

# STEP Model Support In Allegro 16.6 (QIR4) September 2013

Edward Acheson  
Product Engineer  
Cadence Design Systems

## Table of Contents

<b>Preface:</b> .....	<b>3</b>
<b>MODULE 1: STEP Model Support in Allegro</b> .....	<b>3</b>
<b>Overview</b> .....	<b>3</b>
<b>STEP Models</b> .....	<b>3</b>
<b>STEP Model Viewers</b> .....	<b>4</b>
<b>STEP Model Library</b> .....	<b>4</b>
LAB 1: Defining the <i>steppath</i> Allegro env setting: .....	4
<b>MODULE 2: Mapping STEP Models in the Symbol Editor.</b> .....	<b>7</b>
<b>Overview</b> .....	<b>7</b>
<b>Mapping STEP Models in Symbol Editor</b> .....	<b>7</b>
LAB 2: Mapping a STEP Model in the Package Symbol: .....	7
<b>MODULE 3: STEP Model Mapping in Allegro PCB Editor</b> .....	<b>15</b>
<b>Overview</b> .....	<b>15</b>
LAB 3: Mapping a STEP model in the Allegro PCB Editor: .....	15
LAB 4: 3D Viewer with STEP models .....	28
<b>MODULE 4: Mapping Mechanical STEP Models.</b> .....	<b>35</b>
<b>Overview</b> .....	<b>35</b>
<b>Mapping Mechanical STEP Models</b> .....	<b>35</b>
LAB 4: Mapping a Mechanical STEP Model to a Board Drawing: .....	35
<b>MODULE 5: Exporting a Board Drawings to a STEP Model</b> .....	<b>41</b>
<b>Overview</b> .....	<b>41</b>
LAB 5: Exporting Board STEP model .....	41

## ***Preface:***

This document is based on the Allegro 16.6 - QIR 4 Release. The features defined in this document reflect new mapping capabilities introduced since the Allegro 16.6 QIR 3 release in July of 2013.

---

# **MODULE 1: STEP Model Support in Allegro**

---

## ***Overview***

The Allegro PCB Editor products provide 3D viewing of an Allegro board drawing based on the open drawings layer visibility and object selection. The 3D viewer provides a basic rendering of board geometry, conductors, via structures and component geometry with simple detail also referred to as block style or “skyscraper” viewing. STEP model mapping allows the capability of viewing components in with a higher level of detail, where a more realistic image of a component and its association with other devices, or housings can be observed.

Allegro 16.6 provides users the ability to map STEP models to Allegro footprint and mechanical symbols, display the STEP model in the Allegro PCB Editor 3D viewing tool, and to export the Allegro board drawing as a STEP model. The capability of including enclosures associated with the board is also provided for positioning and collision detection.

This Workshop will instruct the user how to:

- Map Allegro symbols to STEP models at the symbol level
- Map Allegro symbols to STEP models at the board drawing level
- Use STEP model features in the 3D Viewer
- Associate enclosure STEP models for 3D Viewing in the Allegro board drawing
- Exporting an Allegro board as a STEP model

This Workshop requires:

- This document, which includes series of labs that describe the basic usage of STEP model support within Allegro.
- A directory that contains a sample printed circuit board
- Two sub-directories,
  - *Symbols* – contains all board package symbols and pad stacks
  - *STEP\_Models* – contains STEP models used in the sample design...

## ***STEP Models***

STEP models are files that describe graphical detail for a physical part. There are currently two STEP model formats that support most electrical and mechanical parts used on a PCB. The format standards are managed by ProSTEP iVip and are often referred to as AP203 and AP214.

## STEP Model Support in Allegro 16.6

An effort by ProSTEP is underway to merge these two formats into one common format named AP242. Until completion of the AP242 standard, the AP214 format currently provides a more detailed description of a part and is most commonly preferred.

Obtaining STEP models is possible through manufacturer sites, but not all manufacturers provide STEP models. There are several 3<sup>rd</sup> party library solutions that, for a subscription or fee, will create and distribute STEP models. Some MCAD tool user communities provide a shared library of STEP models readily available to users. A user may search throughout the internet to locate a site that best works for them.

See the following web site more information on STEP models:

<http://step.nasa.gov/help/faq.html>

<http://www.prostep.org>

<http://www.prostep.org/en/projects/step-ap-242.html>

STEP models, as of this writing, have no standardized origin or orientation more especially in relationships to 2D models. Most STEP model origins generally appear to have an origin located at the absolute center of the model, but this can never be guaranteed. Orientation will also vary from model to model. Correct positioning of the STEP may require adjustment in rotation about the X, Y, and/or Z axis. Proper positioning of the 3D model may also require adjustments in the X, Y, and/or Z axis. The STEP model mapping tool assists the user in defining these offsets by visually comparing the Allegro symbol to the STEP model, where the user adjusts offset and rotation values to position the STEP model correctly to the symbol model.

Important: For a mapped or unmapped model to be displayed in the Allegro 3D Viewer, the board or package symbol MUST contain a PACKAGE GEOMETRY PLACE BOUND TOP or PLACE BOUND BOTTOM element.

## STEP Model Viewers

There are several free or trial versions of 3D STEP model viewers available by searching the World Wide Web. If permissible, locate and install one for use in viewing an Allegro board STEP model.

## STEP Model Library

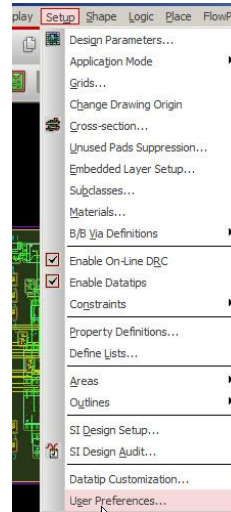
STEP models should be placed into a Library in a similar manner as other Allegro PCB Library models. A specific directory that all users have at a minimum read access permission should be defined. This path location is then assigned as a value to the Allegro library path environment **steppath**. The **steppath** environment variable may be defined through the user preferences setting in the Allegro PCB Editor tool.

## LAB 1: Defining the *steppath* Allegro env setting:

- Start the Allegro PCB Editor

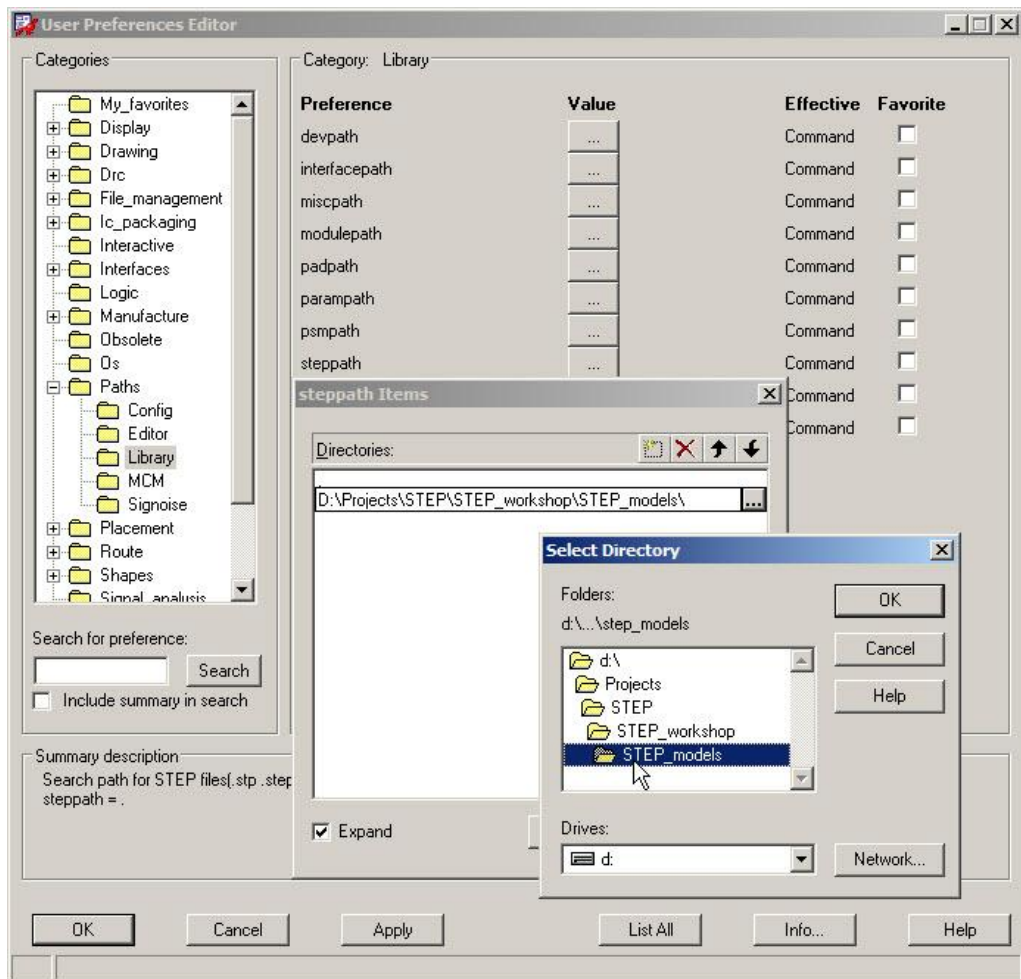
## STEP Model Support in Allegro 16.6

- **File->Open** navigate to the STEP\_workshop directory and open the **start\_step.brd** file.
- From the Allegro editor menu select **Setup->User preferences**



- In the *User Preferences* form, open the node for **Paths->Library**
- Select the **Value** button for the **steppath** preference in the *Category: Library* form.
- Add a new path, using the navigation button for the path, locate the STEP\_models directory and apply this path to the **steppath**.

## STEP Model Support in Allegro 16.6



- **Apply** this Setting and close the *User Preferences Editor*.
- Do not close the Allegro PCB Editor.
- End of Lab1.

# MODULE 2: Mapping STEP Models in the Symbol Editor.

## Overview

STEP models may be mapped to symbols in the Allegro symbol editor. Once the mapping is completed, and the symbol drawing is saved, the mapping data is preserved in the Allegro symbol.

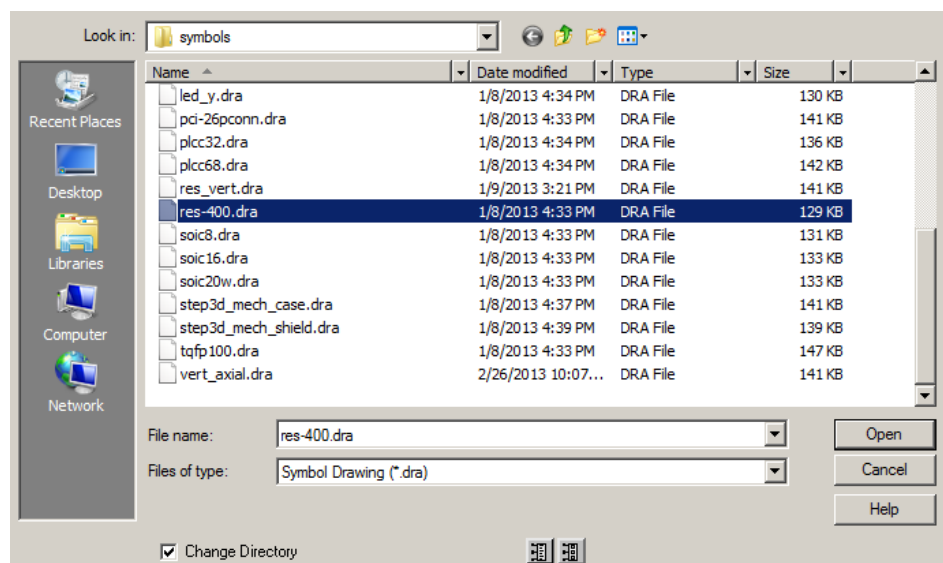
## Mapping STEP Models in Symbol Editor

STEP models are mapped through the Step Package Mapping tool located under the *Setup* menu in the Symbol Editor. Before mapping ensure that the *steppath* environment path is defined (Refer to MODULE1, LAB 1 for STEP model library path setup). Using the mapping tool, the user selects the STEP model, applies appropriate offsets, and then saves the symbol drawing. Once complete, any drawing that loads this symbol will have the mapped data contained within the symbol.

The mapping tool provides the ability to associate two STEP models to a symbol as, **Primary** and **Secondary** STEP models. Examples for using a primary and secondary model may include highly detail STEP model ( large STEP model file size) as a primary, with a less detailed (smaller STEP file size) model as the secondary or possible different mounting styles, or other features that will require a two STEP models.

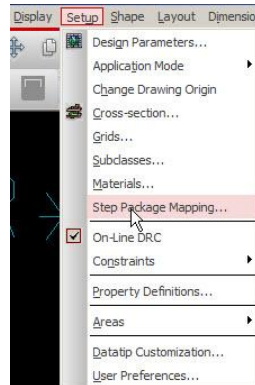
## LAB 2: Mapping a STEP Model in the Package Symbol:

- In the **Allegro Symbol Editor**, open the **res-400.dra** file located in the workshop **symbols** directory. From the Symbol Editor menu **File->Open**

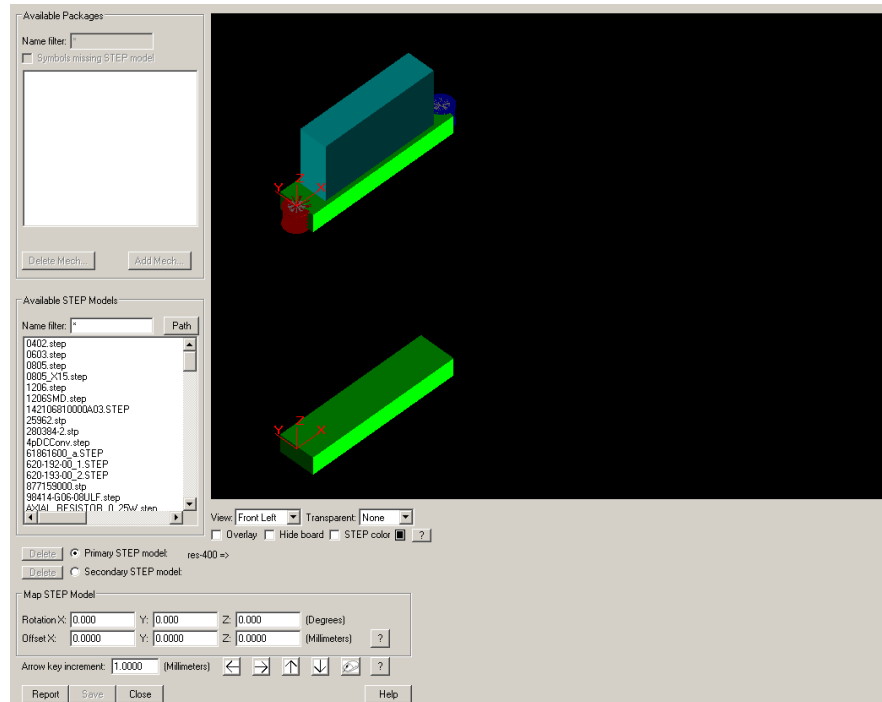


## STEP Model Support in Allegro 16.6

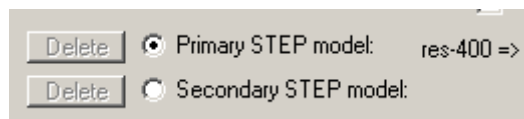
- In the *Allegro Symbol Editor* menu, start the STEP package mapping tool **Setup->Step Package Mapping...**



- The mapping tool opens.



- Set the radio button to define the **Primary STEP model**

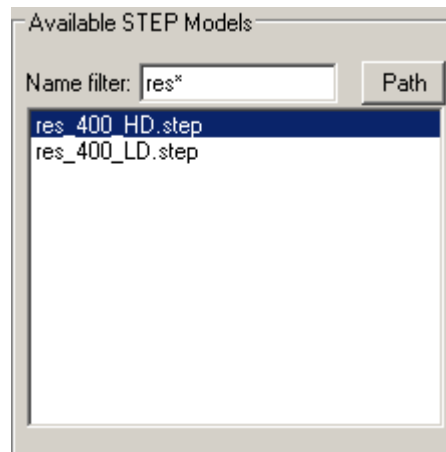


- Select the STEP model to be mapped to this symbol
  - In the **Name filter** field located in the **Available STEP models** window, enter the value **"res"**
  - Press the **"TAB"** key.

