STEP Model Support in PCB Editor

Overview

All versions of PCB Editor (OrCAD and Allegro) allow users to see their board design in a 3d canvas. This methodology uses STEP models that are mapped either directly in the library (associated with the filename.dra) or assigned directly in the board file (filename.brd) itself. You can also define mechanical components (covers, brackets, enclosures) and assign step models to these parts in the board.

This note will cover:

- **Map PCB Editor symbols to STEP models**

- **Use new STEP model features in the 3D Viewer**

- **Associate enclosure STEP models for 3D Viewing**

- **Exporting a PCB Editor board as a STEP model**

STEP Models

STEP models are XML formatted files that describe graphical detail for a physical part. There are currently three STEP model formats that support most electrical and mechanical parts used on a PCB.

- AP203 - Aerospace's STEP AP203 - Configuration controlled 3D design.
- AP214 - Automotive's STEP AP214 - Core data for automotive mechanical design processes.
- AP242 - Merging of 2 ISO standards, AP203 and AP214. AP242 is designed modular, with regular increments in order to bring added value to users. A first edition (AP242E1) brings a first set of new features (overall for PMI, tessellation, external element references). A second edition (AP242E2) is in preparation to complete the case coverage.

Obtaining STEP models is possible through manufacturer sites, but not all manufacturers provide STEP models. There are several 3rd part library solutions that, for a subscription or fee, will create and distribute STEP models. Some MCAD tools provide the capability to export STEP models of their libraries and their 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. A recommended free site (although you must register) for 3D Step Models is: - http://www.3dcontentcentral.com/Default.aspx

See the following web sites more information on STEP models:

http://www.prostep.org/nc/en.html

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

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 a PCB Editor board STEP model.

STEP Model Library

STEP models should be placed into a Library in a similar manner as other 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 PCB Editor Library path environment steppath. The steppath environment variable may be defined through Setup > User Preferences > Paths > Library setting in the PCB Editor.

Add Directory
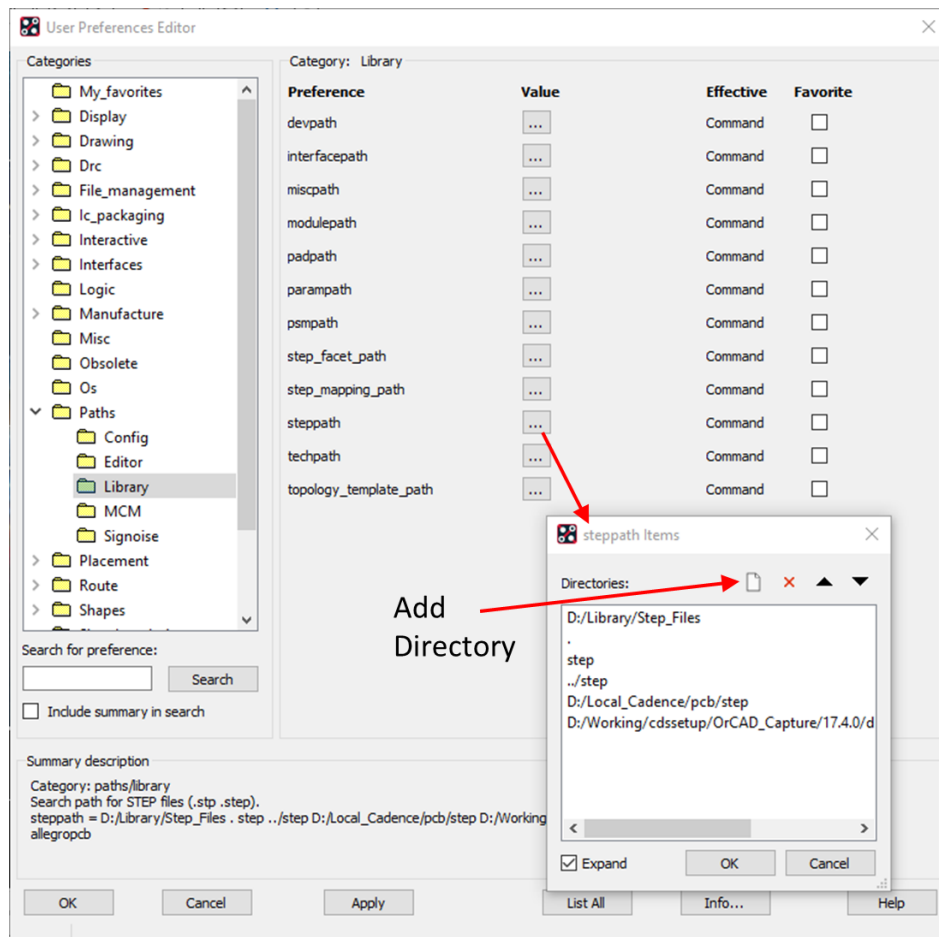
Symbol to STEP Model Mapping

STEP models are associated to PCB Editor package and mechanical symbols through a mapping tool. This mapping tool sets the STEP model name to the symbol and defines offset information to correctly position the STEP model in the 3D canvas. The mapping data created is then instantiated into the symbol as a property.

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 model may require adjustment in rotation about the X, Y, and Z axis. Proper positioning of the 3D model may also require adjustments in the X, Y, and Z axis. The STEP model mapping tool assists the user in defining these offsets by visually comparing the PCB Editor 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 PCB Editor 3D canvas, the board or package symbol MUST contain a PACKAGE GEOMETRY PLACE_BOUND_TOP or PLACE_BOUND_BOTTOM element.

Mapping a STEP model to a symbol

This can be completed directly in a board file, package or mechanical symbol. It is recommended that this process is done in a symbol since every time you use that footprint the step model with already be mapped. The example below goes through mapping directly in a board file.

Open PCB Editor then go to Setup > Step Package Mapping, the STEP Package Mapping tool opens
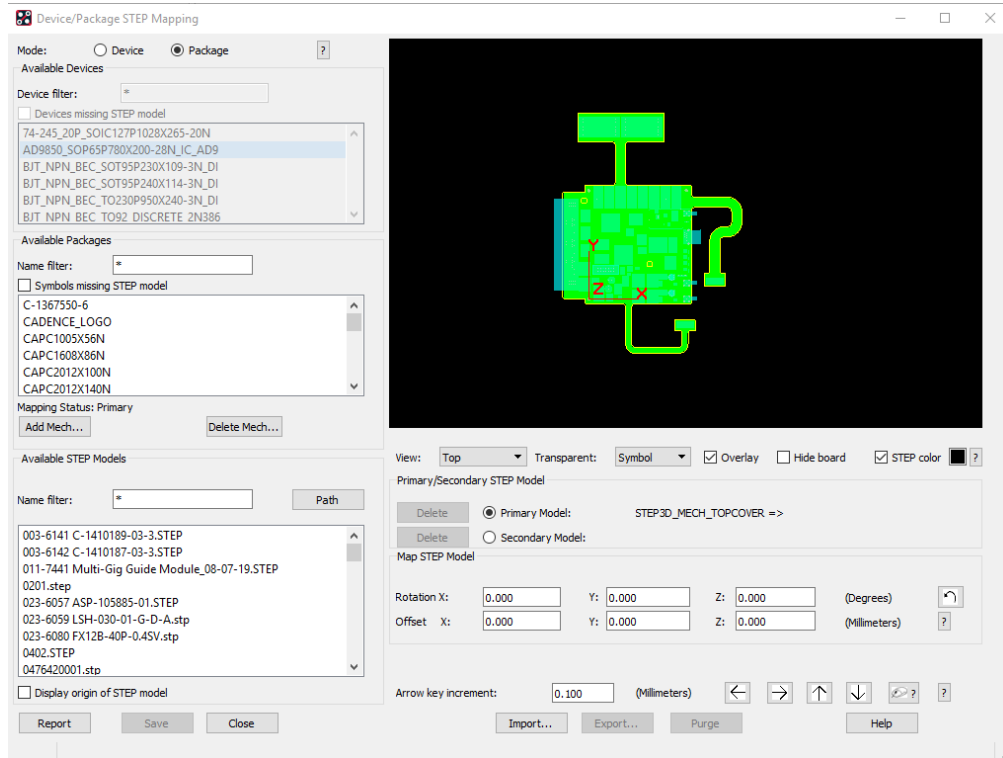
Only available if this is launched from within a Board file.

