STEP Model Support in PCB Editor

Overview

All versions of PCB Editor (OrCAD X and Allegro X) allow users to see their board design in a 3DX 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 3DX Canvas**
- **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

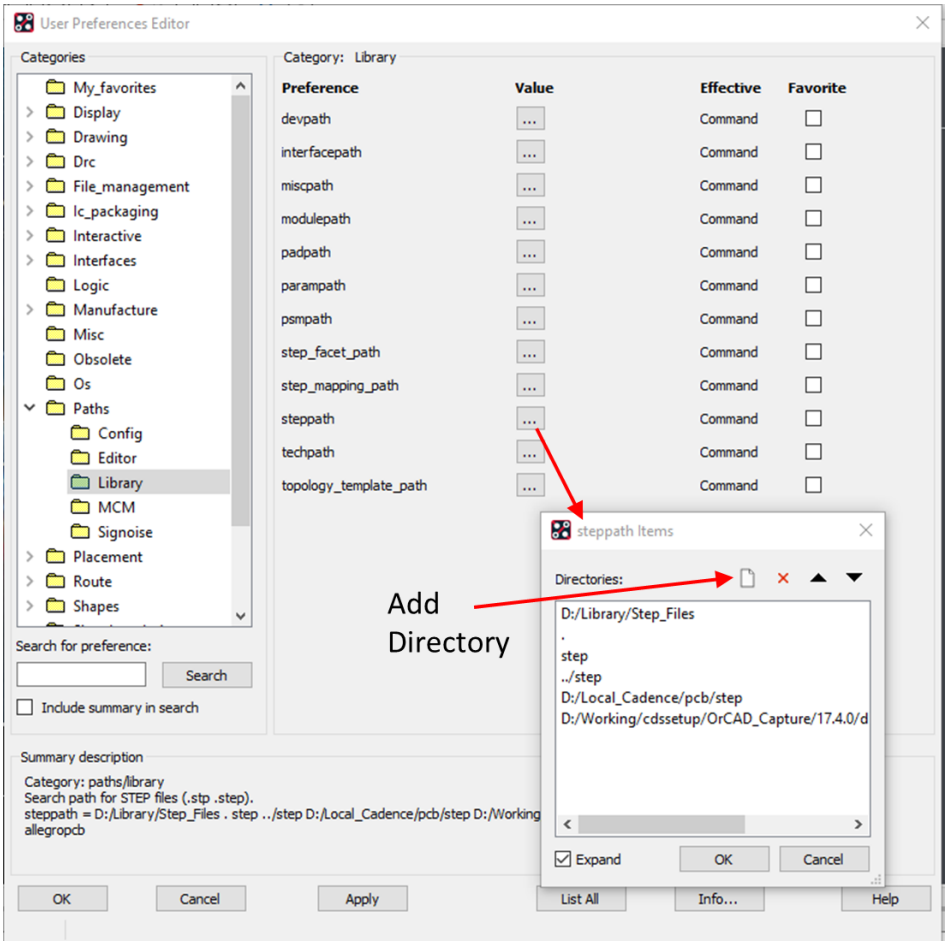
http://www.prostep.org/en/projects/step-ap-242.html

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.



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 3DX 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.

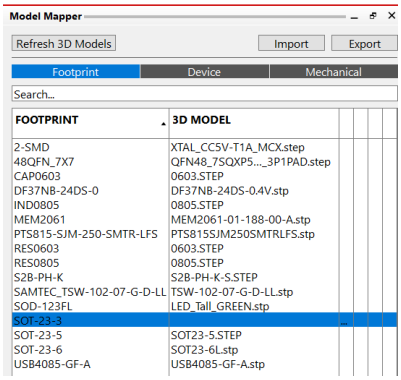
There are two supported 3d canvas methods at the time of writing this application note. You can either invoke the original canvas from Display > 3D Canvas or this  icon. The mapping method although graphically different in principle is the same so won't be covered in this note. The preferred method would be to use Display > 3DX Canvas or this icon. 