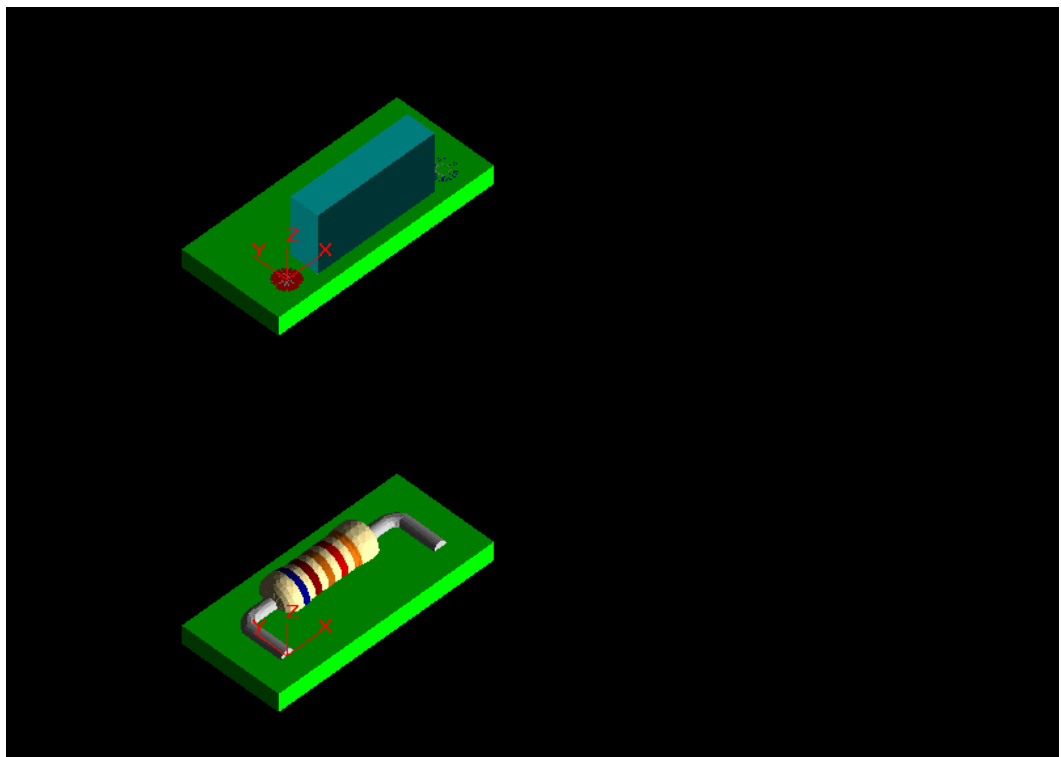


## STEP Model Support in Allegro 16.6

- The list is updated with all STEP models that begin with “res”
- Using the LMB, select the list entry “**res\_40\_HD.step**”



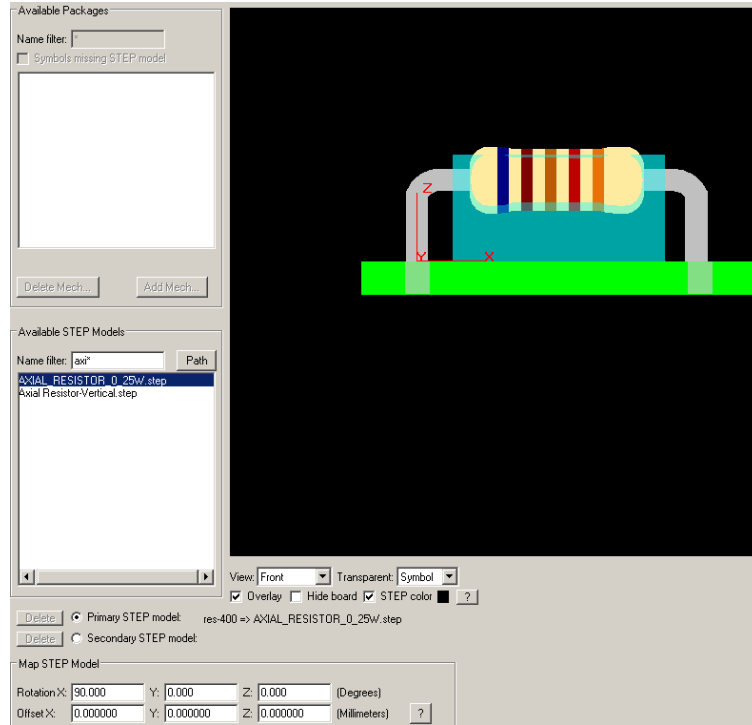
- The graphics in the mapping tool will load the STEP model.



- The STEP model and footprint symbol are not in the same rotation and orientation.
  - Set the **Rotation X**: value to **90.000** and press the **<tab>** key
  - Set the view field to **Front**
  - Enable the **Overlay** checkbox

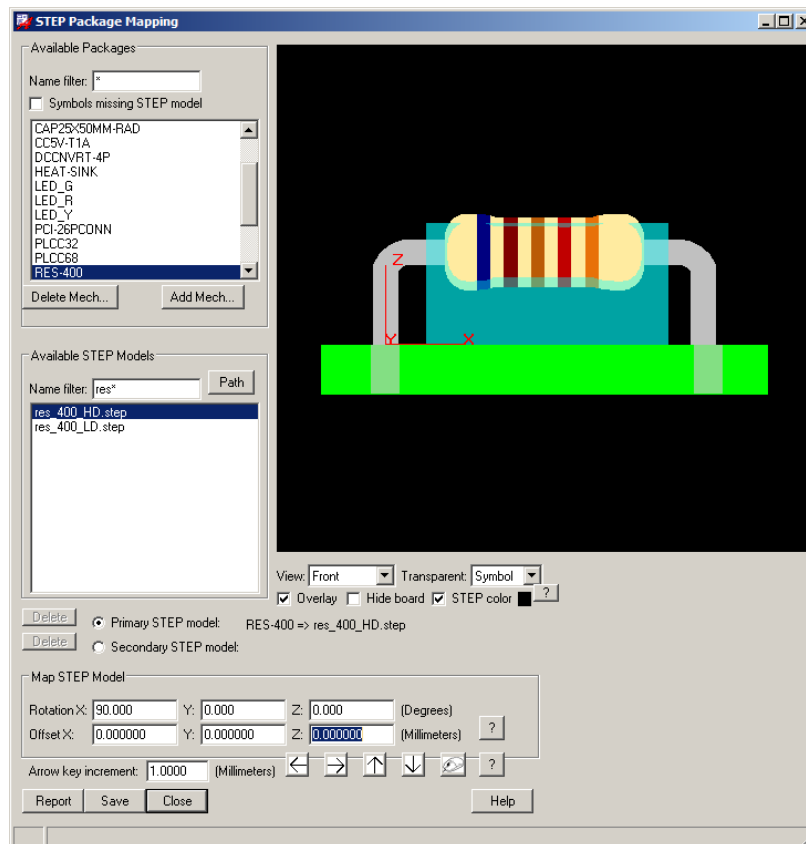
## STEP Model Support in Allegro 16.6

- The graphics will update.

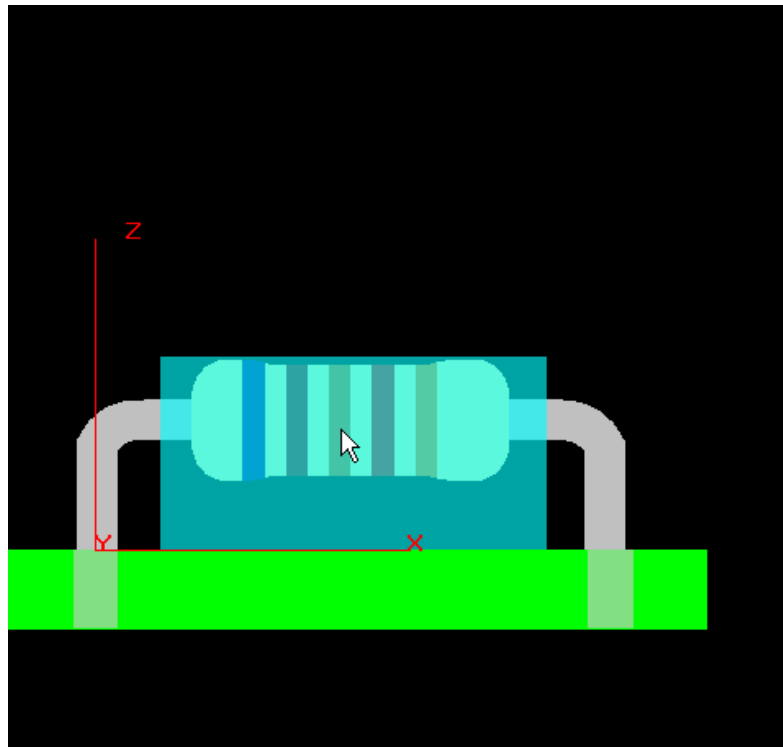


- Adjust the STEP model Z direction.
  - Select the **Offset Z:** value field with the **LMB**.
  - Place the cursor in the graphics window.
  - Using the mouse wheel, zoom into the image of the overlaid models. Roll the mouse wheel forward to zoom in, rotate the mouse wheel back to zoom out.
  - Holding the RMB in the graphics window, pan the graphics in all directions.
  - Using the features from the previous two steps, zoom the window enough to get a view of the board, footprint symbol, and STEP model. In the graphics window

## STEP Model Support in Allegro 16.6



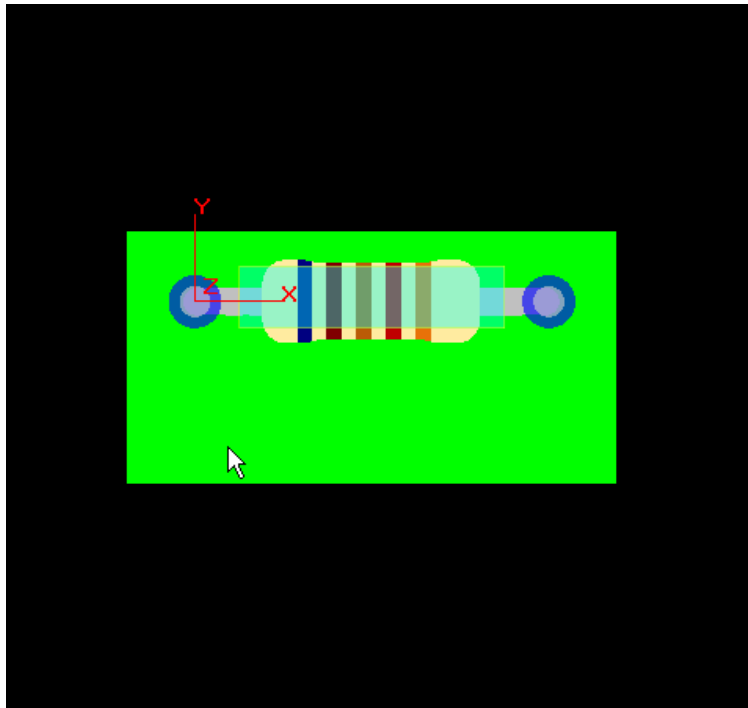
- Place the cursor over the STEP model.
- Holding the LMB, drag the STEP model so that the top of the STEP model is aligned with the top of the Allegro Symbol.



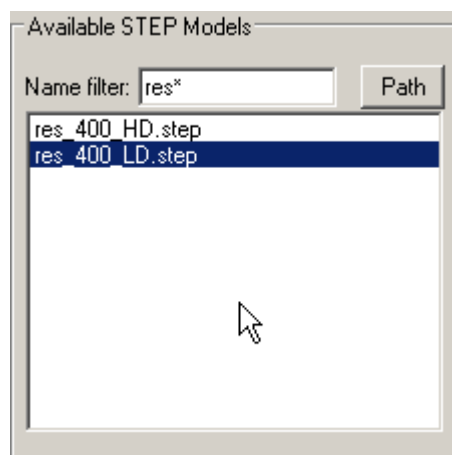
- Set the view to **TOP**

## STEP Model Support in Allegro 16.6

- Verify the STEP model is aligned correctly in the top view.

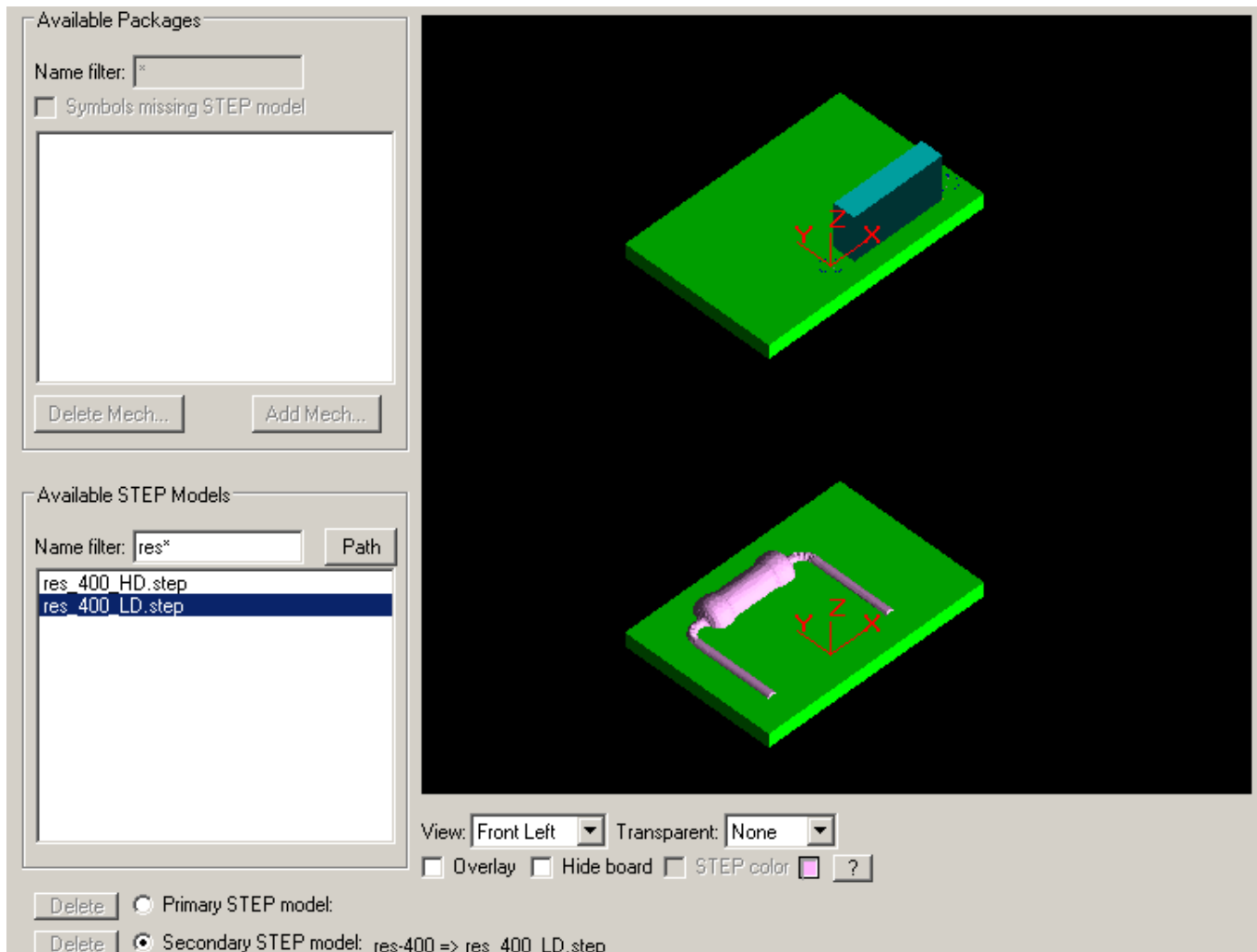


- With the footprint and STEP model aligned, Select the **Save** button in the mapping form.
- Set the radio button to define the **Secondary STEP model**
  - Unselect the **Overlay checkbox**
  - Set view to **Front Left**
- Select the STEP model to be mapped to this symbol
  - Using the **LMB** In the **Name filter** field located in the **Available STEP models** window select the list entry "**res\_40\_LD.step**"

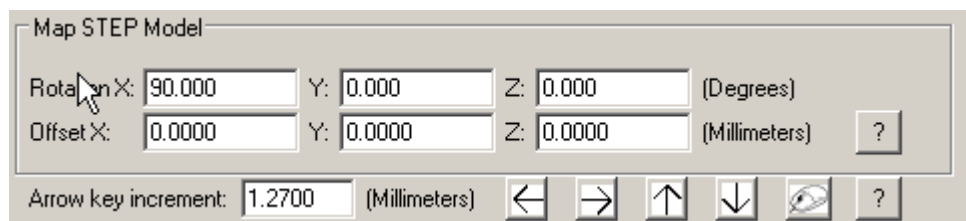


## STEP Model Support in Allegro 16.6

- The graphics in the mapping tool will load the STEP model.



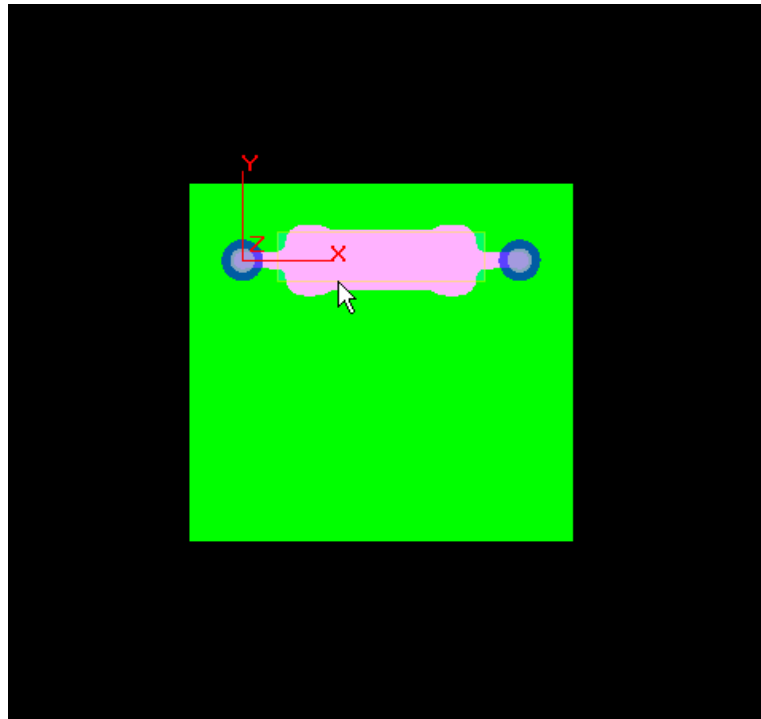
- The STEP model and footprint symbol are not in the same rotation and orientation.
  - Select the **Overlay** checkbox
  - Set the view to **TOP**
  - Set the **Rotation X** value to **90.000**
  - Set the **Arrow Key Increment** value to **1.27** (millimeters)




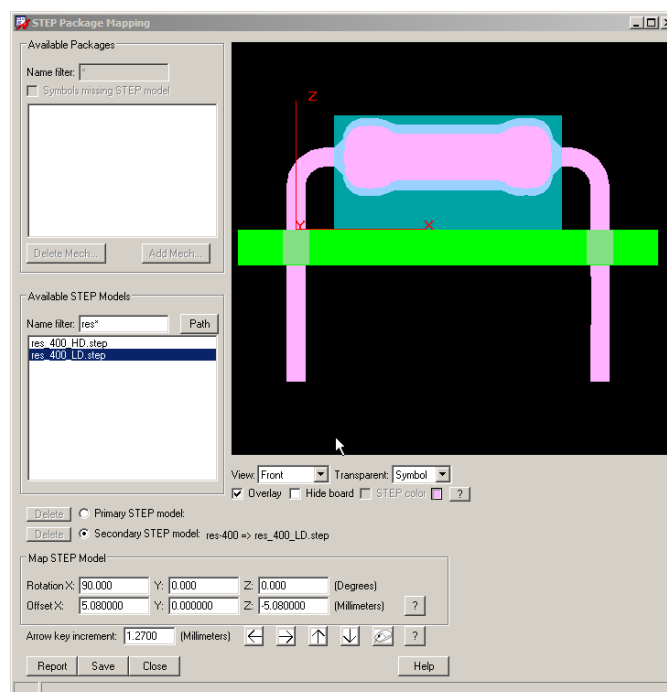
- Using the **LMB** select the right arrow button in the mapping form.