Mode - Choose the mapping mode as follows:

- **Device -** saves the mapping properties on the component definition.
- **Package -** saves the mapping properties on the symbol definition

Available Devices - Displays list of the devices present in the current design.
Device filter - Filters the devices by name.
Devices missing STEP model - Filters the devices that are unmapped.
Available Packages - Displays list of the symbols present in the current design.
Name filter - Filters the symbols by name.
Symbols missing STEP model - Filters the symbol that are unmapped.
Mapping Status - Specifies the mapping status of the selected symbol as Primary or Secondary.

Available in both a board file and symbol file.

Add Mech - Creates a board or mechanical symbol that represents the mechanical model (enclosure).
Delete Mech - Deletes the selected mechanical model from the Available Packages list and removes the mapping.
Available STEP Models - Displays lists of all STEP models available in the library path specified by the steppath environment variable.
Name filter - Filters the STEP models by name.
Path - Provides information of the steppath environment variable value.

Display origin of STEP model - Displays the origin of the STEP models in the graphics pane.

Graphics Pane - Displays the package/device model and STEP model graphics with the board section.

View - Specifies different modes of viewing a STEP model in the graphics pane.

Transparent - Specifies the transparency mode for the STEP model in the graphics pane.

Overlay - Merges the package and STEP model views into a single view.

Hide board - Removes the graphical image of the board in the viewer.

STEP color - Uses the colors defined in the STEP model, if enabled.

? - Displays the colour legend for the graphics.

Delete - Deletes primary STEP model associated with the selected device or package.

Primary model - Choose to specify the primary mapping for the selected device or package.

Delete - Deletes secondary STEP model associated with the selected device or package.

Secondary model - Choose to specify the secondary mapping for the selected device or package.

Rotation X, Y, Z - Specifies the rotation in degrees in X, Y and Z directions to correctly position the STEP model in relation to device/package.

Offset X, Y, Z - Specifies the offset in Millimeters in X, Y and Z directions to correctly position the STEP model in relation to device/package.

Arrow key increment - Specifies the increment value in Millimeters for arrow key movements.

Arrow Buttons - Moves STEP model Up/Down/Left/Right by increment value in the 2D view (Top, Bottom, Front, Back, Left, Right).

Mouse Button - Enables the mouse buttons actions for moving, panning and zooming the STEP models in the graphical pane.

? - Displays information on arrow and mouse button actions.

Report - Creates a report of symbols and devices in the working directory. The HTML file includes mapping status, STEP model names, rotation, and offset values.

Save - Saves the mapping data of the current symbol or device displayed in the graphics pane.

Import - Imports the mapping data from a STEP device/package mapping file (.map) into a design and attach the STEP facet files into the design for 3D display. Before importing, extract the STEP facet files into the <working_directory>/stepFacetFiles4Map/.

Export - Saves the mapping data into an XML format to a file (STEP device/package mapping file(.map)) for re-use or import at the location specified by the step_mapping_path environment variable. Also saves the facet representation of the STEP geometry and assembly for 3d display into a .zip file (stepFacetFiles4Map.zip) at the location specified by the step_facet_path environment variable.

Purge - Deletes STEP device/package mapping information and the STEP facet data from the database. This option saves the mapping data to a file (.map) and the STEP facet files (stepFacetFiles4Map.zip) into the working directory.