The example below goes through mapping directly in a board file.

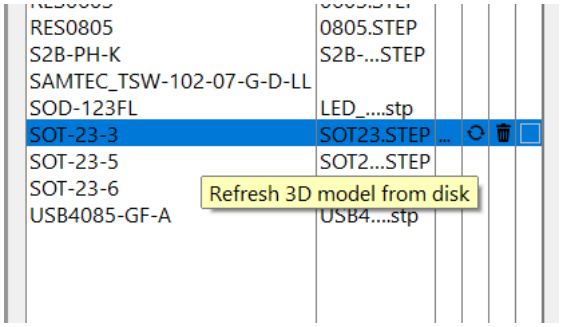
Open PCB Editor then go to Display > 3DX Canvas, the 3DX Canvas opens showing the board.

To change the view, roll the mouse wheel to Zoom, control + middle mouse click to Pan and shift + middle mouse click to Rotate.

The next step is to look for the Model Mapper. This is usually shown as a tab on the right-hand side of the screen. If it is not, then choose View > Model Mapper.

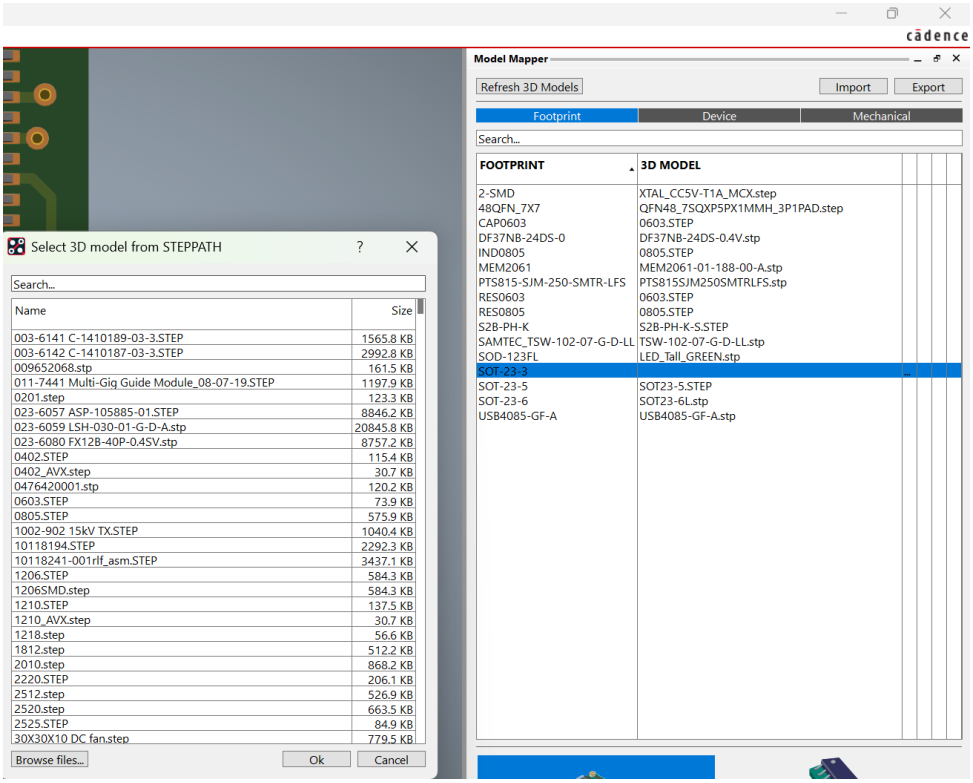


The four columns in the model mapper are explained as follows:-



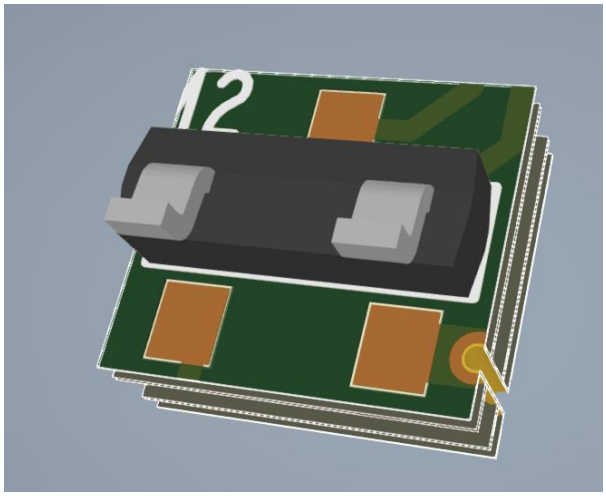
1. Browse for step model.
2. Refresh step model from step path.
3. Delete current step model.
4. Override model colour – In some instances models will be a default colour, this option allows you to choose a different colour.

