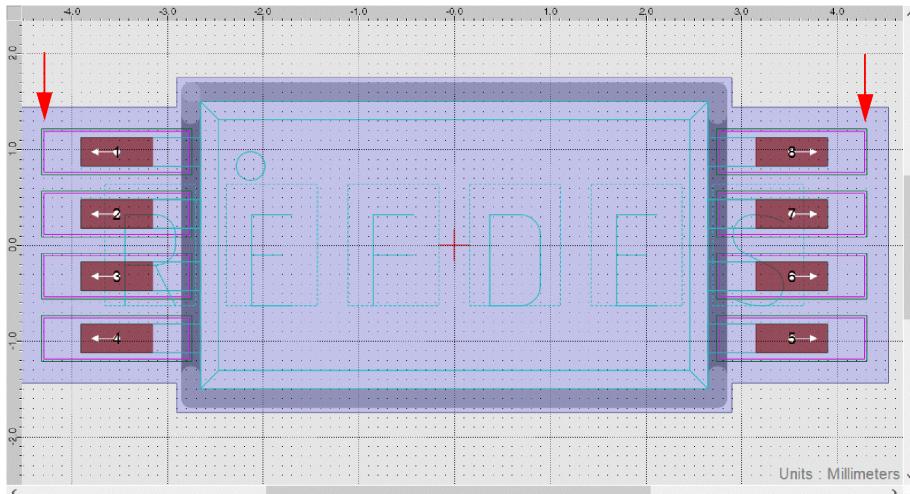
For more information about adding rules to the Apply menu, refer to the *Configuration* section of the *ECAD-MCAD Library Creator User Guide*.

6. Choose *Apply - Training/IPC-B*.

SMD Pads are added to the footprint. Notice how the SMD pad corners are rounded. This is because in the default *IPC-B* rule, the *Variable – Rounded Rectangle* is set to *True*.



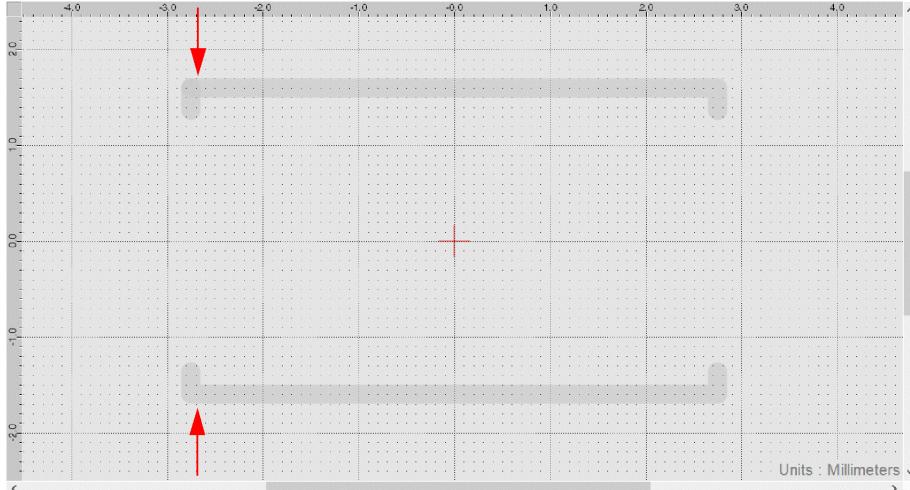
7. Choose *Apply - IPC-B_demo_s005[Footprint]* to test the new rule you just created. The SMD pad corners are now changed to square. This is because in the default *IPC-B* rule, the *Variable – Rounded Rectangle* is set to *False*.



Next, you will verify the changes done to the rule in the *Editing Geometry Procedures* section.

8. Choose *Apply - Training/IPC-B*.
9. In the *Layer Control* panel, right-click and choose *Hide - All* to turn off all the layers.
10. Select the *Silkscreen_Top* check box to display only the outline created by the normal *IPC-B* rule.

The horizontal silkscreen lines are much longer.



Publishing Rules

In this section, you will publish the recently edited rule to make it available for all the other users.

1. Choose *Tools - Configuration*.
2. Right-click *Workspace* and choose *Publish Workspace*.

A confirmation message is displayed.

 Executing this command will publish all work that is displayed under *Workspace*.

3. Click *Yes*.

Review Changes dialog is displayed.

4. Click *OK*.

The rule is published and removed from *Workspace* in the *Configuration (Viewing)* dialog.

Summary

This concludes rule editing and publishing. In this module, you learned to edit rules and how to publish them so that these rules are available to all other users.