**Close -** Close the STEP Package Mapping dialog box without saving the mapping.

To map a 3D STEP model in a board file from the Available Packages list, scroll to locate and then select the required package symbol name to map a 3D STEP model to or select the required symbol in PCB Editor and with the mapping tool open, navigate in the drawing to locate the required symbol and left-click on it. The graphics for this are displayed in the display panel.
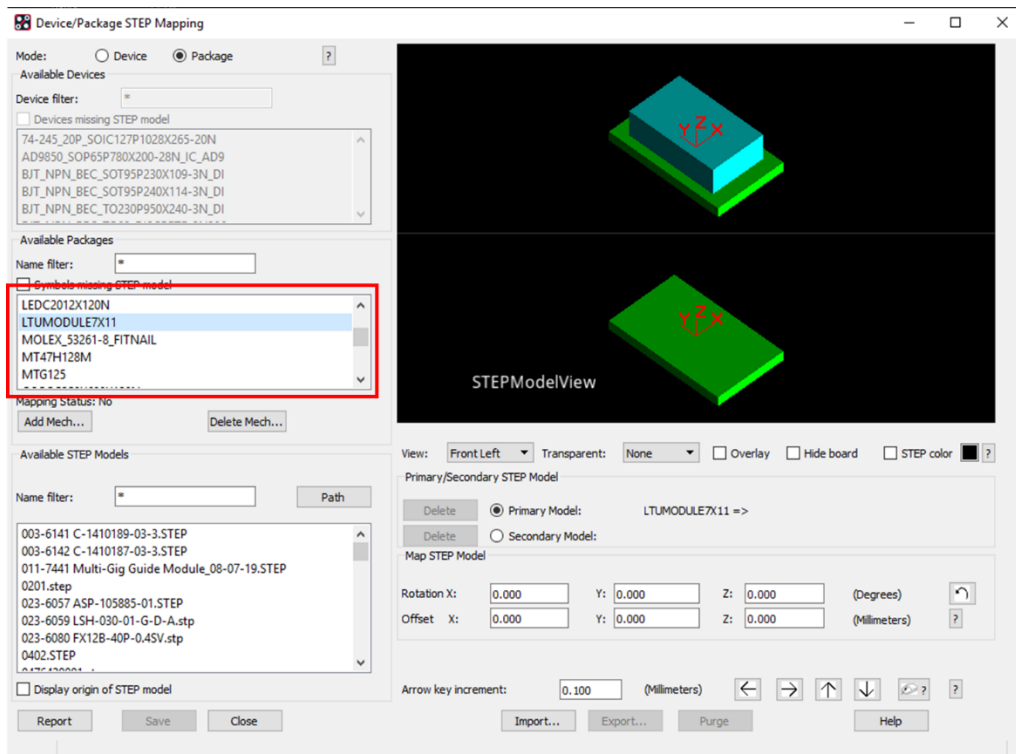
In the Available STEP model pane list, scroll to locate and then select the required STEP model name. The STEP model graphics are now displayed. This is an example of an AP-214 STEP model. Notice the detail and colour features that are available. If the STEP model is one single color this is normally an example of an AP-203 STEP model, some detail but one single colour. You can change the colour if the model is in AP203 format using the Step Colour option. This color is derived from the STEP model.
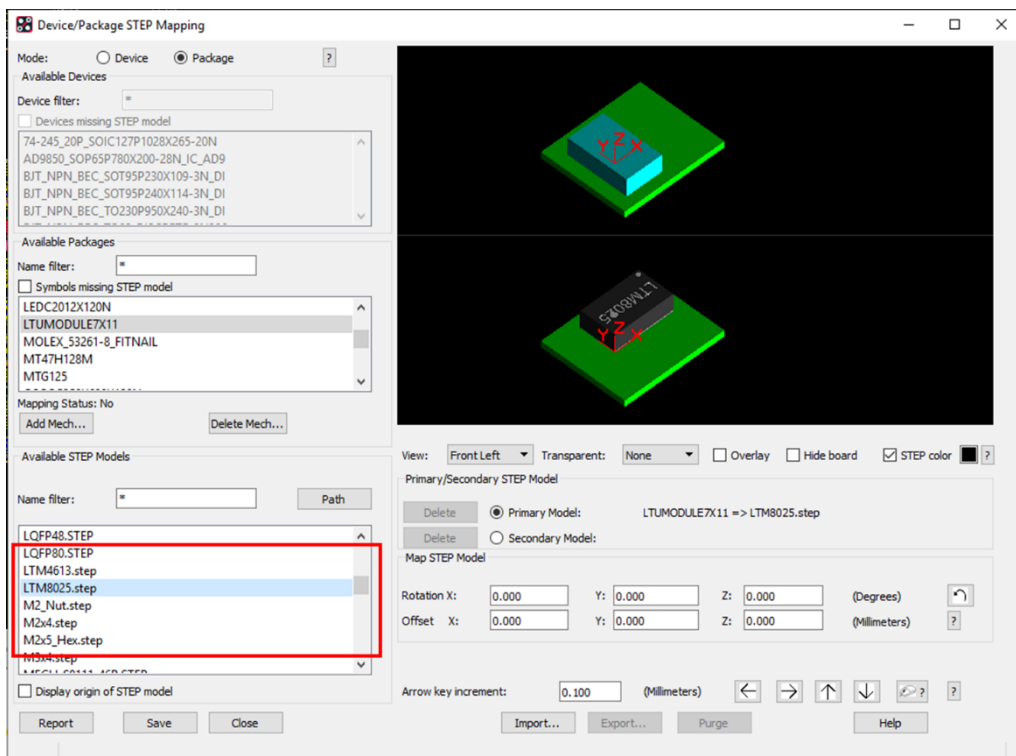
At first glance we can see that the symbol origins are not the same. To further observe the differences, first set the Transparent value to **Symbol**, then set the view to **Top** and notice how the packages differ in their origin.
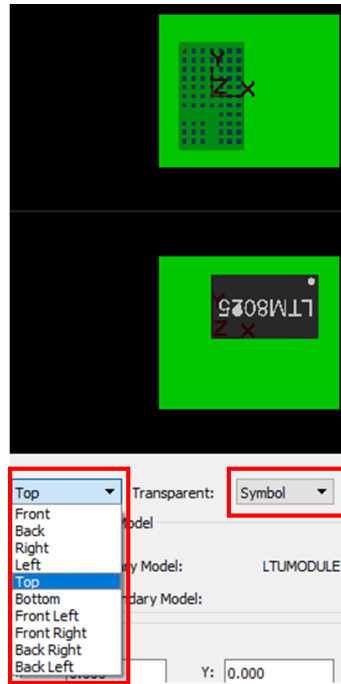
Now overlay the two symbols by selecting the **Overlay** checkbox to further see the differences.
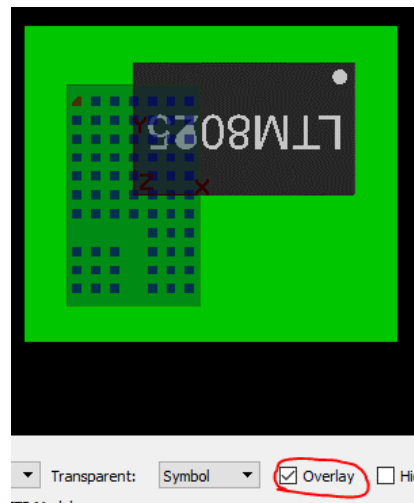
In this view, we can see that the X and Y offset values need to be set. When Setting offset values, use the **Tab** key to complete the entry, or using a **left click**, select the next field. **DO NOT use the Return key as this will close the Mapping tool.** The STEP model can be mapped either by entering values or using the arrow key increment buttons at the bottom of the Map STEP model area.
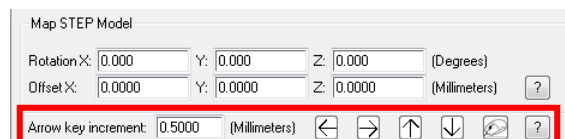
Enter a value for the increment and move the part in the required direction by clicking on the relevant arrow. If you wish to move the STEP model manually select the Mouse button icon then click and drag the STEP model to its required location. Then if required you can fine tune the values for Offset afterwards. For this part the values are Rotation X, Y 0, Z 90 and Offset X 4.5, Y -7.5, Z 0.
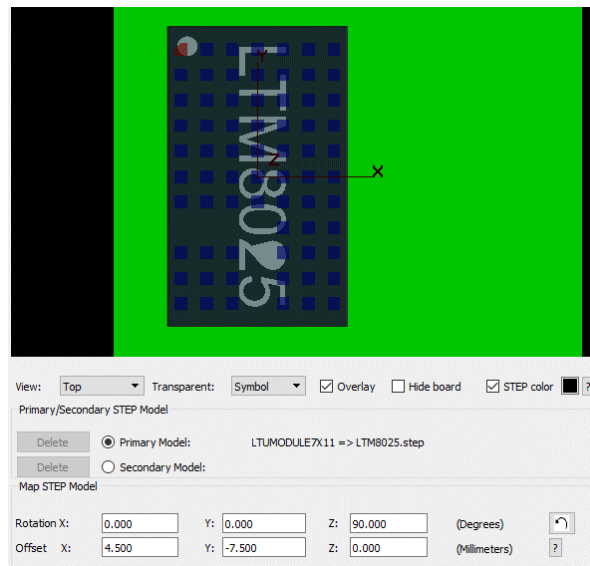
As each value was entered, the graphics displayed the change in the mapping showing the alignment of the STEP model to the package model. Change the **View** to **Front**.
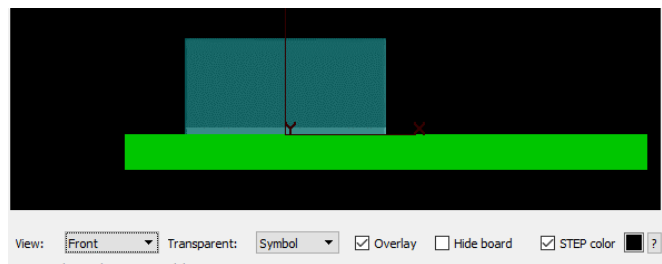
This view shows that the STEP model is properly seated on the board in the Z axis, requiring no Z Offset. (A Z Offset might be required for other models). Select the **Save** button to preserve the current mapping for this package model. Once the save is complete, the Save button is disabled. **Close** the STEP Package Mapping tool. In the PCB Editor, locate the symbol (U31 in this reference) that has been mapped and hover right click > Show element
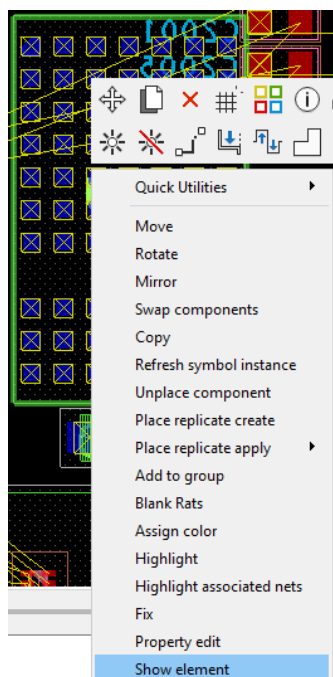
In the Show Element form, the STEP model mapping properties are displayed. These properties become part of the symbol definition and cannot be modified outside of the STEP Package Mapping tool. You can see all X, Y and Z rotation and offsets are stored as part of the package symbol.
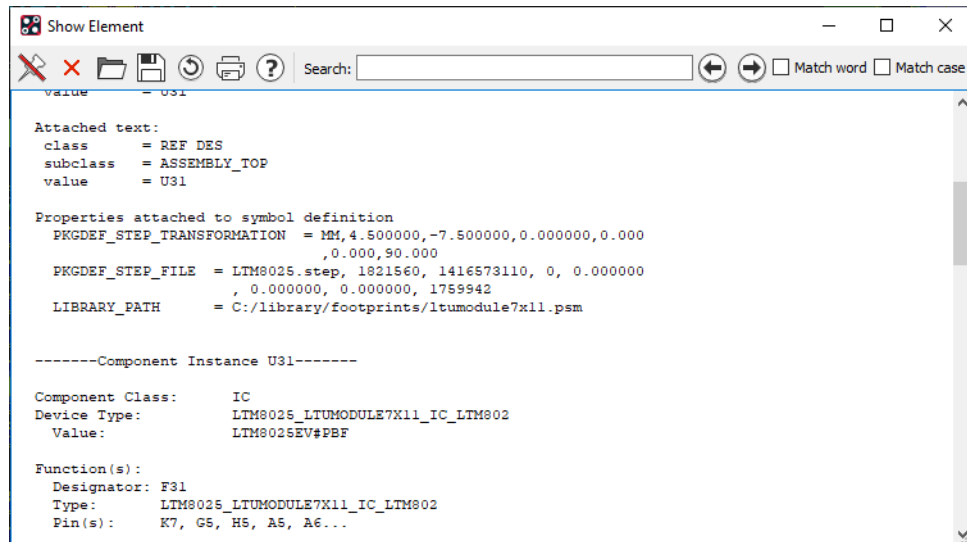
There are also options to map a Primary STEP model and a Secondary STEP model for each symbol. This gives users the ability to have a detailed and non-detailed model for each part. The secondary model is mapped by selecting the secondary step model checkbox then mapping the STEP model using the steps mentioned above.
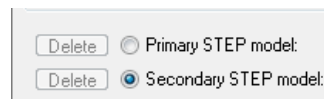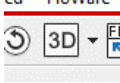
If both are defined there is an option to display the 3D view in the secondary mode (the default is to always show the Primary model view). Go to Setup > Design Parameters > Display tab and check the option for "Use secondary step models in 3D viewer" and the secondary model will be shown.
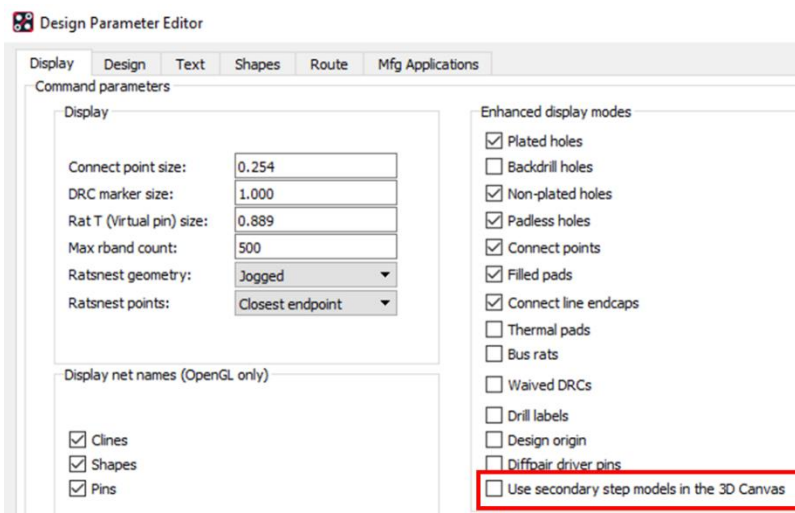
You can also open the relevant Package Symbol file (filename.dra) and repeat the steps above to map STEP Models directly to the library of package and mechanical symbols. Once complete, any drawing that loads this symbol will also have the mapped data included. This is the preferred method.