To map a 3D STEP model in a board file, select the part in the model mapper – in this example SIOT23-3 is chosen. Select the … icon in the first column to browse for an appropriate step model. The list shows items in the steppath locations. Find the step model required, select then click OK. In this example a SOT23-3.stp is chosen.

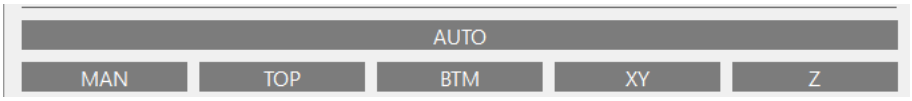


Once selected the model is shown in the main canvas, it will be easier to align by choosing Device Only (to only show one part).

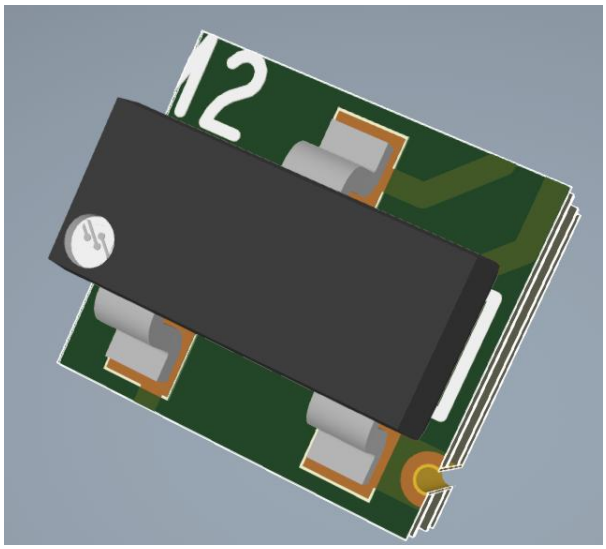
The part may or may not be aligned correctly with the footprint based on how the step model coordinate system was defined. In this example it is not:-



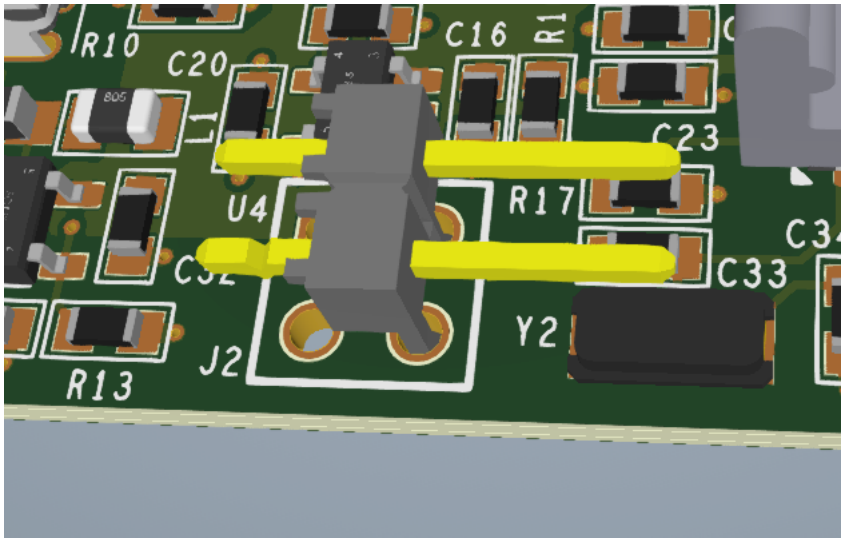
There are a few methods to map align this correctly, the first step should always be to use the Auto button.