## STEP Model Support in Allegro 16.6

- The STEP model move to the right by the distance defined in the increment field.
- Continue to select the right arrow key until the pins align with the holes.



- Adjust the STEP model Z direction.
  - Set the **View** to **Front**
  - Using the **LMB** and the down arrow  button, adjust the STEP model position until it is aligned correctly.



- The symbol and STEP model are now correctly mapped. Save the mapping data to the symbol by selecting the **Save** button.
  - Mapping is complete. **Close** the mapping form.
  - Save the symbol drawing, from the Symbol Editor menu **File->Save**
  - The mapping for the Primary and Secondary STEP models is now associated in the res-400.psm file.
  - Close the Symbol Editor, from the Symbol Editor menu select **File->Exit**
- Lab 2 is Complete**

---

# MODULE 3: STEP Model Mapping in Allegro PCB Editor

---

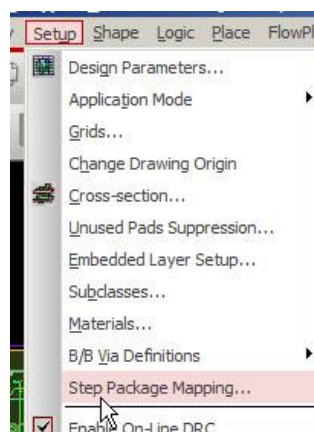
## Overview

STEP models may be associated to Allegro package and mechanical symbol models through the mapping tool from within the Allegro PCB Editor. This mapping tool sets the STEP model name to the symbol and defines offset information to correctly position the STEP model in the 3D viewer. The mapping data created is then instantiated into the Allegro symbol as a property. Symbols with STEP mapping already defined in the library may have those overridden in the board drawing using the mapping tool within the PCB Editor.

**Note: Labs 1 and 2 must be completed before proceeding to Lab 3.**

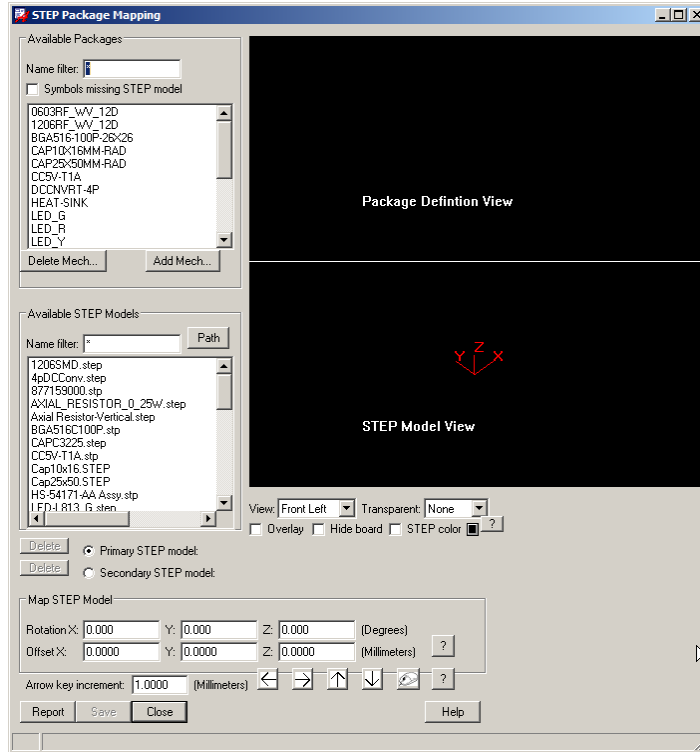
## LAB 3: Mapping a STEP model in the Allegro PCB Editor:

- Open the Allegro board file **start\_step.brd** located in the workshop directory.
- In the Allegro Editor menu select **Setup->Step Package Mapping...**



## STEP Model Support in Allegro 16.6

- The STEP Package Mapping tool opens



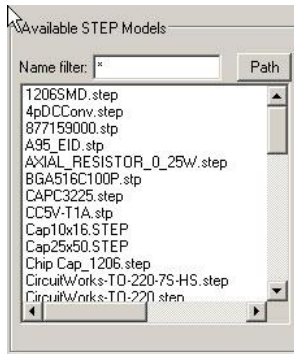
- Available Packages pane lists the symbols in the existing board drawing, with options for filtering by symbol name or by unmapped status.



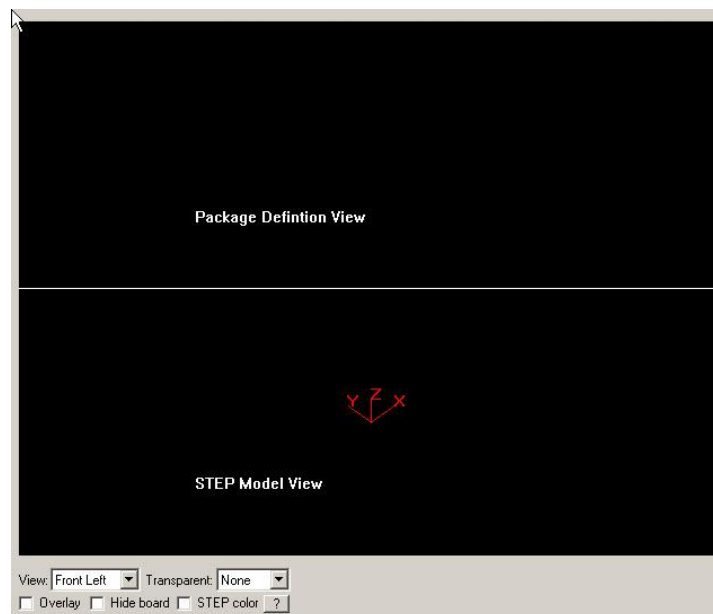
- Available Step Models pane lists all step models available in the library path specified in the *steppath* Allegro env settings. Name filtering is provided to minimize STEP model list length. Selecting the *Path* button allows modification of the *steppath* env value.



## STEP Model Support in Allegro 16.6

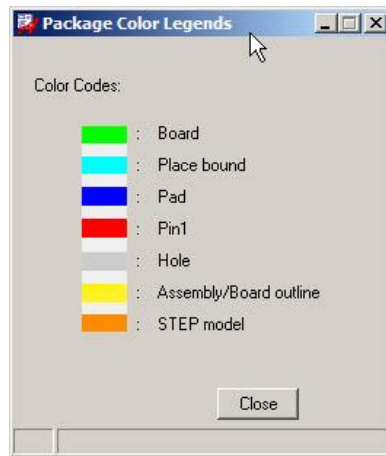


- Graphics pane displays the package model and Step model graphics with a graphical representation of a board section. Viewing tool functions assist the user in manipulating the graphics.

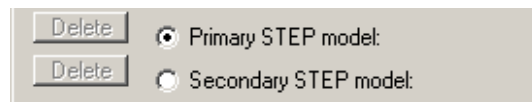


- View: various points of view are available in a pull-down menu.
- Transparent: Sets the selected model type to be transparent in the display.
- Overlay: When selected, merges the Package and STEP model views into one view.
- Hide board: When selected removes the graphical image of the board.
- STEP color: When selected uses the colors defined in the STEP model.
- ?: Displays the color legend for the graphics.

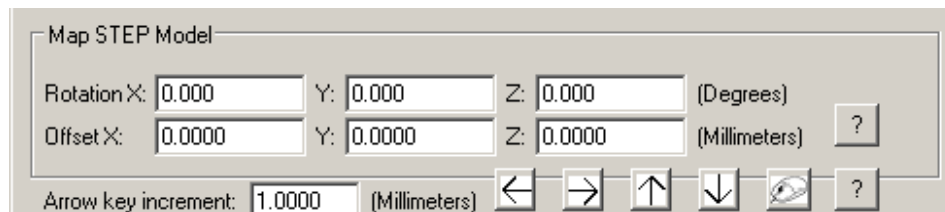
## STEP Model Support in Allegro 16.6



- Primary/Secondary selections: This option allows the creation of two mapped STEP models onto a single symbol in Allegro. This selection in the Allegro PCB editor also defines which STEP model is to be used in viewing.



- Map STEP Model pane provides the rotation and offset value entries to correctly position the STEP model in relationship to the package model. Increment keys are also used to define a specific increment to map the STEP model onto the Allegro symbol



- Report button displays a report detailing the package to STEP model name and offset mapping.

STEP Package Mapping Report

Design Name D:\Projects\STEP\STEP\_workshop\_QIR4\STEP\_workshop\start\_step.hrd  
Date Wed Oct 02 10:16:39 2013

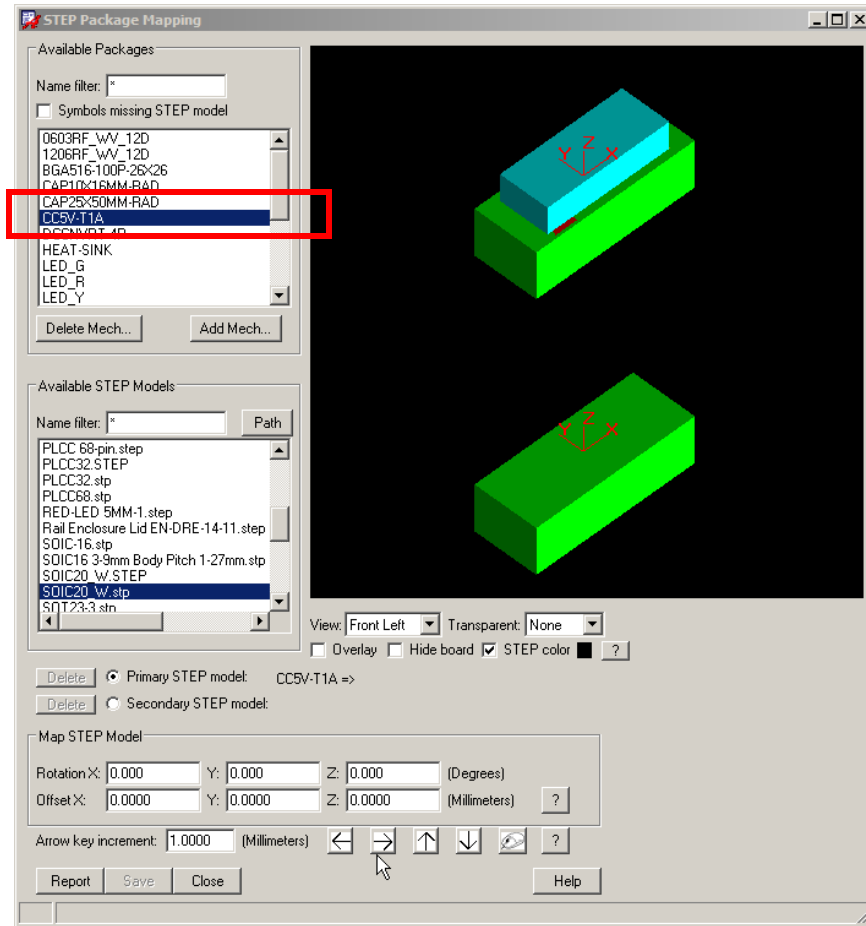
STEP Package Mapping Report (UNITS: MM)

SYMBOL_NAME	PRI/SEC	STEP_FILE	STEP_SIZE	STEP_DATE	STEP_ATTACHED	ROTATION_X	ROTATION_Y	ROTATION_Z	OF
0603RF_WV_12D	Primary	cap_0603_lowprofile.stp	141689	12/11/12 09:57:17	YES	0.000	0.000	0.000	
1206RF_WV_12D	Primary	1206SMD.stp	598323	12/19/12 09:45:43	YES	90.000	0.000	0.000	
BGA516-100P-26X26	Primary	BGA516C100P.stp	126274	12/19/12 10:11:50	YES	90.000	0.000	90.000	
CAP10X16MM-RAD	Primary	Cap10x16.STEP	639158	01/08/13 12:43:23	YES	0.000	0.000	0.000	
CAP25X50MM-RAD	Primary	Cap25x50.STEP	704105	01/08/13 13:43:30	YES	0.000	0.000	0.000	
CC3V-T1A									
DCCNVRT-4P	Primary	4pDCCnv.stp	95786	12/19/12 12:20:20	YES	90.000	0.000	0.000	
HEAT-SINK	Primary	HS-54171-AA Assy.stp	493414	12/19/12 10:27:12	YES	90.000	0.000	0.000	
LED_G	Primary	LED-L813_G.stp	141822	01/08/13 13:50:05	YES	90.000	0.000	90.000	
LED_R	Primary	LED-L813_R.stp	141860	01/08/13 13:52:25	YES	90.000	0.000	90.000	
LED_Y	Primary	led-L813_Y.stp	141805	01/08/13 13:50:55	YES	90.000	0.000	90.000	
PCI-26PCONN									
PLCC32	Primary	PLCC32.stp	233961	12/19/12 09:51:47	YES	0.000	0.000	0.000	
PLCC68	Primary	PLCC68.stp	2706439	12/19/12 10:04:35	YES	90.000	0.000	0.000	
RES-400	Primary	res_400_HD.stp	189300	01/08/13 13:38:00	YES	90.000	0.000	0.000	
	Secondary	res_400_LD.stp	95249	08/29/13 14:29:14	NO	90.000	0.000	0.000	

- Save button saves the mapping data of the current objects displayed and defined in the graphics pane.
- Delete button removes mapping data from the package symbol displayed in the graphics pane
- Close button ends the mapping session (without saving) and closes the STEP Package Mapping tool.

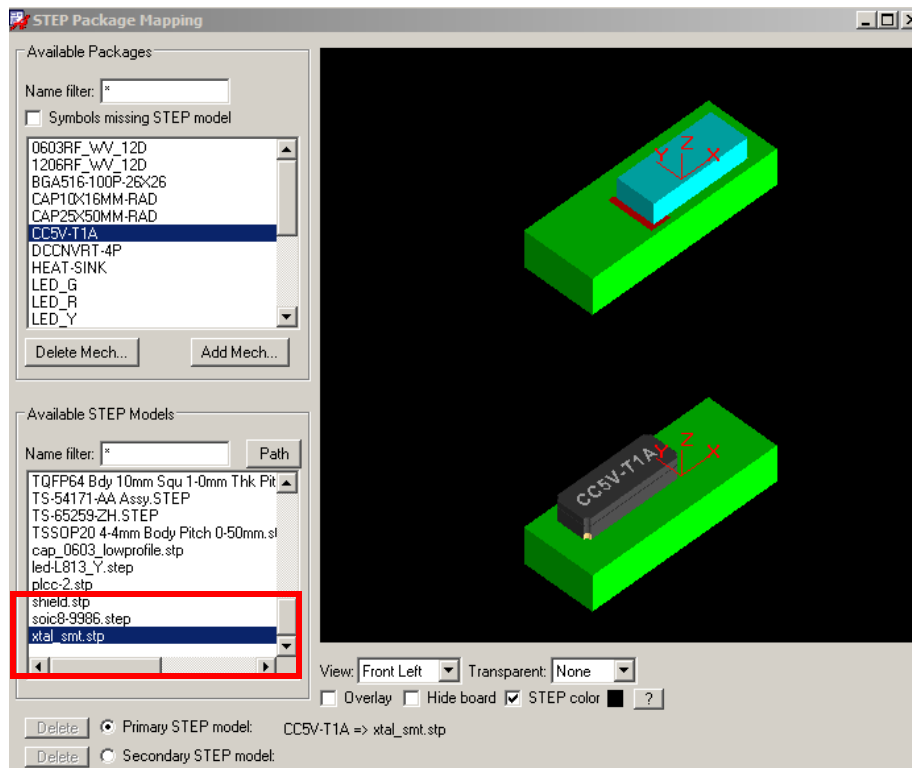


- In the **Available Packages** list, using the **LMB** locate and select the **CC5V-T1A** package symbol name. The graphics for this are displayed in the display panel.

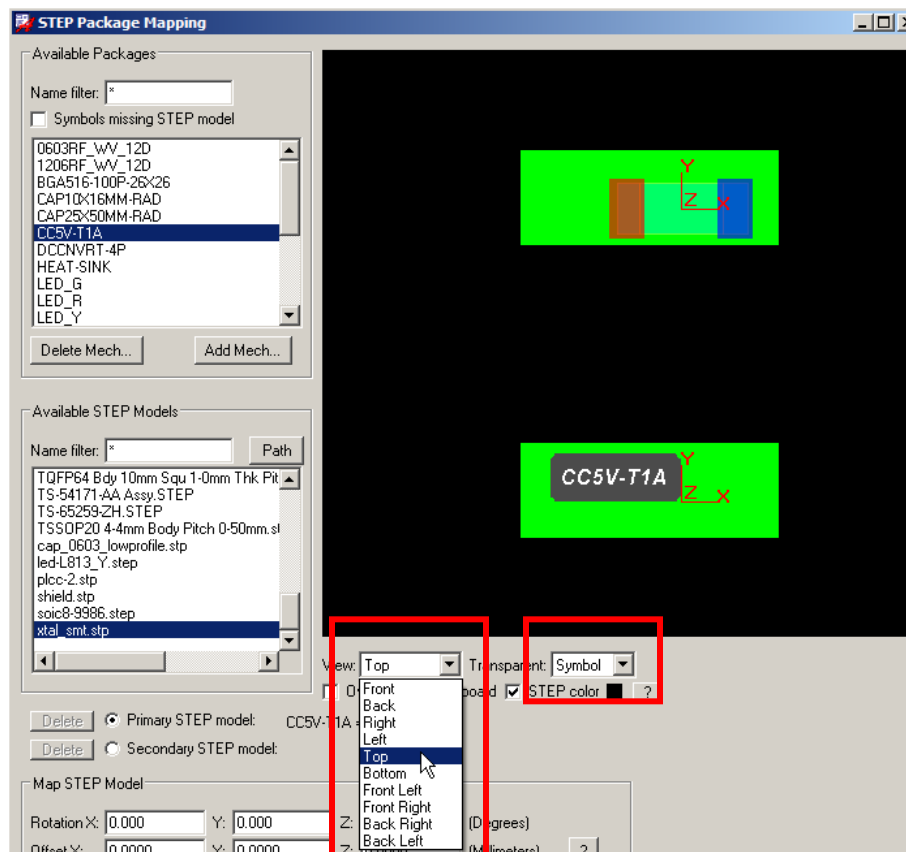


- In the **Available STEP model** pane list, using the **LMB**, scroll down and locate, then select the **xtal\_smt.stp** STEP model name. The STEP model graphics are now displayed. This is an example of an AP-214 STEP model. Notice the detail and color features that are available.