3D Canvas with STEP models

Once STEP model mapping is completed, the PCB Editor 3D Canvas will display the graphical representations of the STEP models.

Select the 3D viewer icon in the PCB Editor toolbar , or from the menu select View - 3D Canvas (Allegro) or Display > 3D Canvas (OrCAD).

You can also select the drop down arrow next to the icon to filter items for selection before opening the 3d Canvas. This can be useful depending on the design size.

When the 3D canvas 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.
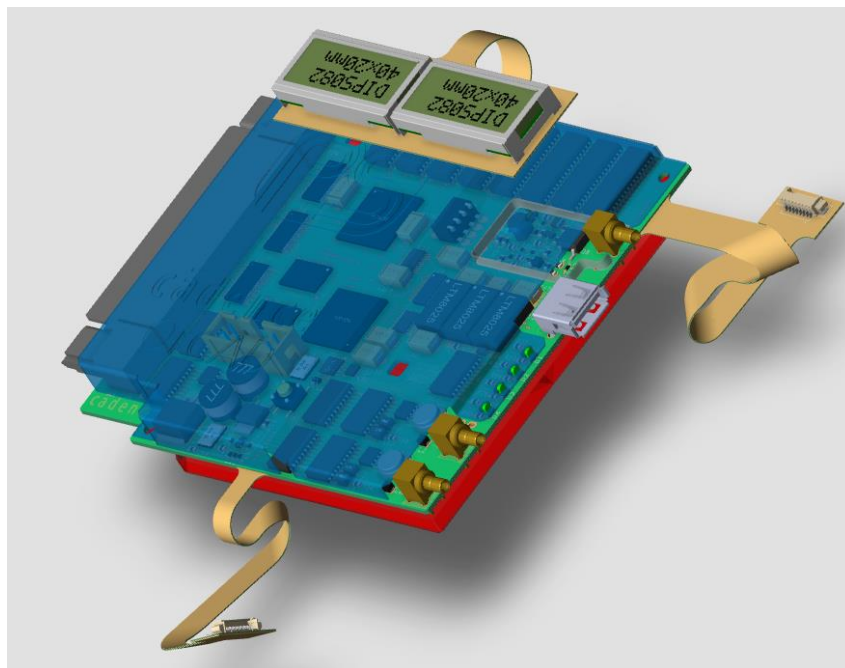
This design has mechanical models (top and bottom covers and a dimm module PCB that is mated into the connector on the back side of the PCB). We have the ability to run a Collison Detection which will compare all step models to each other. This also takes the flexi's into account when they are in a bent state (more about this later).

Go to the Collison Detection tab and choose the Calculate button to run the collision detection. Once complete the pane will list any collisions it found.
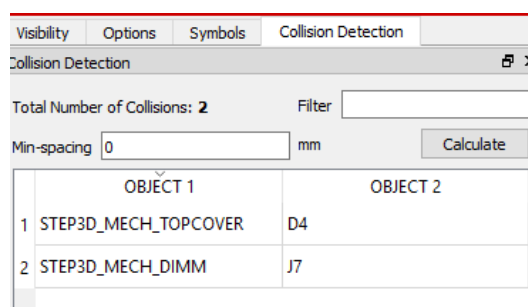
You can left click the objects and they will highlight in the 3d canvas which will help you locate the part(s).
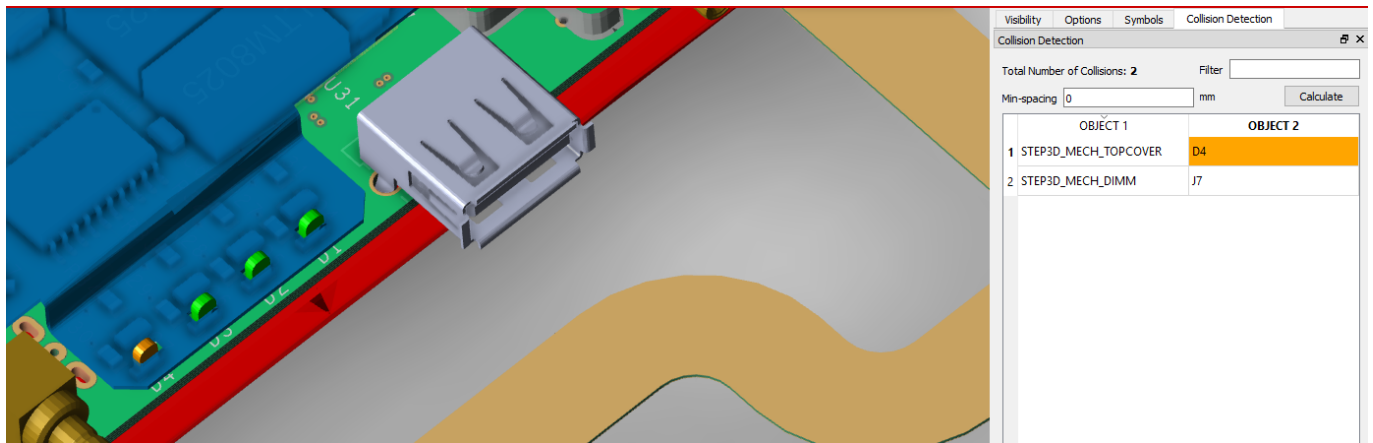