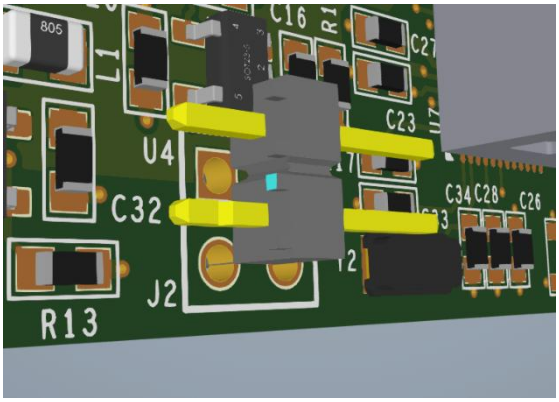
In general many parts will be aligned correctly using this selection, for this example the part is aligned correctly.

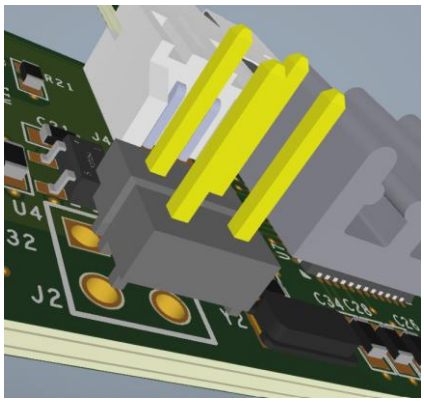


This next example will use a different method, this time XY which will allow us to align chosen pins on a through hole device to holes in a PCB. I have a SAMTEC 4 pin connector that needs to align with holes in the PCB:-

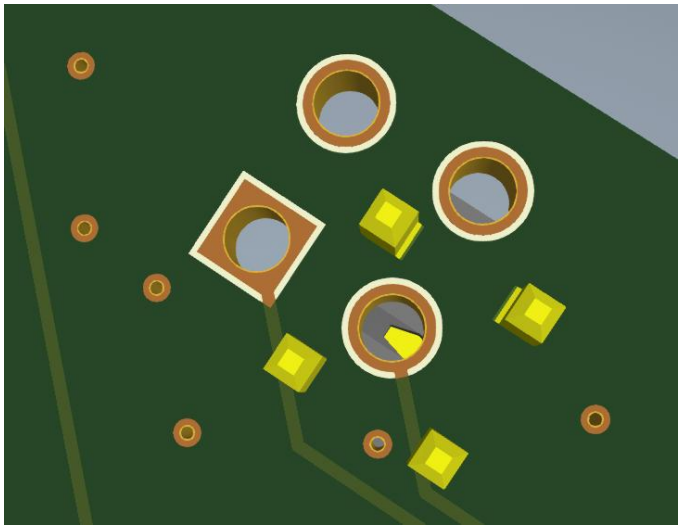


When the model is mapped the orientation is not correct as shown above. For this example we will use TOP initally to get the connector bottom align#ed with the top face of the PCB. Choose TOP then using the SHIFT + Middle mouse button, spin the model around until the correct face of the connector you want to align with the top face of the PCB is selectable, hover your mouse over the different faces (they will highlight blue), once you are happy, left click the face and the connector face selected is aligned with the top face of the PCB.