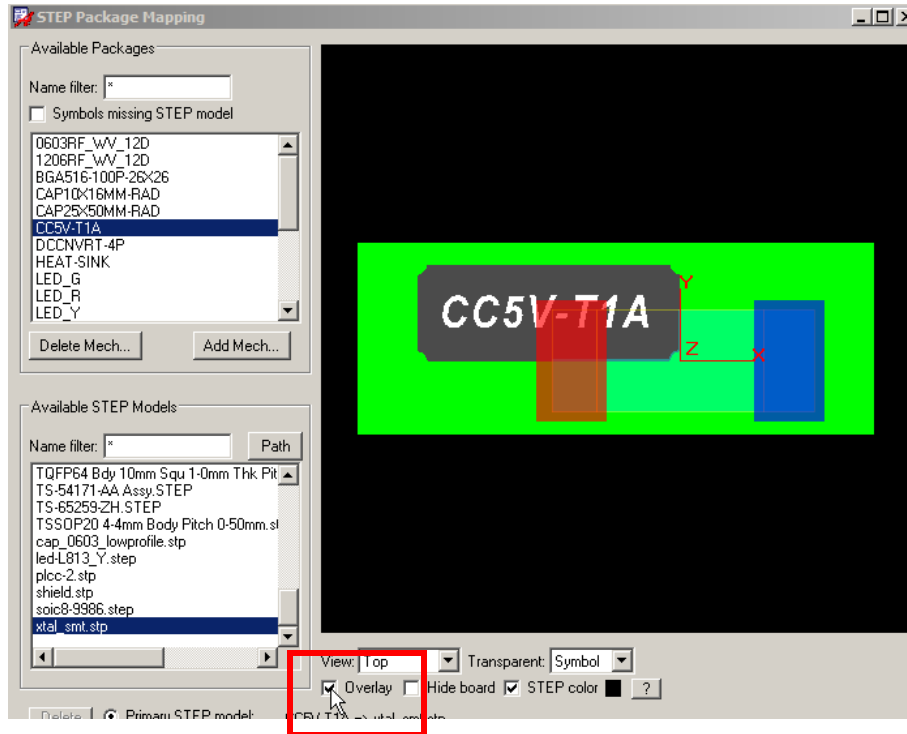
## STEP Model Support in Allegro 16.6



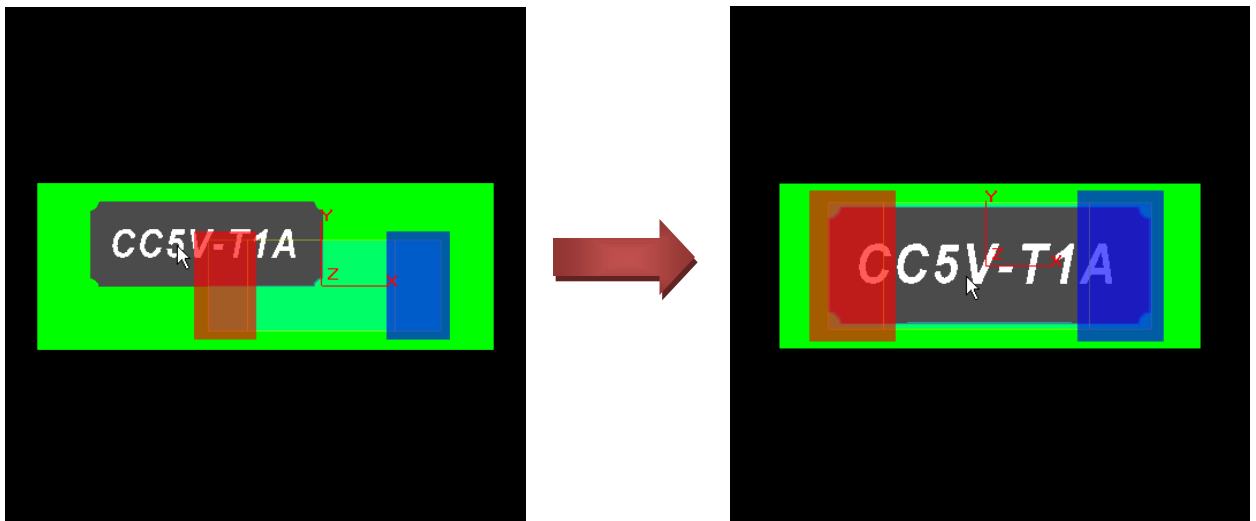
- At first glance we can see that the symbol origins are not the same. To further observe the differences, first set the Transparent value to **Symbol**, then set the view to **Top** and notice how the packages differ in their origin.



- Now overlay the two symbols by selecting the **Overlay** checkbox to further see the differences.



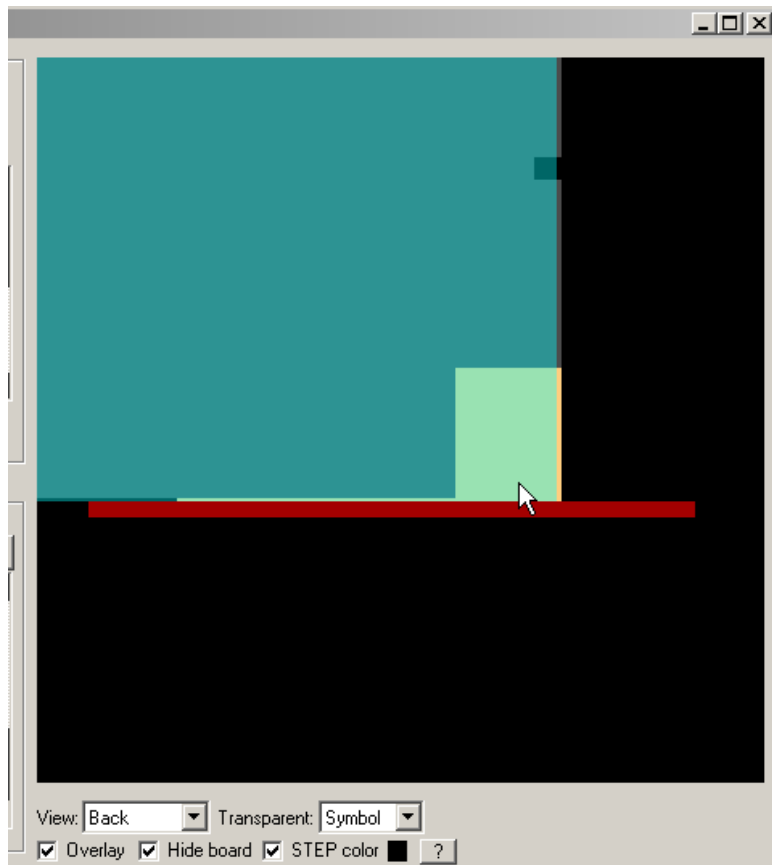
- In this view, we can see that the X and Y offset values need to be set. Using the **LMB**, select the **Offset X:** field. Next place the cursor onto the image of the STEP model in the graphics window. Holding the **LMB**, drag the STEP model into proper position. When alignment is complete, release the **LMB**.



- Change the **View** to **Front**. Select the Hide board option. Select the **Offset Z:** field. Place the cursor in the graphics window. Using the mouse wheel (if available), scroll inward to zoom into the graphics, outward to zoom out. Use the **RMB** to pan the

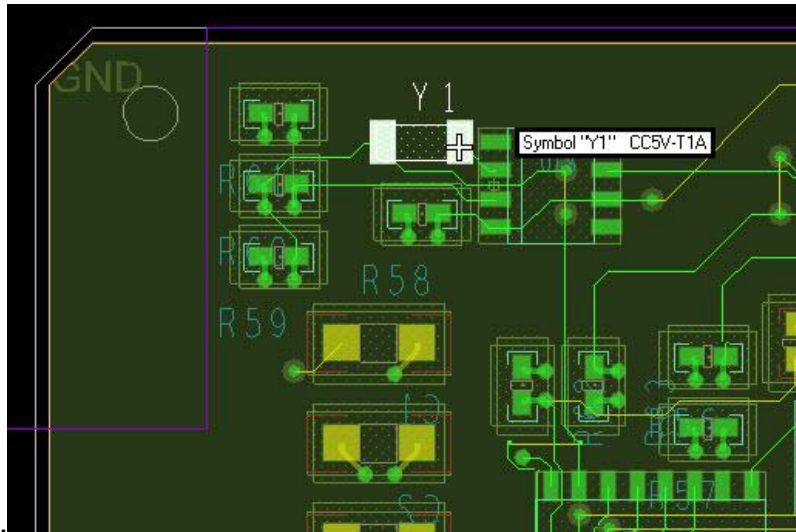
## STEP Model Support in Allegro 16.6

graphics window. Using the zoom and pan features, manipulate the graphics window to zoom into and view the contact position.

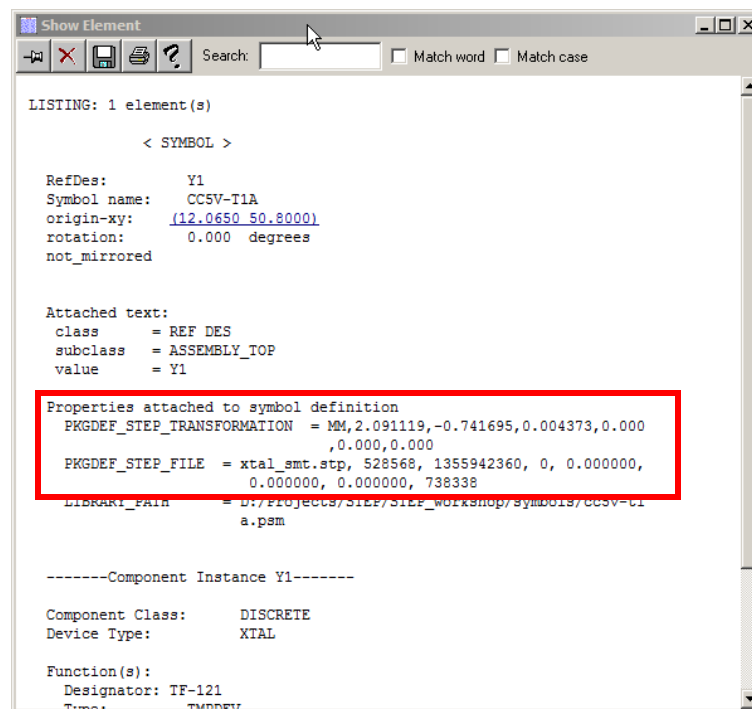


- This view shows that the STEP model is not properly seated on the board in the Z axis.
- Using the LMB with the cursor over the STEP model, position the STEP model to be correctly positioned on the pad in the Z axis.
- Select the **Save** button to preserve the current mapping for this package model. Once the save is complete, the Save button is disabled. **Close the STEP Package Mapping tool.**
- In the Allegro Editor, locate symbol Y1 (upper left hand corner of board) and show element

## STEP Model Support in Allegro 16.6

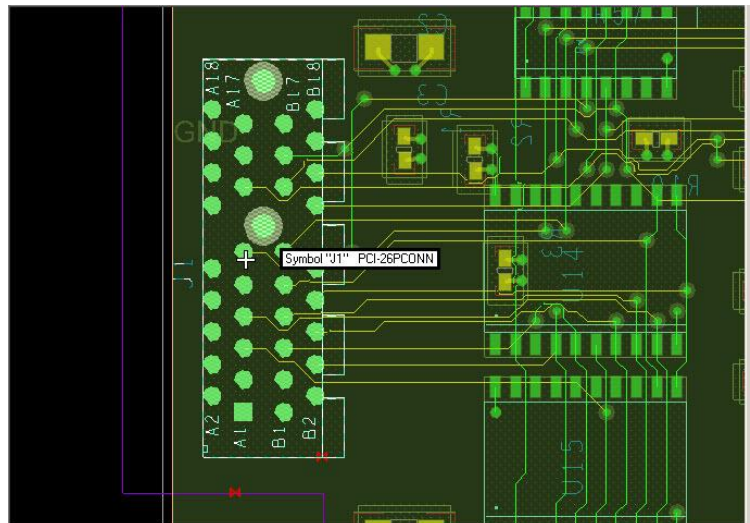


- In the Show Element form, the STEP model mapping properties are displayed. These properties become part of the symbol definition and cannot be modified outside of the STEP Package Mapping tool.

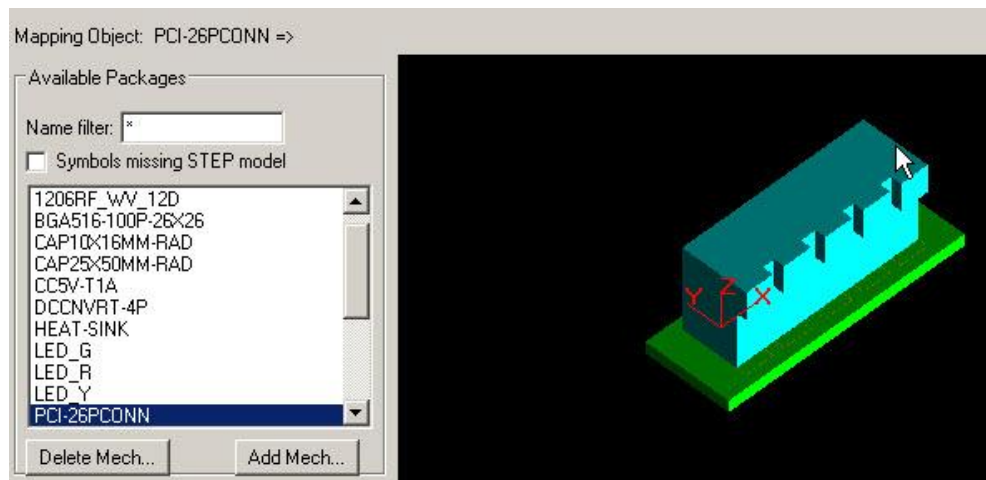


- When mapping of all models is complete, running the File->Export->Libraries... utility saves the embedded mapping values into the symbol.
- Close the Show Element form.
- Open the STEP Package Mapping tool. (*Setup->Step Package Mapping...*)
- Selection of package models for mapping may also be accomplished from the Allegro PCB Editor. With the mapping tool open, in the PCB Editor tool, navigate in the drawing to locate connector symbols **J1**, (Left side of board drawing). Using the **LMB**, click on the symbol **J1**.

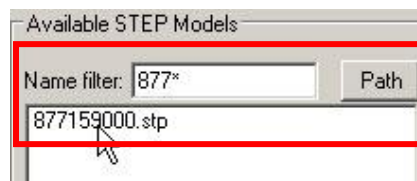
## STEP Model Support in Allegro 16.6



- The symbol is now active in the mapping tool.



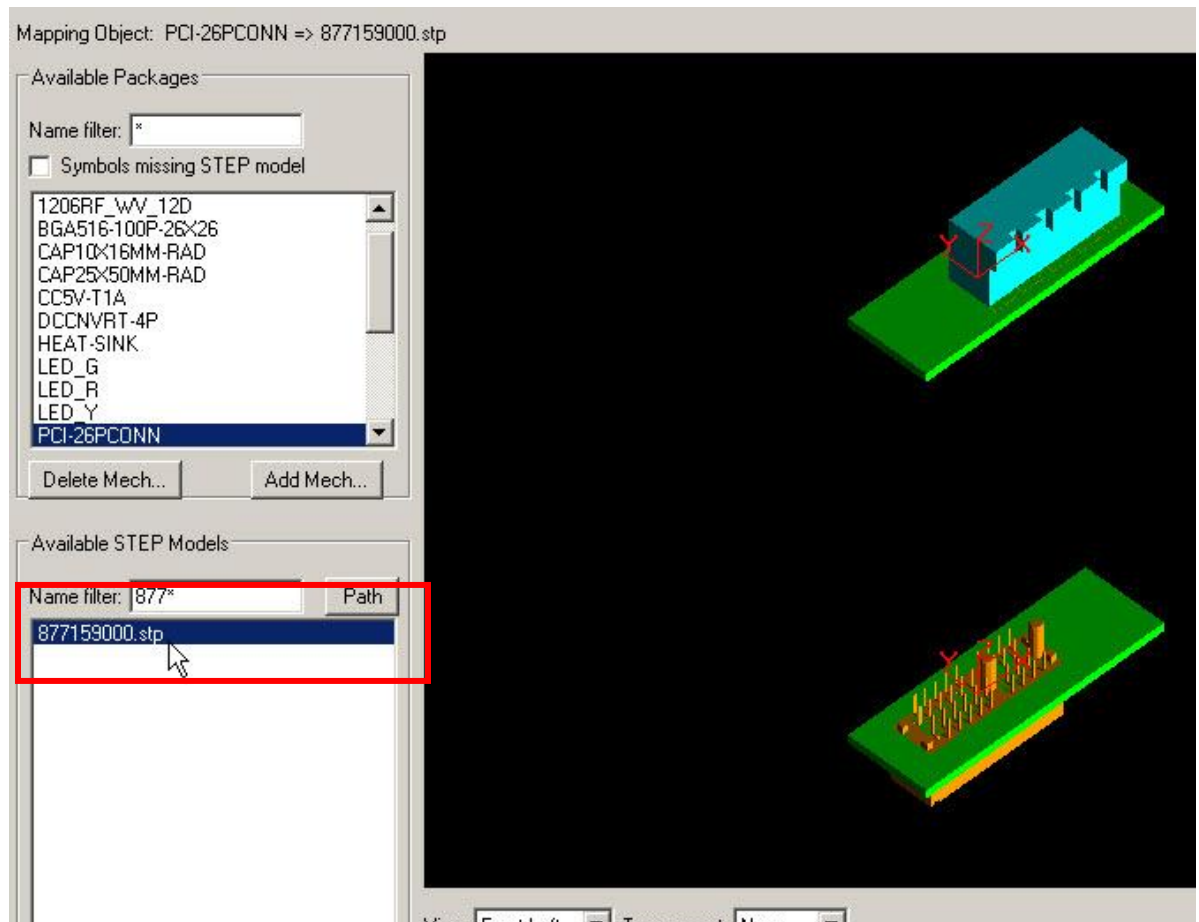
- In the *Available STEP models* pane **Name** filter field, enter **877\*** (Tab).