In this example the LED is too far down and is clashing with the top cover opening. A quick fix to move the LED can resolve this issue and maybe stop a transition with the MCAD team. The other issue on this design is an interference fit where the DIMM PCB does touch the connector (but this is how it fits into the connector so we can happily ignore this issue.
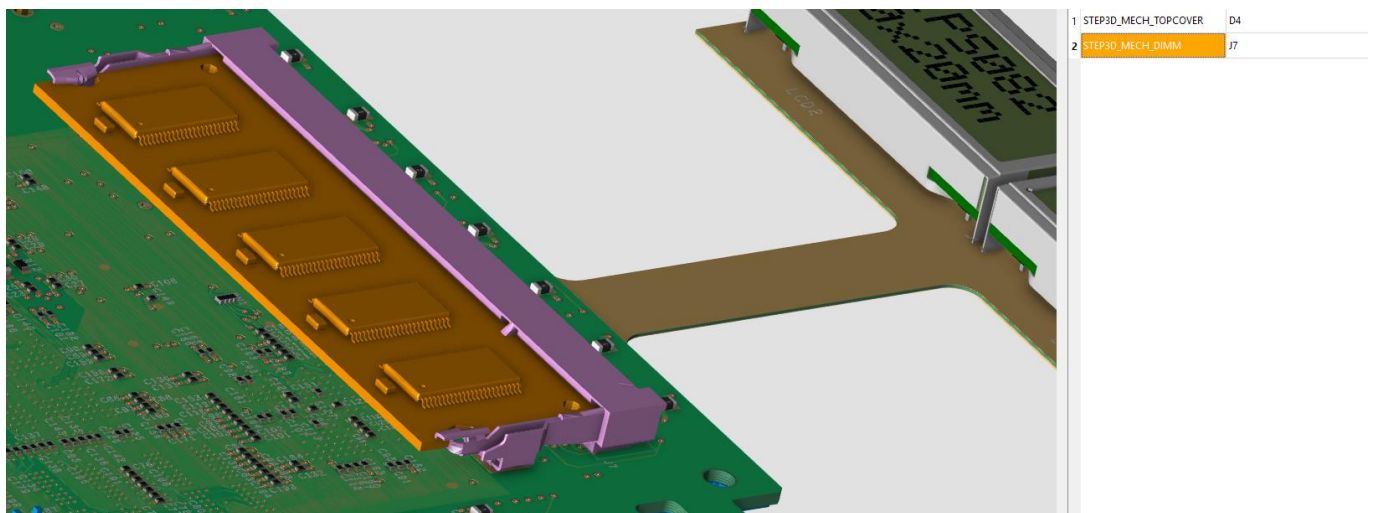
Mechanical parts can be set to be transparent if required. Look at Setup > Preferences > Symbol Representation for this and also how a symbol (footprint) is displayed.
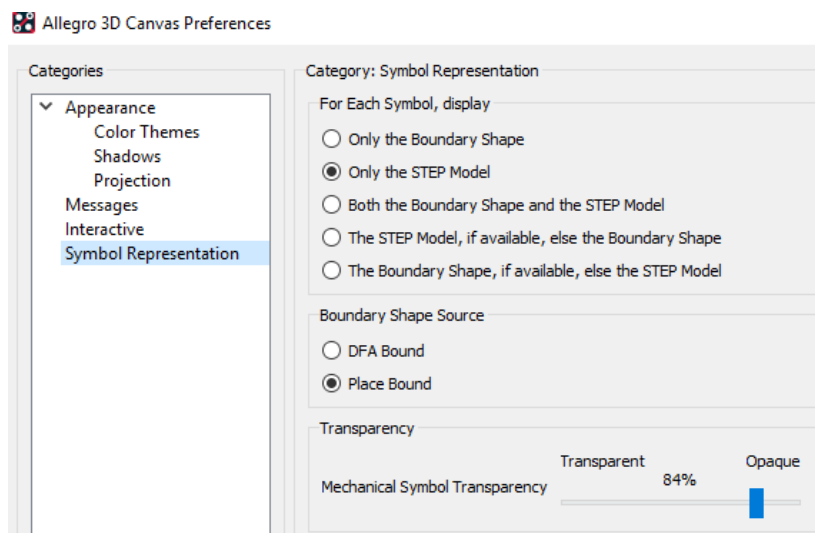
As you have seen earlier the 3d Canvas does have the ability to represent a flexi rigid design accurately. Once the zones and bends have been defined (as well as the 3d anchor) you can show them in the 3d Canvas. In the 3d Canvas window right click and choose Bend then loom at the Options Pane
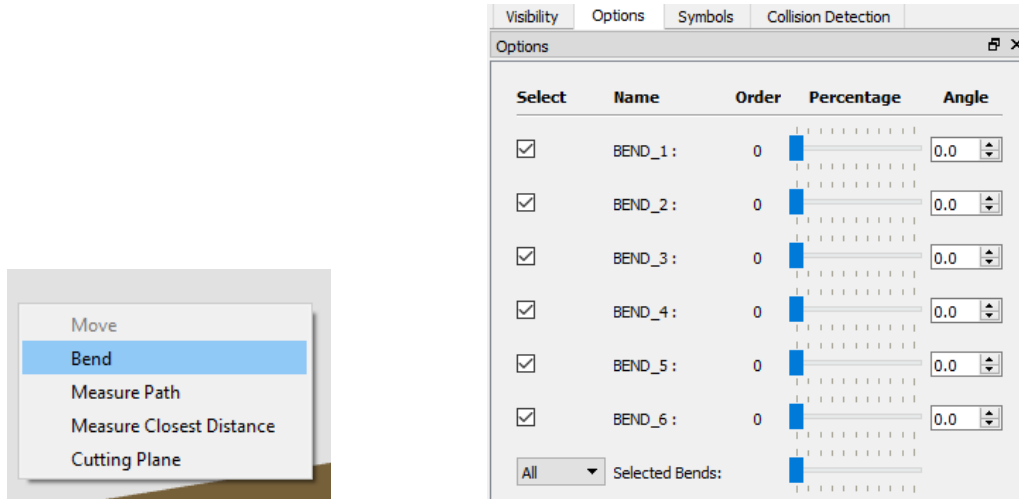

You will see a list of all the bends that are available. You can bend each one at a time, enter the angle you want to see or have them all bent in one step.