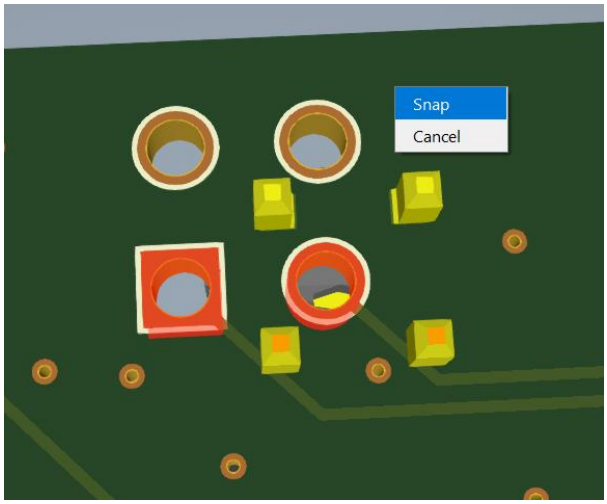




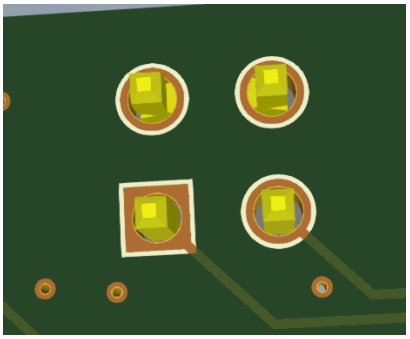
Next spin the board around so you can see the pins and holes:-



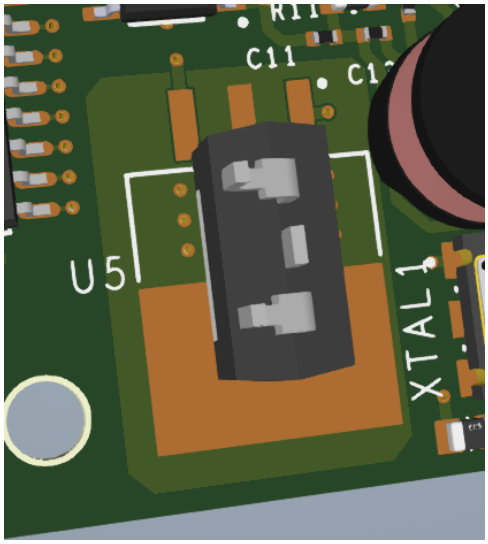
Now choose XY then left-click two pins the connector then choose two holes (that match the order of the pins), then right-click - Snap

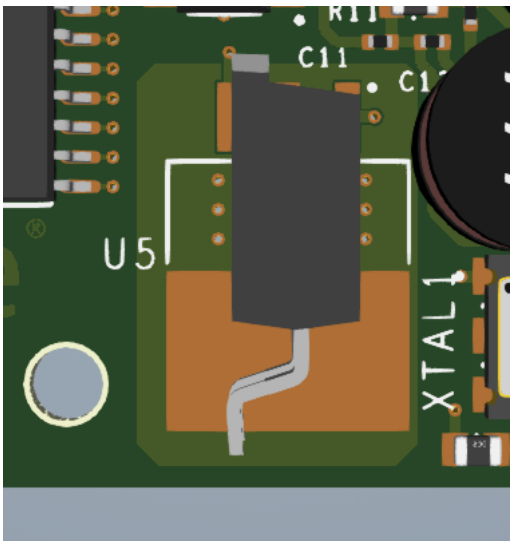


The connector is now aligned with the holes.