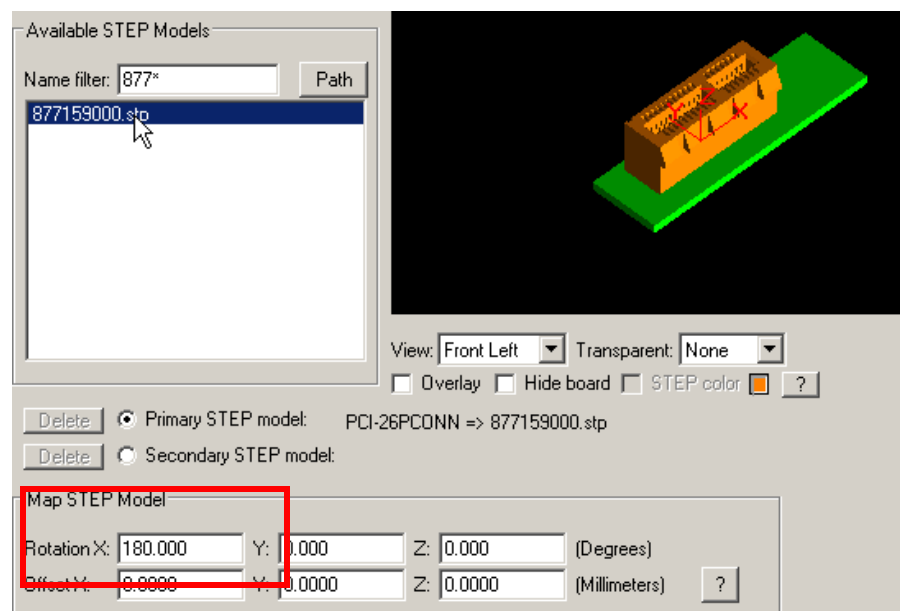
- The STEP model names that match the filter are displayed. **LMB Select** STEP model **877159000.stp**. The graphics displays the STEP model. This is an example of an AP-203 STEP model. Some detail but one single color.



## STEP Model Support in Allegro 16.6



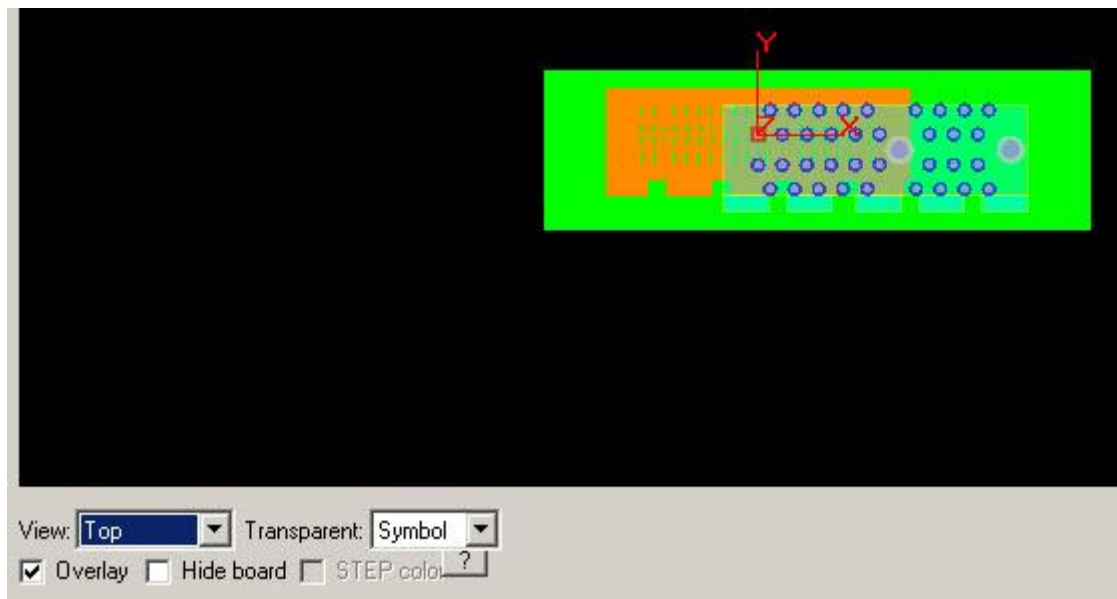
- From the graphics it is obvious that the current origin and orientation do not match. To align the STEP model to the proper mounting surface, the value for Rotation X must be set to 180. Set the value for **Rotation X: 180 (Tab)**



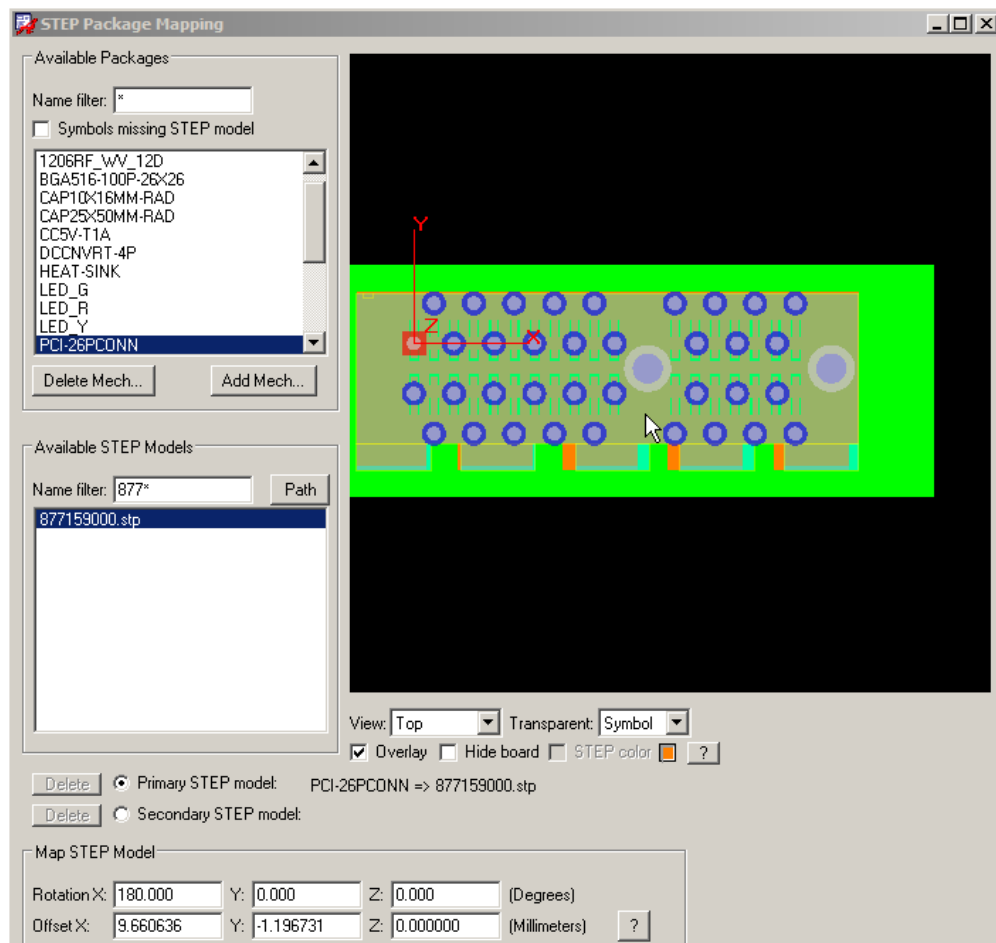
- The STEP model is now positioned onto the proper assembly layer.

## STEP Model Support in Allegro 16.6

- Set the **View:** to **Top**, Set the **Transparent** to **Symbol**, and select the **Overlay** checkbox.

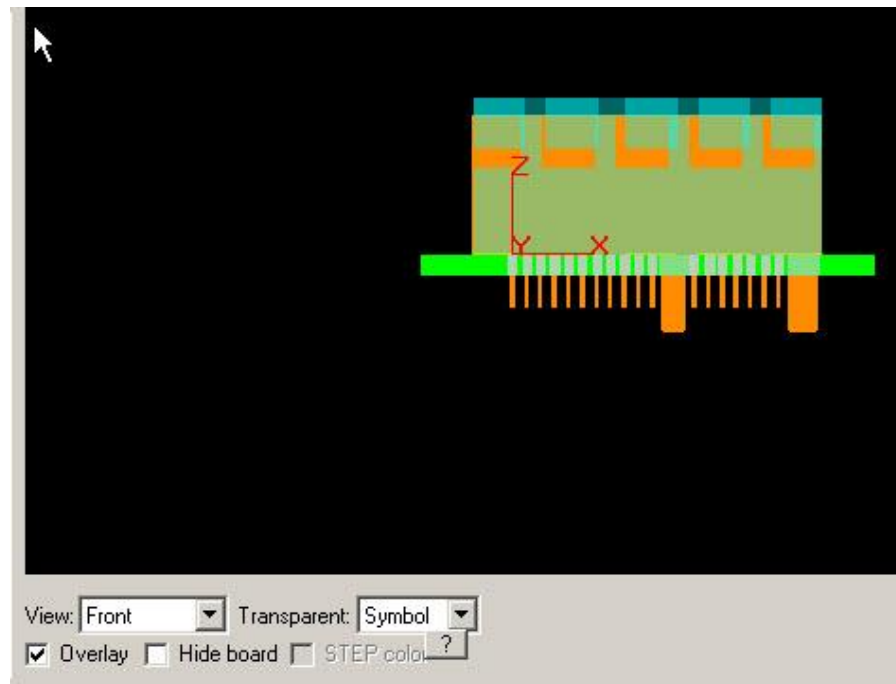


- This view shows the X and Y are mismatched and offsets must be applied. Using the **LMB** to align the STEP model to the footprint.

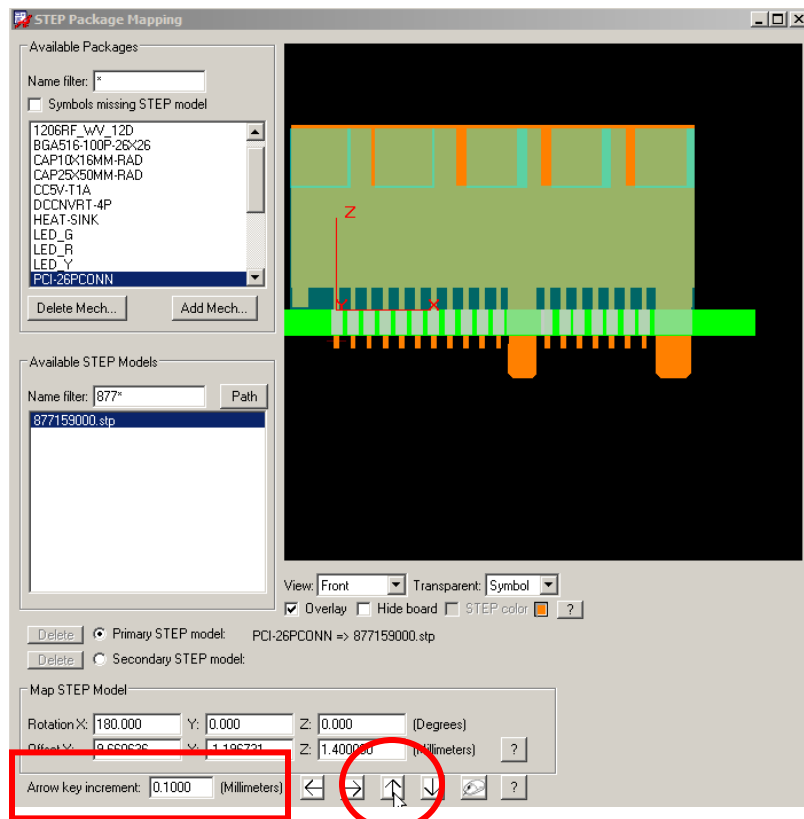


## STEP Model Support in Allegro 16.6

- The Graphics update and pins are aligned. Set the *View:* to **Front**.



- The base of the connector is resting into the board surface. Raise the Connector off by using the incremental option. Set **Arrow Key increment** value to **0.1000** (millimeters).
- Using the LMB, select the up arrow. Continue to select the up arrow until the STEP model is position correctly in the Z-Axis.



## STEP Model Support in Allegro 16.6

- Select the **Save** button to preserve these setting to the package symbol, and **Close** the STEP Package Mapping tool.
- Save the drawing under a new name **File->Save as mapped1.brd**
- **Lab 2 is complete.**

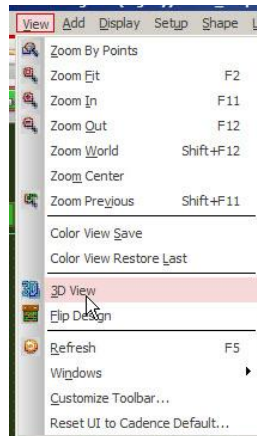
**Note: Labs1,2, and 3 must be completed before proceeding to Lab 4.**

### LAB 4: 3D Viewer with STEP models

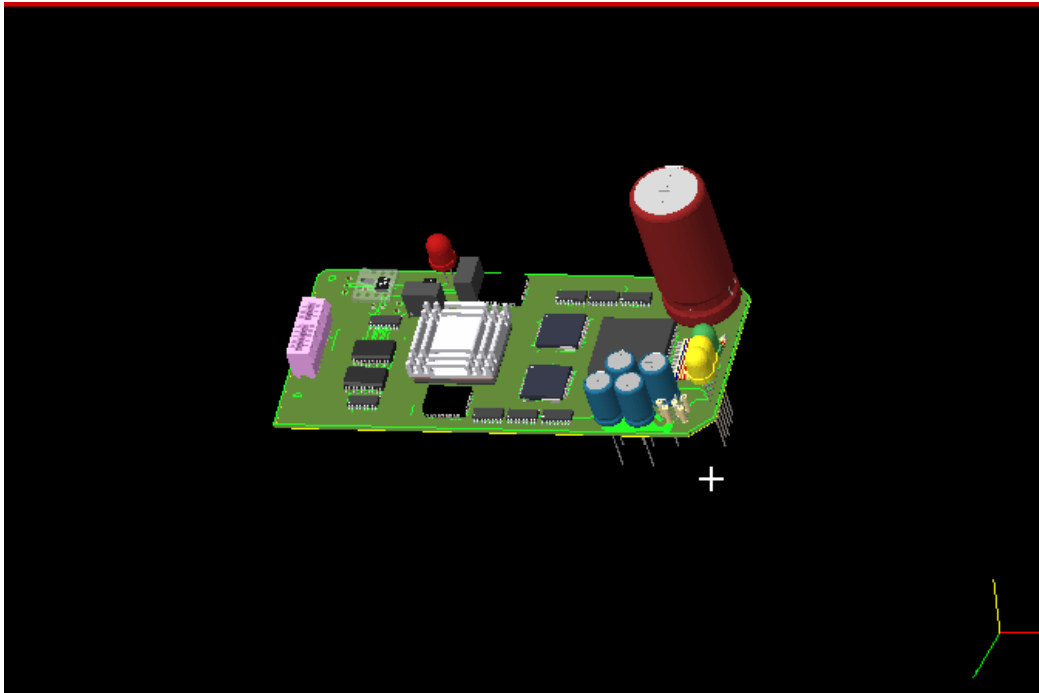
Once STEP model mapping is completed, the Allegro 3D viewer will display the graphical representations of the STEP models. The 3D viewer relies on the currently visible layers in the Allegro PCB editor to determine what is displayed in the viewer. To view the 3D STEP models, the following CLASS/SUBCLASSES must be visible in the Allegro PCB Editor window:

```
PACKAGE GEOMETRY/PLACE_BOUND_TOP  
PACKAGE GEOMETRY/PLACE_BOUND_BOTTOM  
MANUFACTURING/STEP3D_ASSEMBLY_ENCLOSURE
```

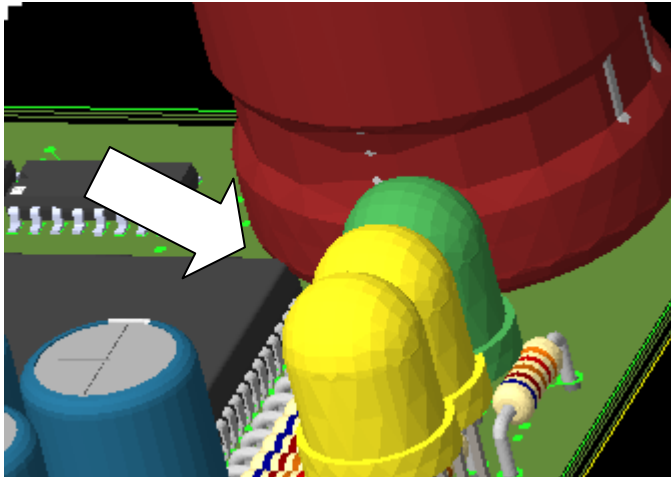
- Open the Allegro Color Dialog and verify that the subclasses for PACKAGE GEOMETRY/PLACE\_BOUND\_TOP, PACKAGE GEOMETRY/PLACE\_BOUND\_BOTTOM, and MANUFACTURING/STEP3D\_ASSEMBLY\_ENCLOSURE are set to visible.
- Select the 3D viewer ICON in the Allegro Editor toolbar or from the menu select View->3D View.