Individual components or groups of selected components may be viewed in PCB Editor, locate a symbol; place the cursor over the symbol and **right click - 3D Canvas**
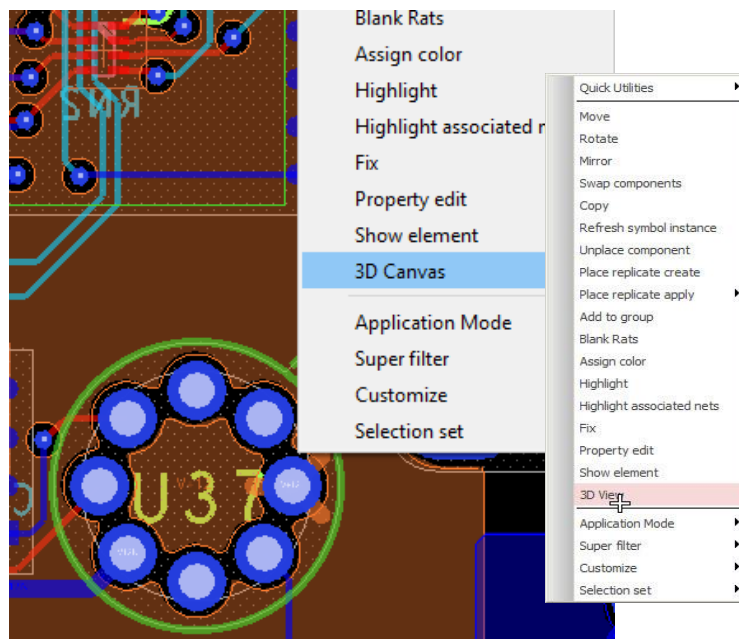
The 3D Canvas opens and displays the STEP model for this package symbol. The STEP model and pad stack (if set to be visible) are displayed in the 3D Viewer.
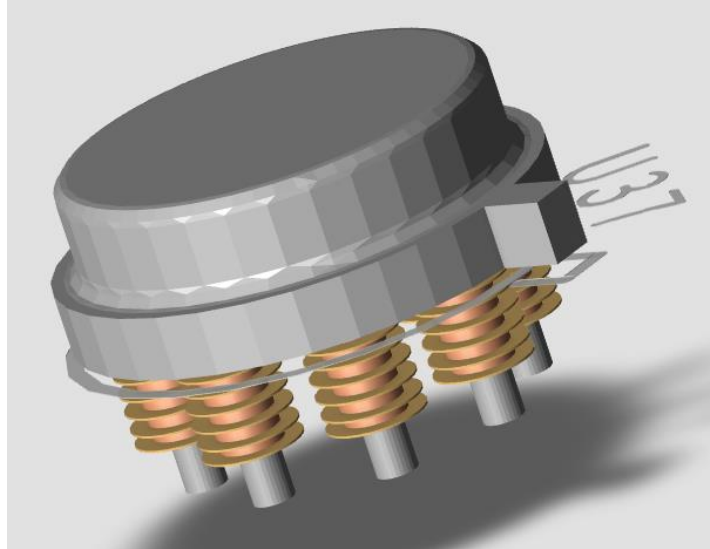
You can also view nets, clines, shapes – basically any object type can be viewed in the 3d Canvas.
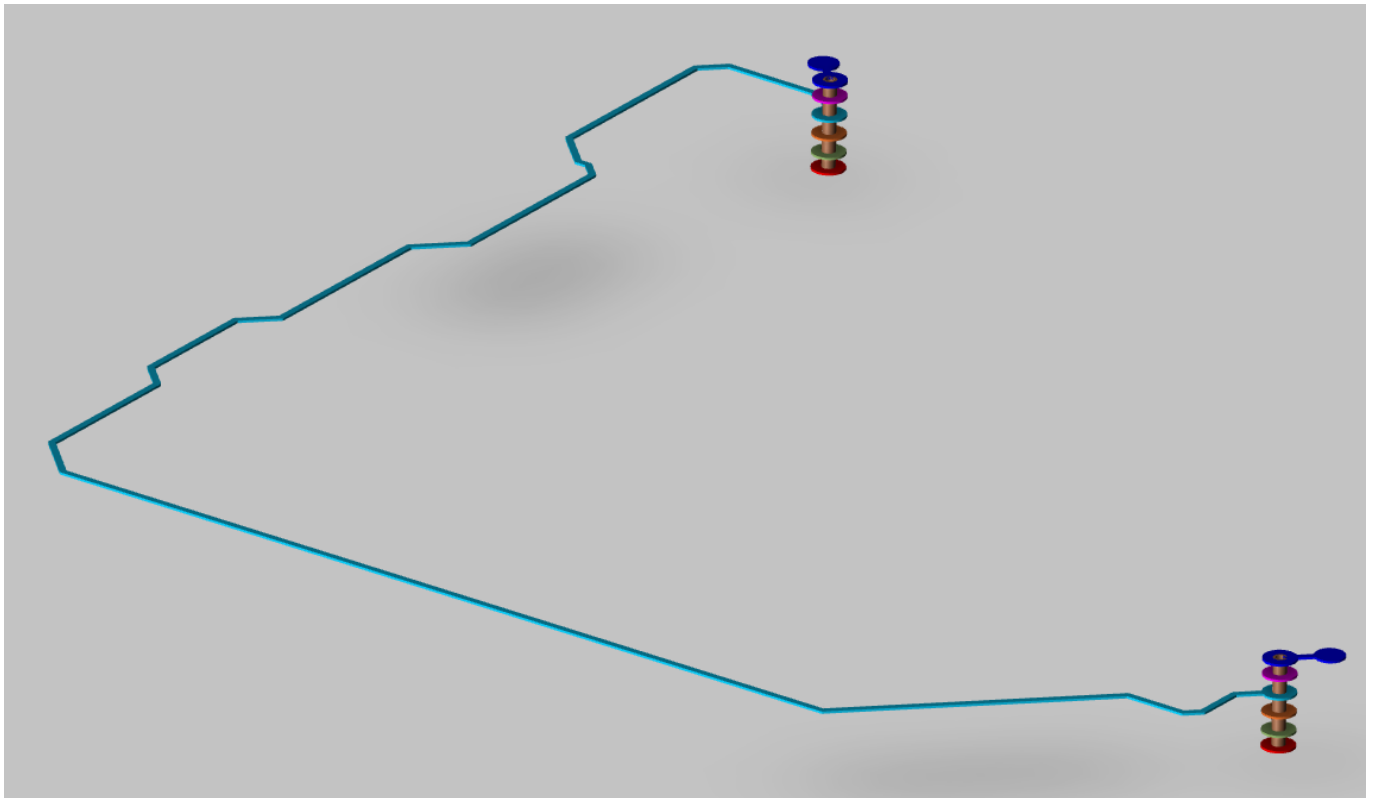
Mapping Mechanical STEP Models

Viewing STEP models in the PCB Editor 3D VCanvas 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 has occurred. The ability to view other objects, such as shields and housings is possible if those objects are represented by STEP models. PCB Editor STEP model support provides the ability to map these other types of objects. Once mapped to the board design, the 3D Canvas will display these models where the user can then run collision detections to check for any clashes or other component placement issues.

Mechanical STEP models must be placed within the path defined by the steppath (same location as your STEP Models). 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.
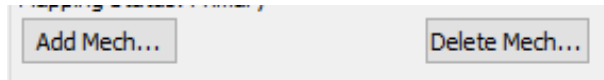
## Mapping a Mechanical STEP Model to a Board.

Open the board file that requires the mechanical STEP part. Start the STEP Package Mapping tool from the PCB Editor menu. **Setup > Step Package Mapping (Allegro) or Setup > Step Mapping (OrCAD).** In the Available Packages pane, select the **Add Mech..**. button.
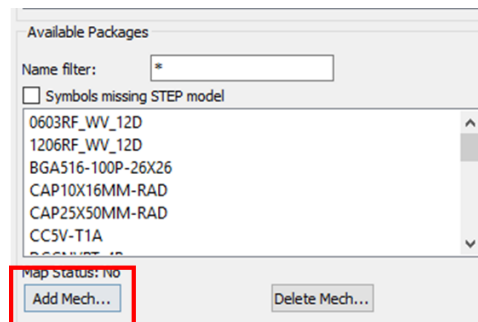

A prompt will display asking to enter a name for the mechanical part. The name of the model is prefixed with **STEP3D_MECH_** and cannot be modified. The value entered into the name field will be appended to the prefix.
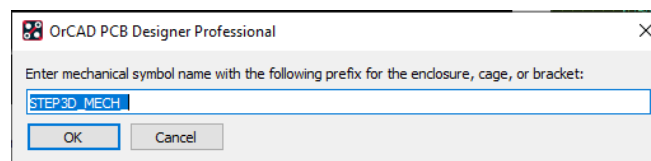

Append a suitable name, in this example cage has been added to the name field and then select OK.
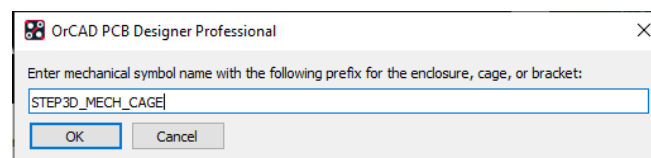