- When the 3D viewer opens, the symbols (with mapping associated) are now replaced by STEP models. Rotating and panning through the 3D image, placement of devices can be viewed for possible issues in the design.

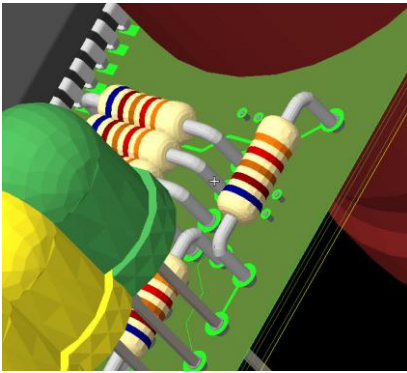


- The example design created in this workshop large radial capacitor has two issues:
  - Mapping for model is incorrect and should be corrected.
  - Collides with the 68 pin PLCC device

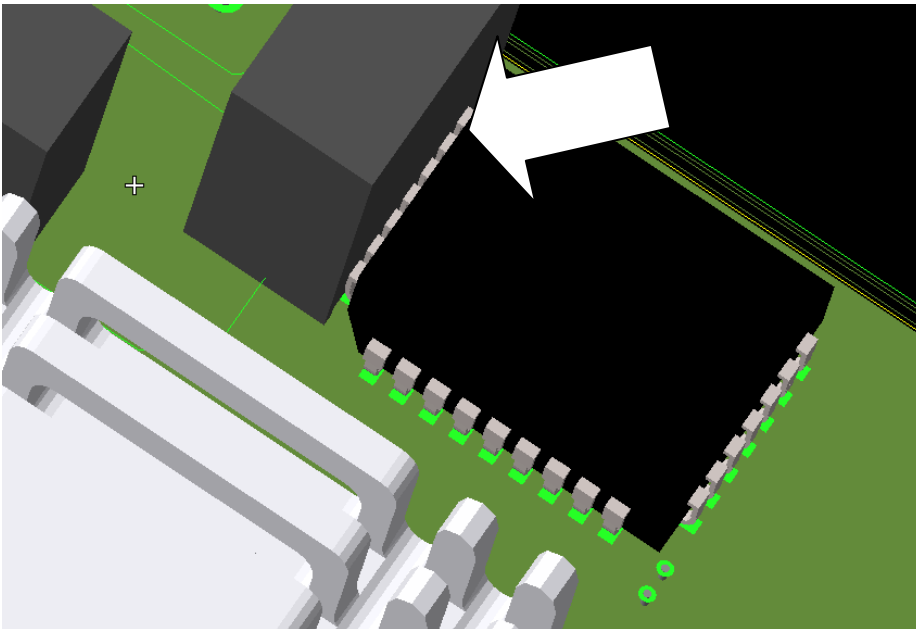


- With mapping Z offset set to actual mounting on the board for the LED, as in this example, the resistor positioned in the LED's shadow does not appear to be colliding.

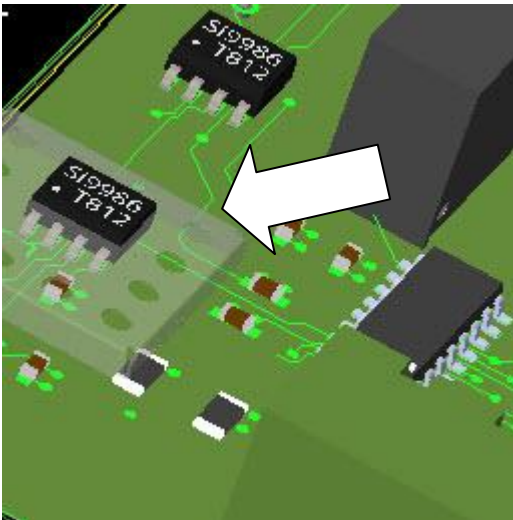
## STEP Model Support in Allegro 16.6



- This is an example of a part that may be incorrectly defined in the library as the pins align, but the bodies do not.

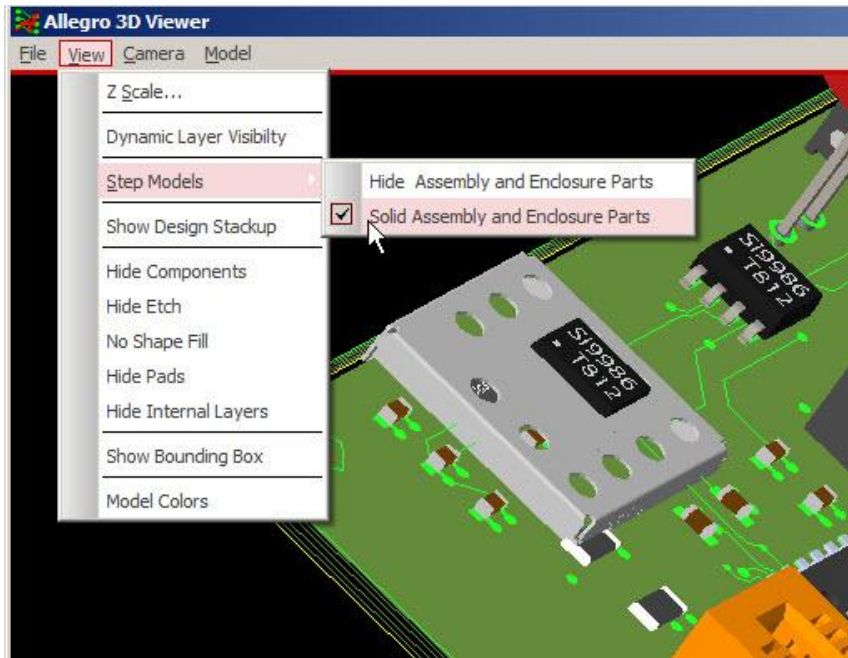


- In the next example, a component appears to be protruding through a shield.

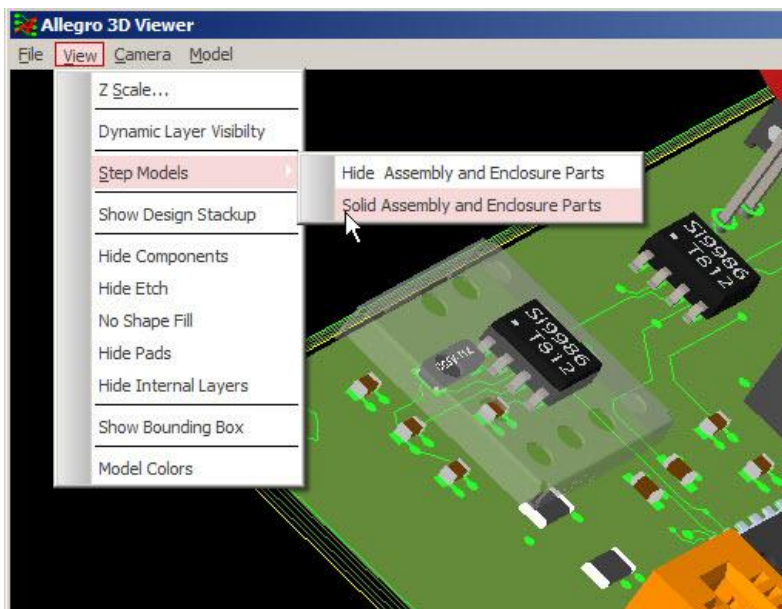


## STEP Model Support in Allegro 16.6

- Mechanical models added to the board drawing as mapped STEP models are transparent by default. The 3D viewer **View** menu provides two options for managing the display of these model types.
- Select **View->STEP Models->Solid Assembly and Enclosure Parts**. The mechanical STEP models that were transparent are now solid.



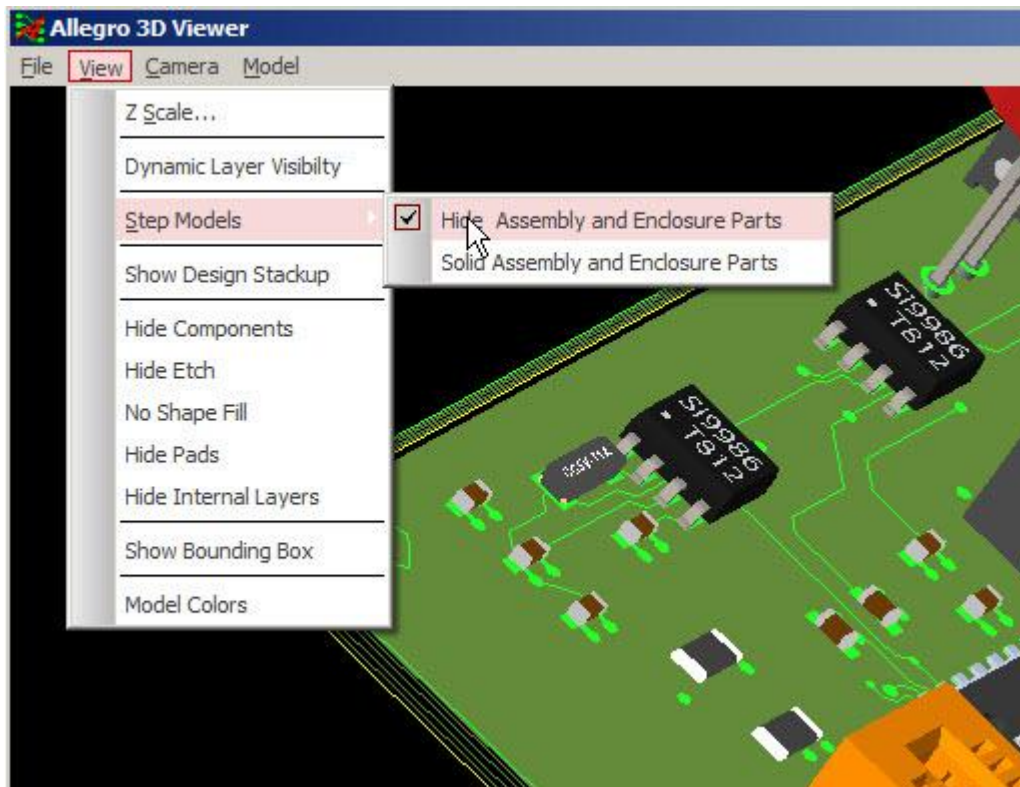
- Selecting **View->STEP Models->Solid Assembly and Enclosure Parts** again will toggle the same models to transparent.





## STEP Model Support in Allegro 16.6

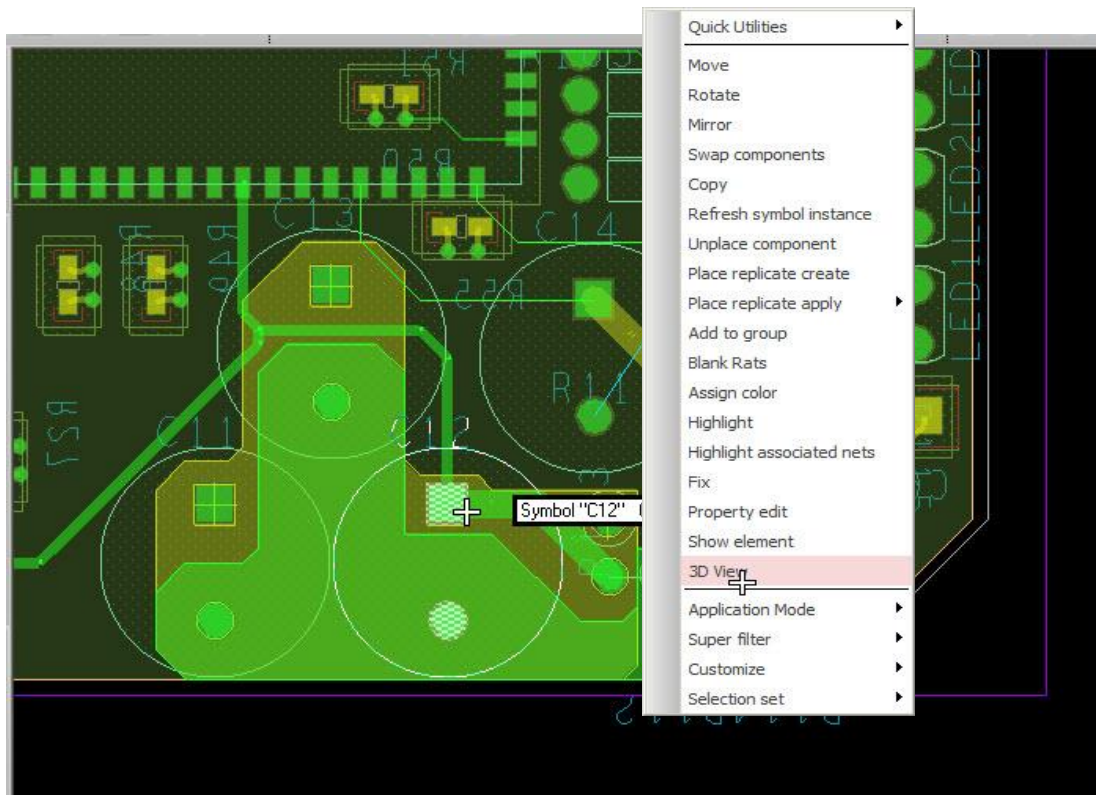
- Select **View->STEP Models->Hide Assembly and Enclosure Parts**. The mechanical STEP models that are now hidden from the display.



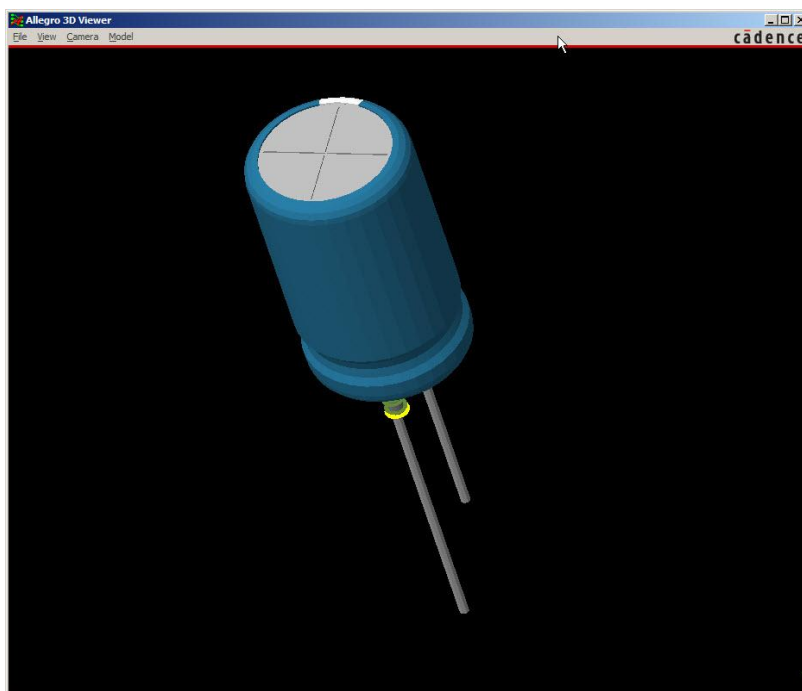
- Selecting **View->STEP Models->Hide Assembly and Enclosure Parts** again will enable display of the hidden mechanical STEP models.
- Close the 3D Viewer **File->Exit**
- Individual components or groups of select components may be viewed in the Allegro Editor, locate **C12** and place the cursor over the symbol. **RMB->3D View**



## STEP Model Support in Allegro 16.6



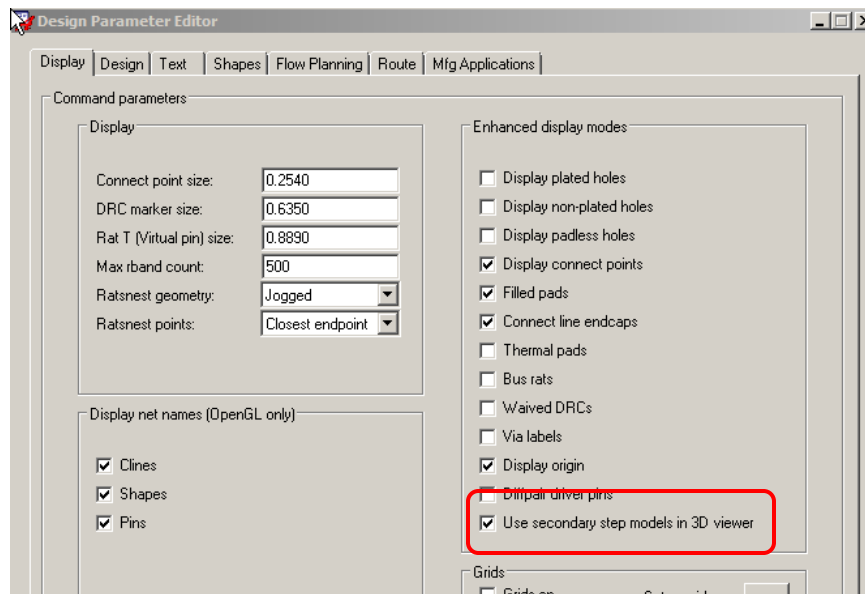
- The 3D Viewer opens and displays the STEP model for this package symbol. The STEP model and pad stack (if set to be visible) are displayed in the 3D Viewer.



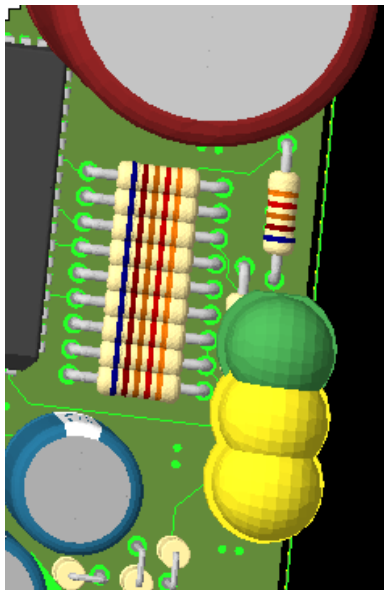
- Close the 3D Viewer.
- To view Secondary STEP models in the 3D view:
  - In the Allegro PCB Editor menu select **Setup->Design Parameters...**

## STEP Model Support in Allegro 16.6

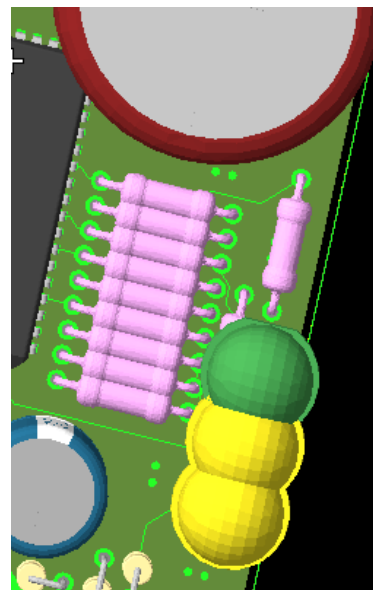
- In the **Display** tab, enable the **Use Secondary step models in 3D viewer** checkbox.



- Select the **OK** button.
- Launch the 3D viewer. Notice the axial resistors are now viewed using the **Secondary STEP** model.



Primary STEP model view



Secondary STEP model view

- **Close** the 3D Viewer
- **Save** the board drawing as **mapped1.brd**
- **LAB 4 is Complete**

---

# MODULE 4: Mapping Mechanical STEP Models.

---

## Overview

Viewing STEP models in the Allegro 3D Viewer provides a more accurate representation of on-board symbols. The user can view component placement and visually determine one component's proximity to another to decide if a violation may occur. The ability to view other objects, such as shields and housings is possible if those objects are represented by STEP models. Allegro STEP model support provides the ability to map these other type objects. Once mapped to the board design, the 3D viewer will display these models where the user can then visually check for any collisions or other component placement issues.

## Mapping Mechanical STEP Models

Mechanical STEP models must be placed within the path defined by the *steppath* Allegro env definition. The STEP Package Mapping tool will list the mechanical STEP model as an entry in the *Available STEP Models* list.

The mapping tool contains two buttons, *Add Mech...* and *Delete Mech...*

**Add Mech...:** Creates a board or mechanical symbol that represents the mechanical model (enclosure) that the STEP model is mapped to. This board symbol used for mapping is placed on the board drawing origin. The offset values defined in the mapping tool position the enclosure STEP model onto the proper location and orientation in the board drawing.

**Delete Mech...:** Deletes the selected mechanical model from the Available Packages list and removes the mapping.

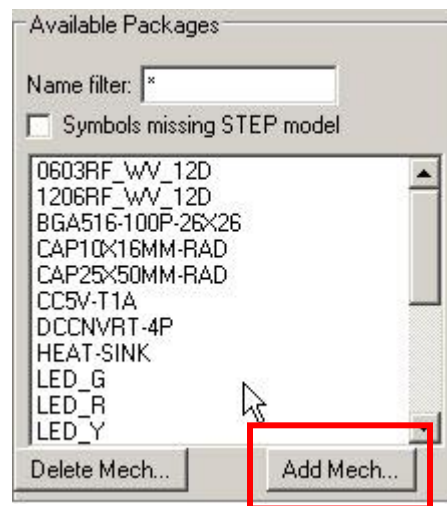


**Note:** *Labs 1,2, and 3 must be completed before proceeding.*

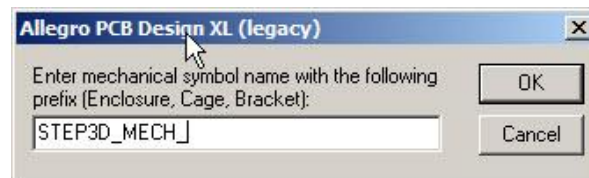
## LAB 5: Mapping a Mechanical STEP Model to a Board Drawing:

- In the *Allegro PCB Editor*, open the board drawing saved from Lab3 mapped1.brd.
- Start the STEP Package Mapping tool from the PCB Editor menu. **Setup->Step Package Mapping...**
  - In the Available Packages pane, select the **Add Mech...** button.

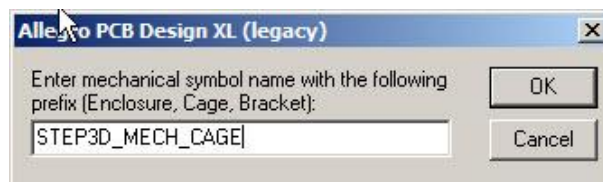
## STEP Model Support in Allegro 16.6



- A prompt will display asking to enter a name for the mechanical part. The name by of the model is prefixed with **STEP3D\_MECH\_** and cannot be modified. The value entered into the name field will be appended to the prefix.



- Append cage to the name field and then select **OK**.

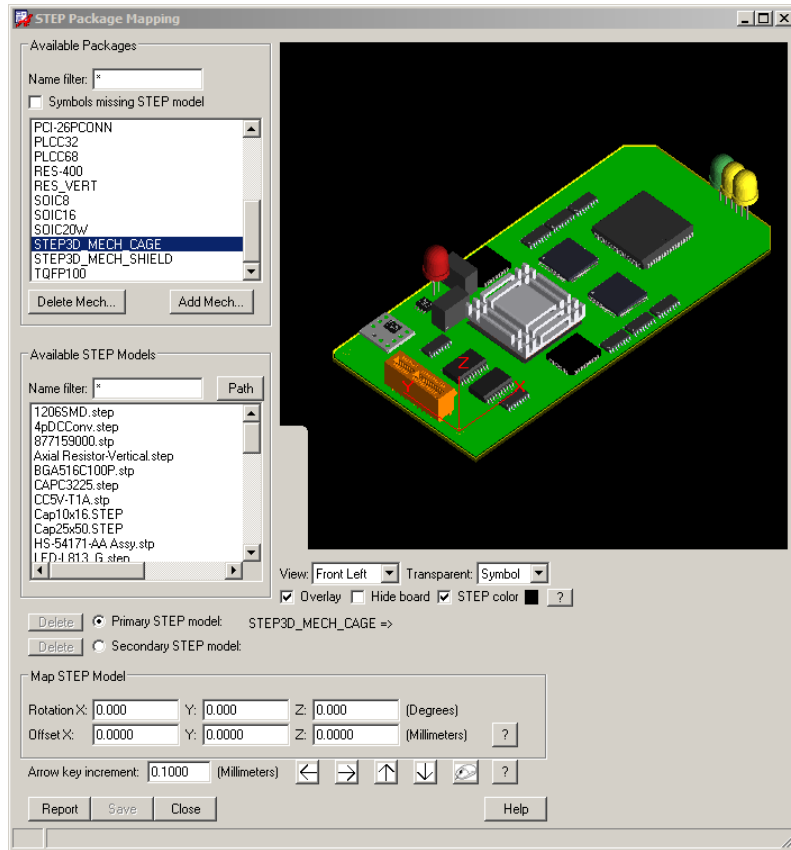


- The mechanical model name is now included in the Available Packages list.
- Select the check box next to the **Symbols Missing STEP model** in the Available Packages pane. The newly created mechanical model entry is shown as not having a STEP model associated to it.

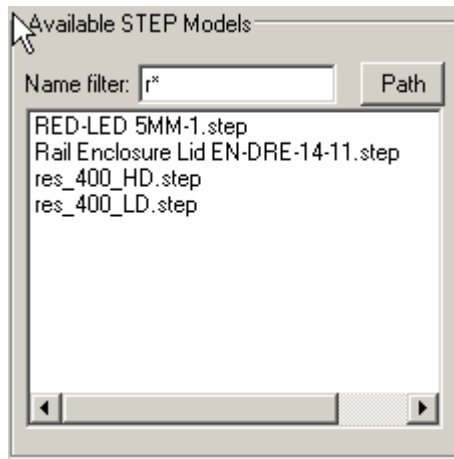


## STEP Model Support in Allegro 16.6

- Select the **STEP3D\_MECH\_CAGE** entry in the list. The board drawing is now imported into the Package definition view window.

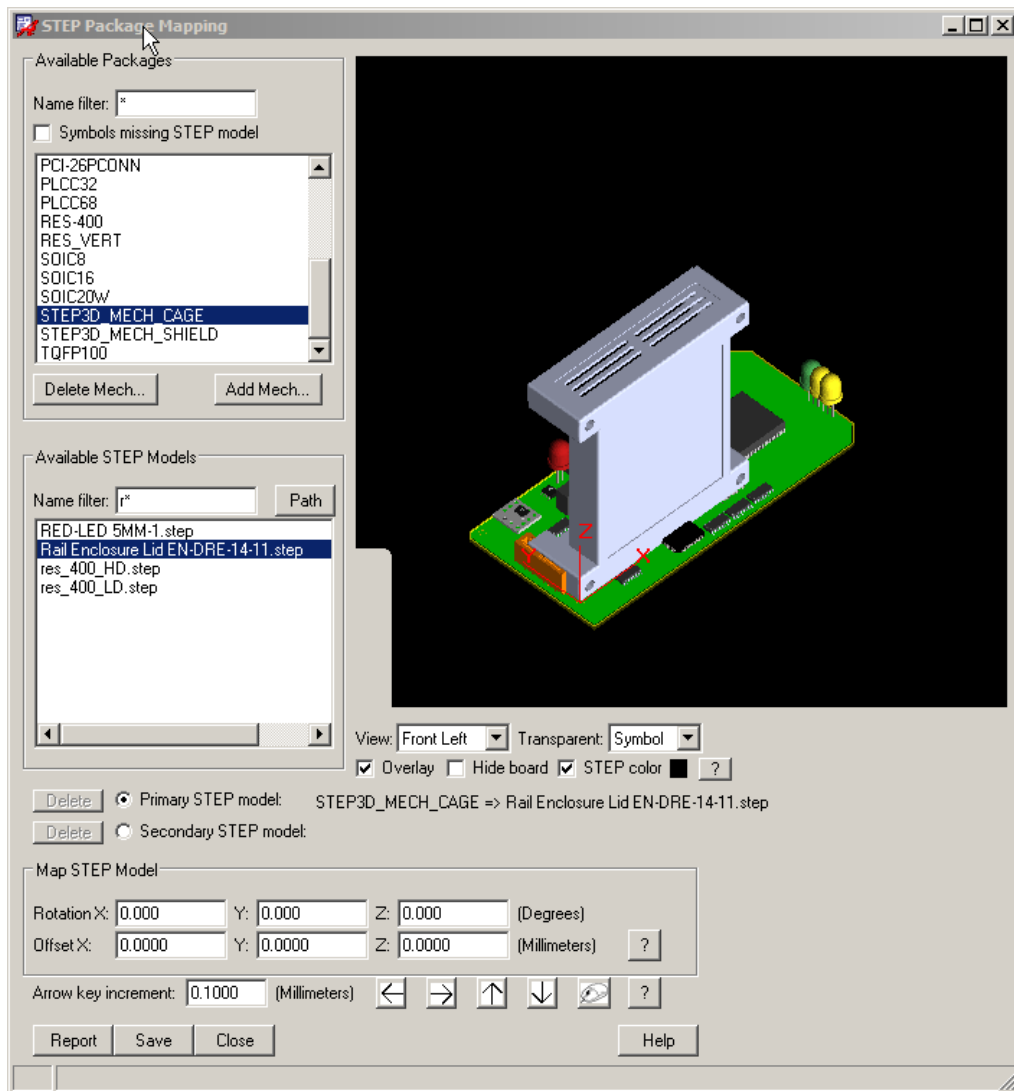


- In the *Available STEP Models* pane **Name filter** field, enter **r\*** (Tab)



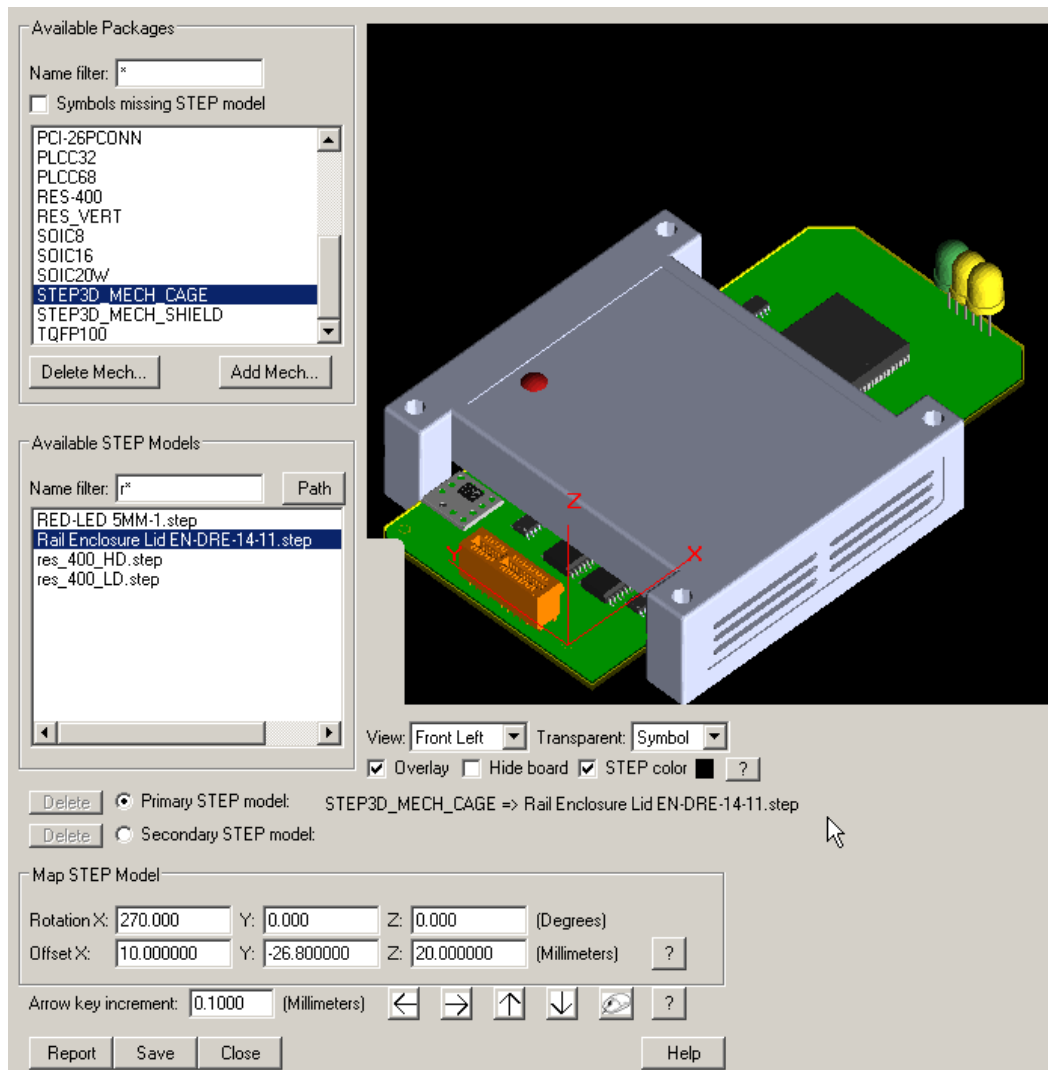
- In the filtered list results, select **Rail Enclosure Lid EN-DRE-14-11.step**. The mapping tool graphics will display with the selected STEP model place in the design at the board drawing XY Origin, TOP layer

## STEP Model Support in Allegro 16.6



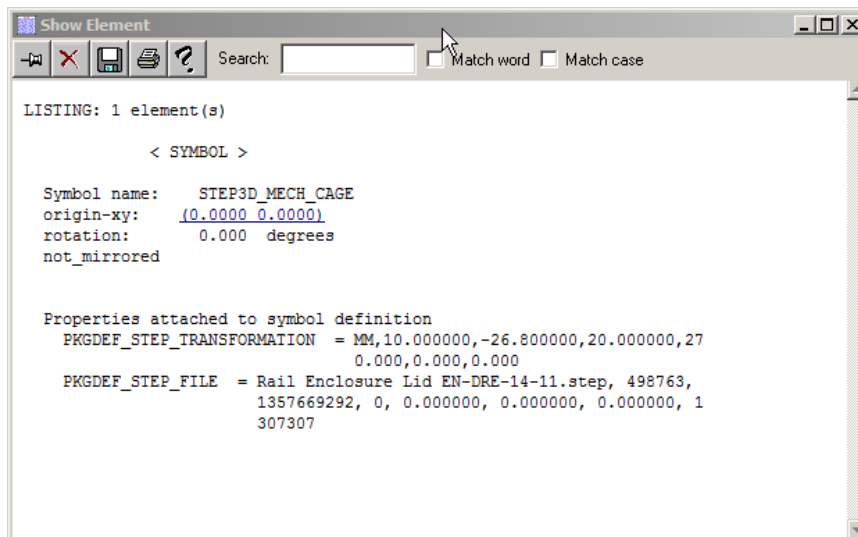
- Using the Rotation and Offset fields, position the mechanical model on the PCB. Suggested values are:
  - **Rotation X: 270.0**
  - **Offset X 10.00 (tab) Y: -26.8 (tab) Z: 20.00 (tab)**, or using the **Top**, **Front** and **LEFT** views, use the cursor to drag the case into proper alignment.