The mechanical model name is now included in the Available Packages list. You can either browse the list for the newly created package or check the box for **Symbols Missing STEP Models**. To map a STEP part, select the **STEP3D_MECH_CAGE** in the available packages list, the complete board drawing is now imported into the Package definition view window.
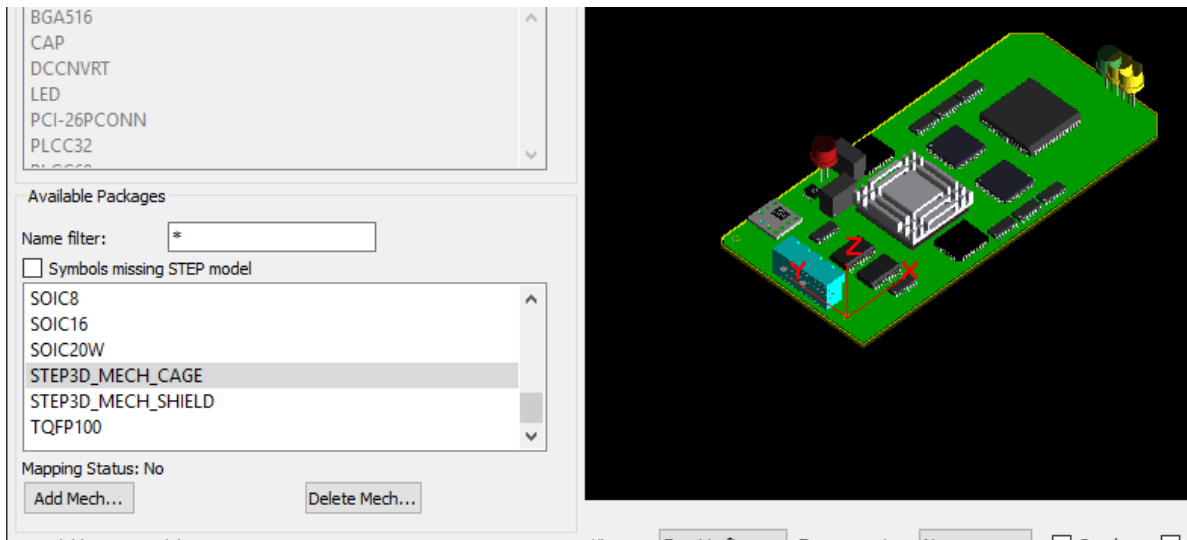
In the Available STEP Models pane locate the relevant Mechanical STEP Model. In this example we are looking for the Rail Enclosure Lid. The filters can be helpful when looking for specific parts, r* followed by a Tab key filters the list accordingly.
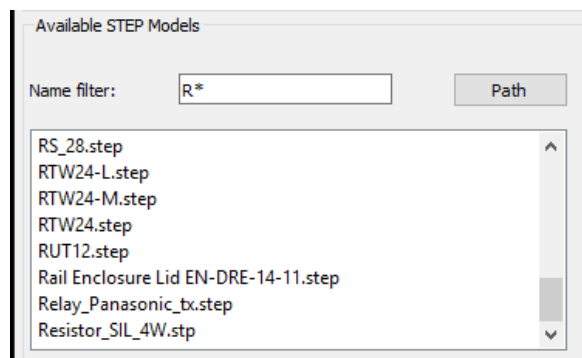

In the filtered list results, select the required STEP model. The mapping tool graphics will display with the selected STEP model place in the design at the board drawing XY Origin, TOP layer.
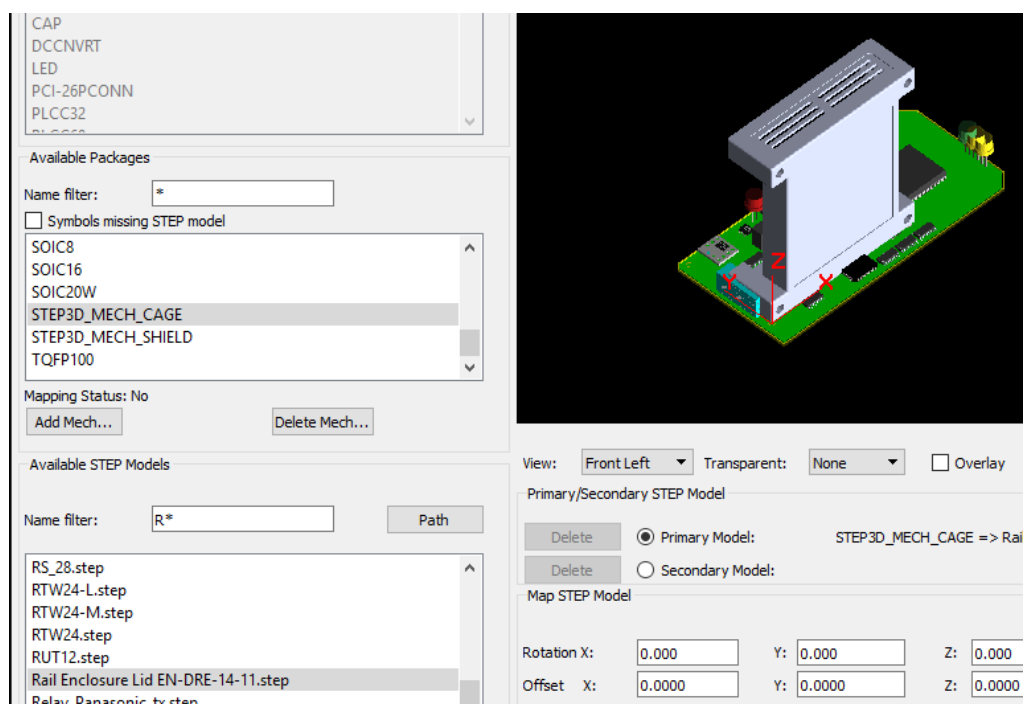
Using the Map STEP Model fields as mentioned earlier, position the mechanical model on the PCB. In this case the required values are: Rotation X: 270.0, Offset X 10.00 Y: -26.8   Z: 20.00.
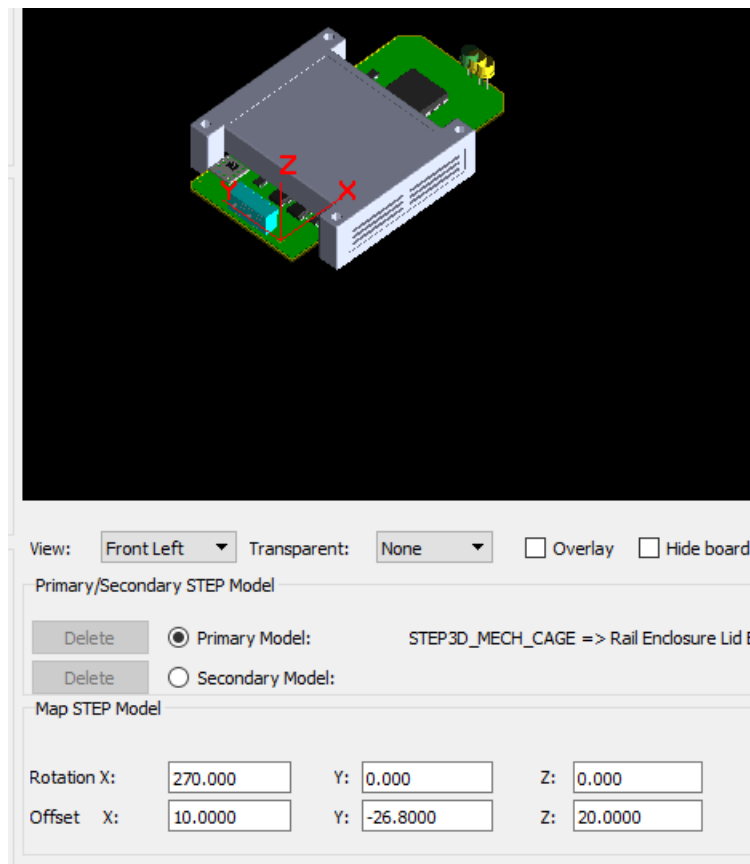
Once satisfied with the position of the mechanical model, select the Save button to preserve the offset. The mapping tool may now be closed. A board symbol with the name STEP3D_MECH_CAGE is placed on the drawing origin.
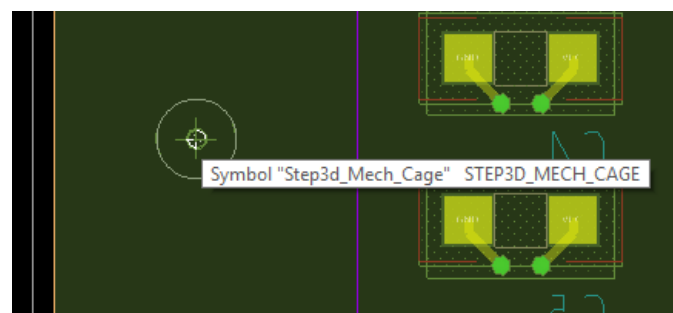

Launch the 3D Canvas, The mechanical model is now mapped into the 3D Canvas.

Exporting a Board Drawing to a STEP Model.