## STEP Model Support in Allegro 16.6

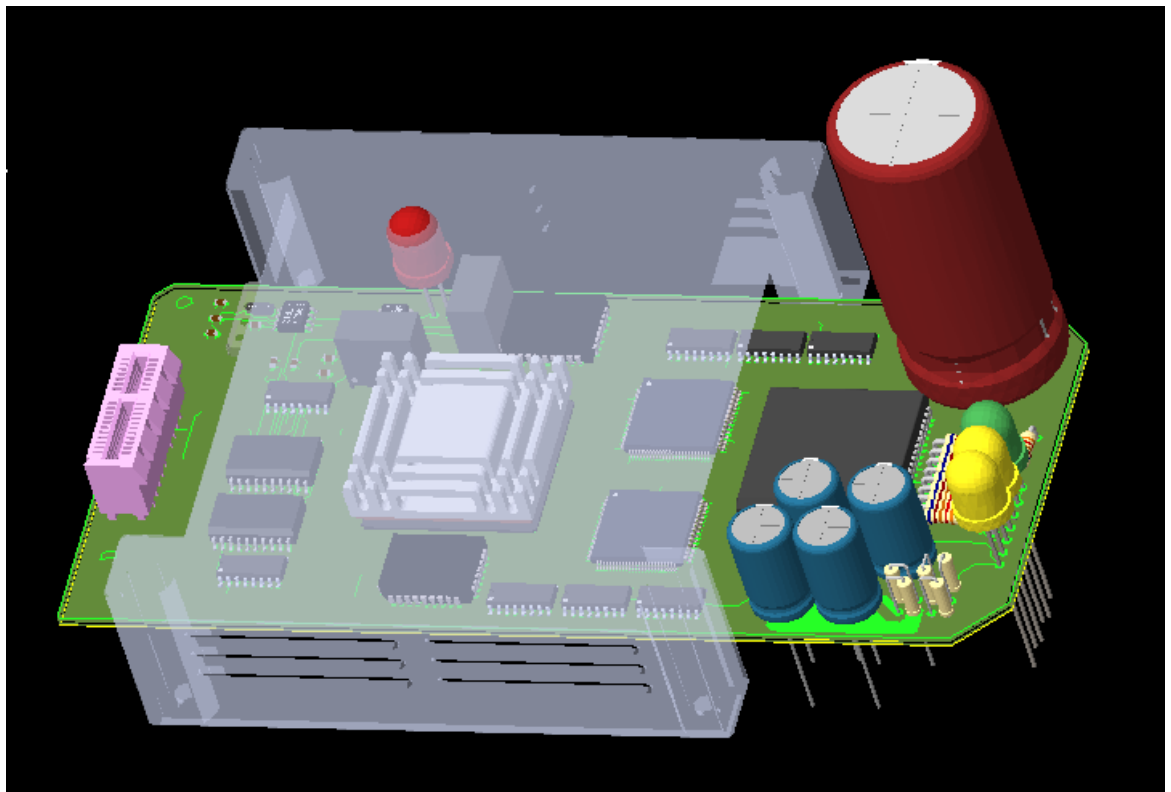


- Once satisfied with the position of the mechanical model, select the **Save** button to preserve the offset.
- The mapping tool may now be closed.
- A board symbol with the name **STEP3D\_MECH\_CAGE** is placed on the drawing origin. Using show element will display the symbol details and mapping properties.

## STEP Model Support in Allegro 16.6



- Launch the 3D Viewer, The mechanical model is now mapped into the 3D Viewer



- Exit the 3D Viewer.
- Save the Board drawing as ***Full\_Mapped\_board***
- ***Lab 5 is complete.***



---

# MODULE 5: Exporting a Board Drawings to a STEP Model

---

## Overview

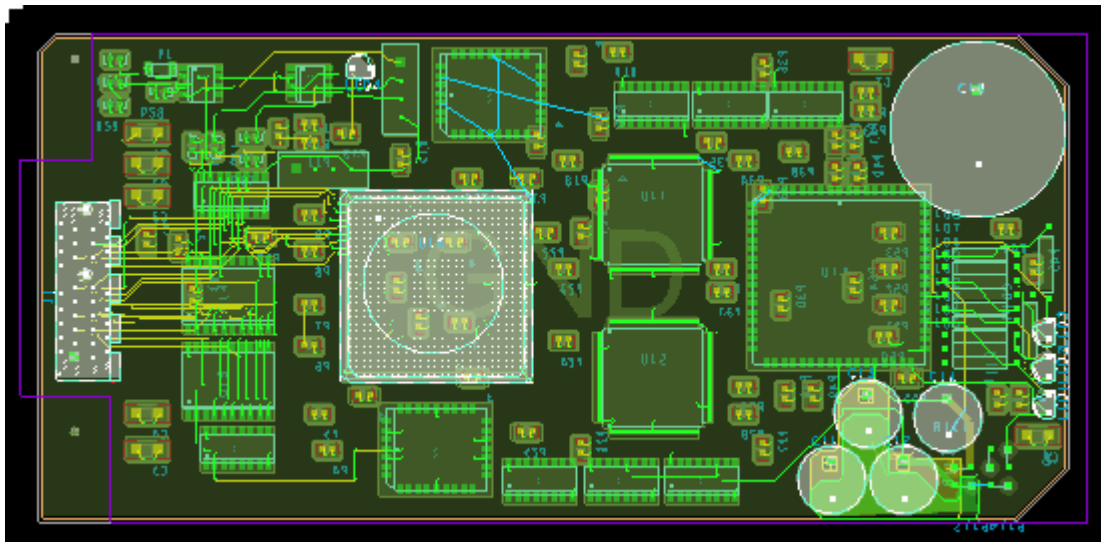
STEP models are used in various ways in the mechanical design environment such as checking for form and fit. Allegro STEP model support provides the ability to export an Allegro board drawing as a STEP model for use in a mechanical design environment.

STEP model export supports AP203, AP204 and AP242 protocols, standard units, and various output option to minimize or maximize STEP model data. **Potential for very large STEP model files exists when exporting STEP Model Parts and external copper data.**

In preparation for this lab, the user must have completed Module 1 and Module 2 labs. The board drawing to be used in the lab was created in Module 2.

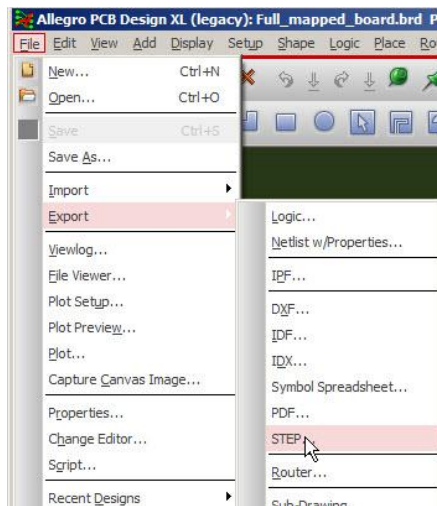
## LAB 6: Exporting Board STEP model

- Open the Allegro PCB Editor and load the board drawing created in the Module 2 lab: **Full\_Mapped\_Board.brd**
- Using the General Edit mode select and highlight J1, LED1-LED4, U16, C10-C14 as shown below.

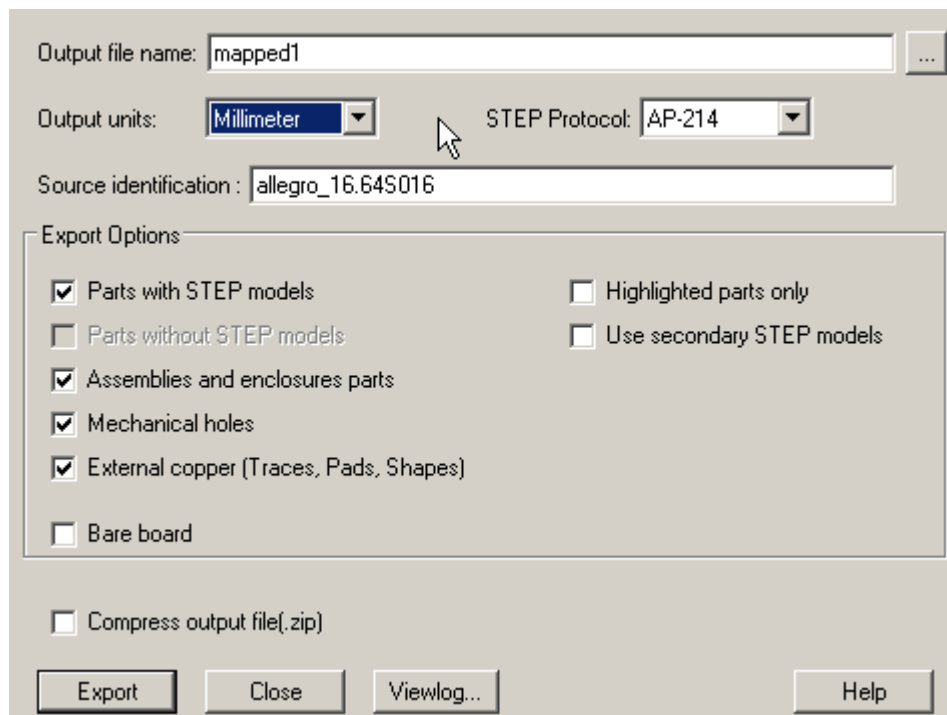


- From the Allegro PCB Editor Menu, select **File->Export->Step**.

## STEP Model Support in Allegro 16.6

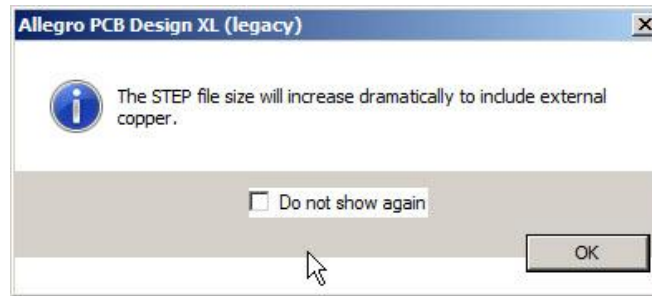


- The STEP Export form opens



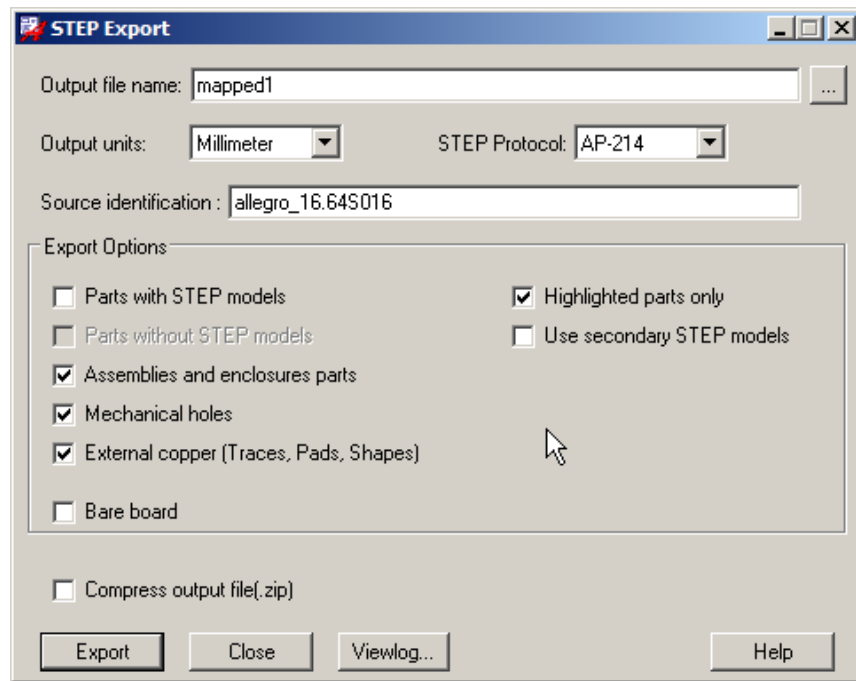
- **Output File Name:** The name is defaulted to the current open drawing name. The user may specify the path and file name for the STEP file to be written.
- **Output Units:** the user specifies the units to be used for the STEP model export. The three available values specified by the STEP model standards and Protocol are **Millimeter**, **Micron**, and **Inch**. The default value is based on the current drawing user units.
- **STEP Protocol:** There are three possible output protocol formats available, **AP-203**, **AP-214**, and **AP-242**. *AP-214 is the recommended default.*
- **Source Identification:** The source identification field is data stored within the STEP model to identify the tool of origin within the STEP model data. The current release of Allegro is the default value.
- **Export Options**

- **Parts with STEP Models:** Includes the STEP models mapped in the current drawing.
- **Parts without STEP models:** If symbols without mapped STEP models exist in the drawing, the symbols are exported as defined by the PACKAGE GEOMETRY/PLACE\_BOUND\_TOP/BOTTOM.
- **Assemblies and Enclosure Parts:** Exports the STEP3D\_MECH models created in the STEP Package Mapping tool.
- **Mechanical Holes:** Includes export of mechanical hole defined in the current board drawing.
- **External Copper (Traces, Pads, Shapes):** Export external traces, pads, and shapes on the ETCH/TOP and ETCH/BOTTOM layers. When this option is selected, the user is warned that the STEP model to be export may become very large.

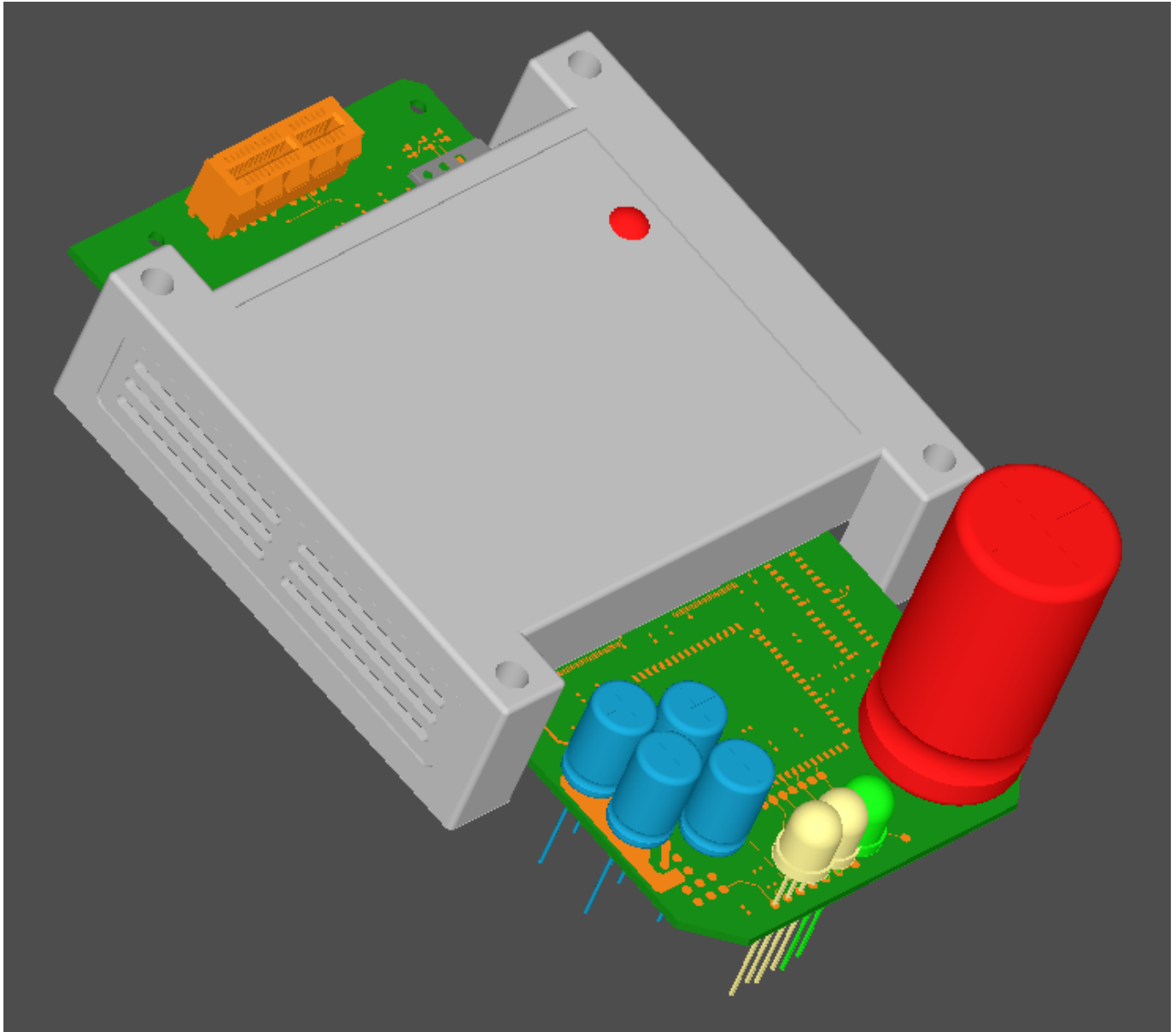


- **Bare board:** This option allows the export of the physical board, mechanical holes and external traces only. All other Export options are unselected.
  - **Highlighted parts only:** This option allows the export of only the part highlighted in the Allegro PCB Editor board. Assembly/enclosure parts, mechanical holes, external copper, and board outline will also be exported if those options are selected.
  - **Use Secondary STEP models:** This option exports STEP model mapped as Secondary STEP models associated to symbol placed in the board drawing.
- **Compress output file (.zip):** compressed the exported STEP file into a .zip file.
  - **Export:** Starts the export Process
  - **Close:** Closes the STEP Export tool
  - **Viewlog...:** Views the current STEP Export log file.
- Set the **Export Options** as shown below

## STEP Model Support in Allegro 16.6



- Select the **Export** button
- When the export process is complete select **Viewlog...** to see a summary of the export process.
- **Close** the Export Form.
- **Close** the Allegro PCB Editor.
- The exported STEP file **mapped1.stp** is saved in the current working directory. If a STEP model viewer is available, this file may be loaded into the viewer.



(Above) Exported Allegro STEP model image viewed from a free STEP model viewer

- Lab 6 is complete.

***Workshop is complete***