STEP models are used in various ways in the mechanical design environment such as checking for form and fit. PCB Editor STEP model support provides the ability to export a PCB Editor 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 options to minimize or maximize STEP model data. **Potential for very large STEP model files exists when exporting STEP Model Parts and external copper data.**

Open the board file that you wish to create a STEP model from, then From the PCB Editor Menu, select **File > Export > Step (Allegro) or Export > MCAD > Step (OrCAD)**.
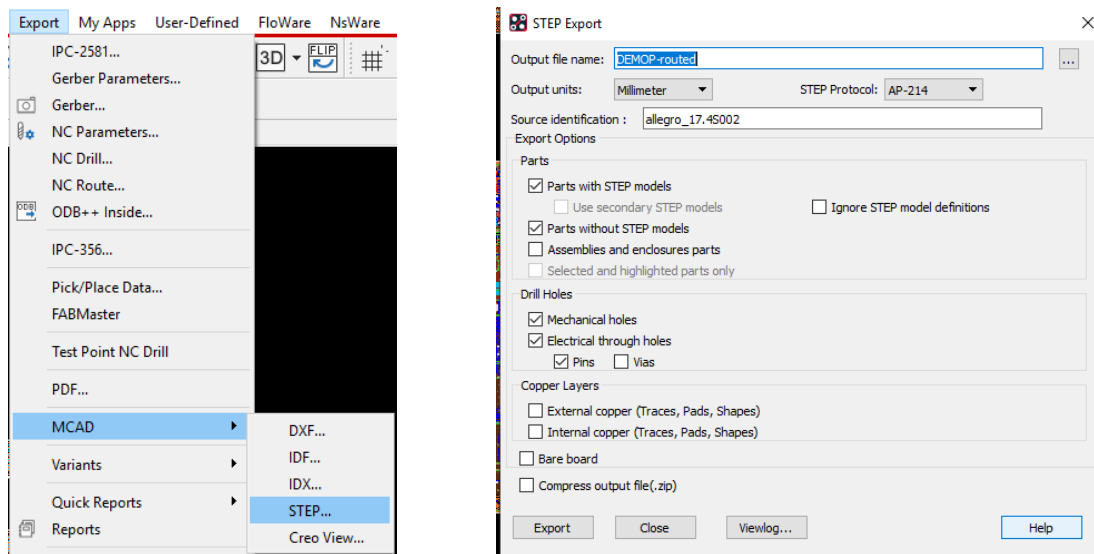
Output file name - Specifies the name for the output file. The default value is the base name of the active design. To search for existing files, click... to display the file browser.

Output units - Specifies the unit for the STEP model export. The default value is the unit of the active design. Choose Millimeter, Micron, and Inch.

STEP Protocol - Specifies output protocol formats. Choose, AP-203, AP-214, and AP-242. The default is AP-214.

Source identification - Specifies the tool of origin within the STEP model data. The default is the current release of PCB Editor.

Parts with STEP models - Includes the STEP models mapped in the current design.

Use secondary STEP models - Includes secondary STEP models.

Ignore STEP model definitions - Ignores STEP model definitions for mapped symbols.

Parts without STEP models - Includes symbols without STEP models mapping. The unmapped symbols are exported as defined by the PACKAGE GEOMETRY/PLACE_BOUND_TOP/BOTTOM.

Assemblies and enclosure parts - Includes the STEP3D_MECH models created in the STEP Package Mapping tool.

Selected and highlighted parts only - Includes only selected and highlighted parts.

Mechanical holes - Includes mechanical holes defined in the current board drawing.

Electrical through holes - Includes electrical through holes defined in the current board drawing.

Pins - Includes electrical through pin holes defined in the current board drawing.

Vias - Includes electrical through via holes defined in the current board drawing

External copper (Traces, Pads, Shapes) - Includes external traces, pads, and shapes on the external (ETCH/TOP and ETCH/BOTTOM) layers.
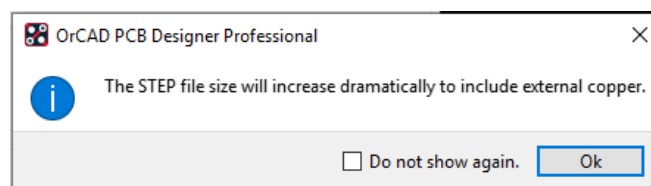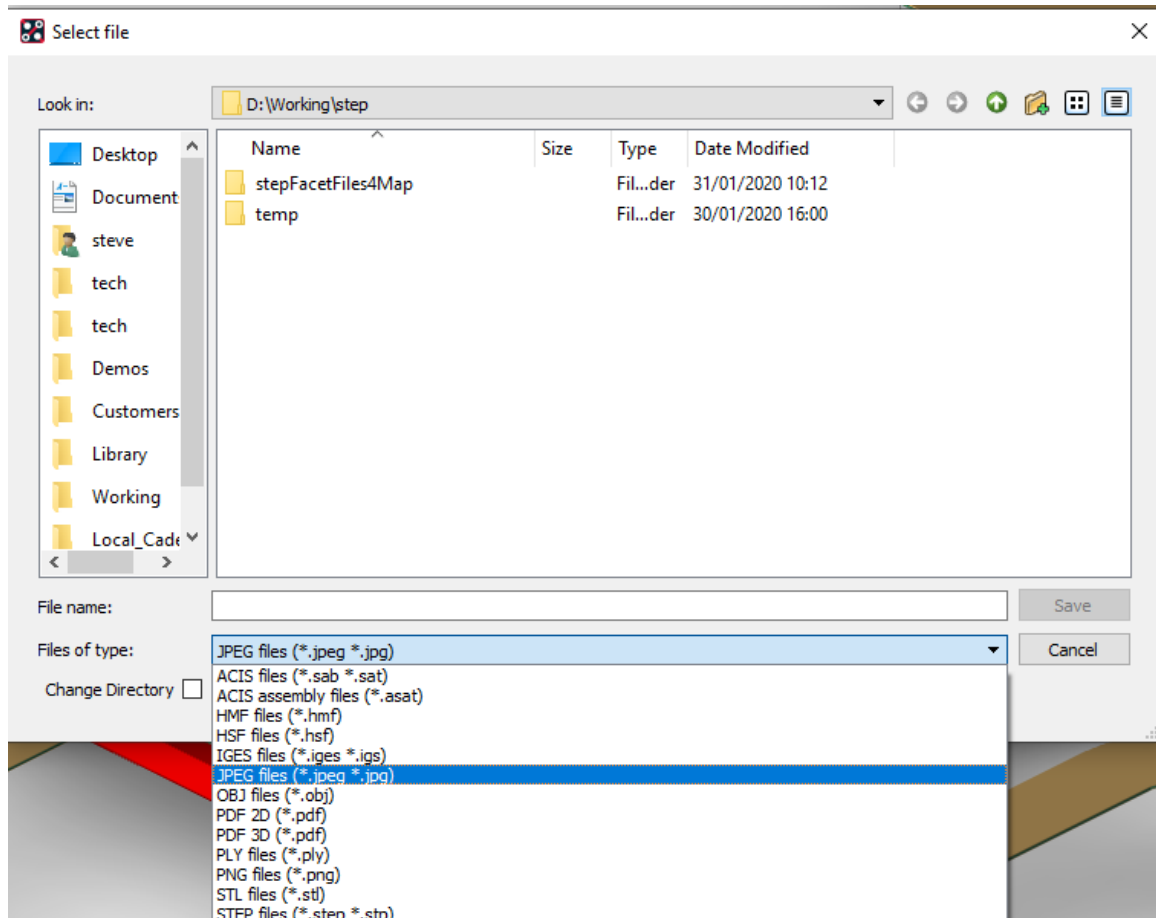

Internal copper (Traces, Pads, Shapes) - Includes internal traces, pads, and shapes on the internal layers. Exports 3D copper that includes internal traces, pads, and shapes. By default, copper is exported as 2D geometries to keep the export data file small in size. Set the user preference environment variable step_3d_copper for 3D copper export with bigger file size.

Bare board - Includes physical board.
Compress output file (.zip) - Compressed the exported STEP file into a .zip file.

Set the required options for the STEP file you wish to create and click on **Export.** A STEP file is saved to the current working directory. If a STEP model viewer is available, this file may be loaded into the viewer.

There are also many options available directly from within the 3d Canvas. Look at **File > Export.** The STEP file produced from the 3d Canvas will be more detailed since it includes all the available layers and models that are visible in the 3d Canvas.