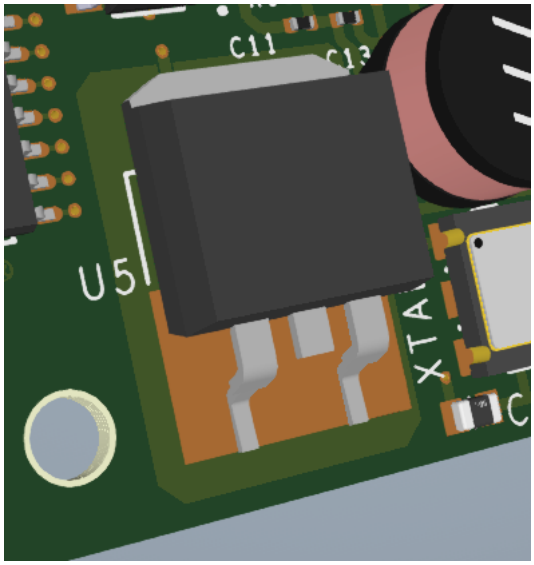


The next option for positioning a step model is to use the MAN (manual) option. Select the required step model and the default position isn't correct. Using AUTO doesn't help too much either.

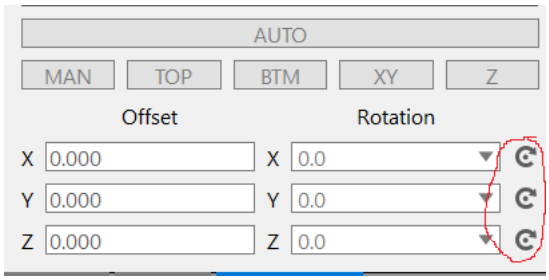




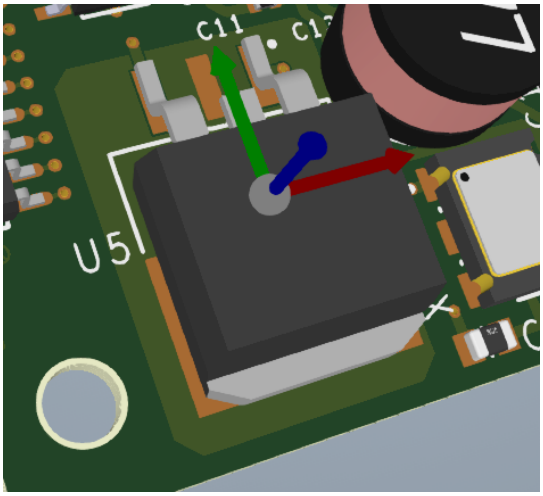
I'll start by using TOP to align the back face of the transistor with the top face of the PCB as we did earlier (so invoke TOP then left-click the back face of the part. This is looking better but the orientation of the part isn't correct.



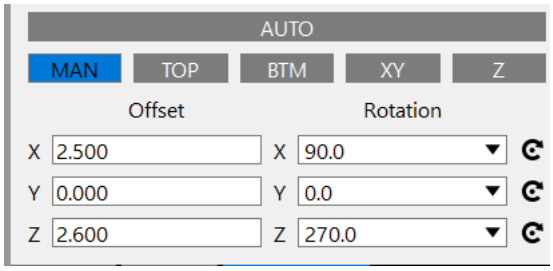
Using the rotation buttons as shown select each one to rotate the step model into the correct orientation.



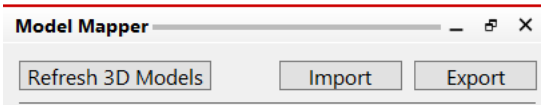
Once the transistor has been rotated correctly use the MAN button to adjust the exact position.



Drag the coloured arrows to position the transistor correctly on the pad. You can also manually type values in the X, Y and Z boxes to fine tune the exact position which are as follows:-



You can export all the mapping information into an xml-based file. Use the Export button at the top of the Model Mapper window to export this data out. You can also import this mapping using the import button.



Back in the main canvas window (2d) you can invoke 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 3d Mapper. You can see all X, Y and Z rotation and offsets are stored as part of the package symbol.

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 recommended method.

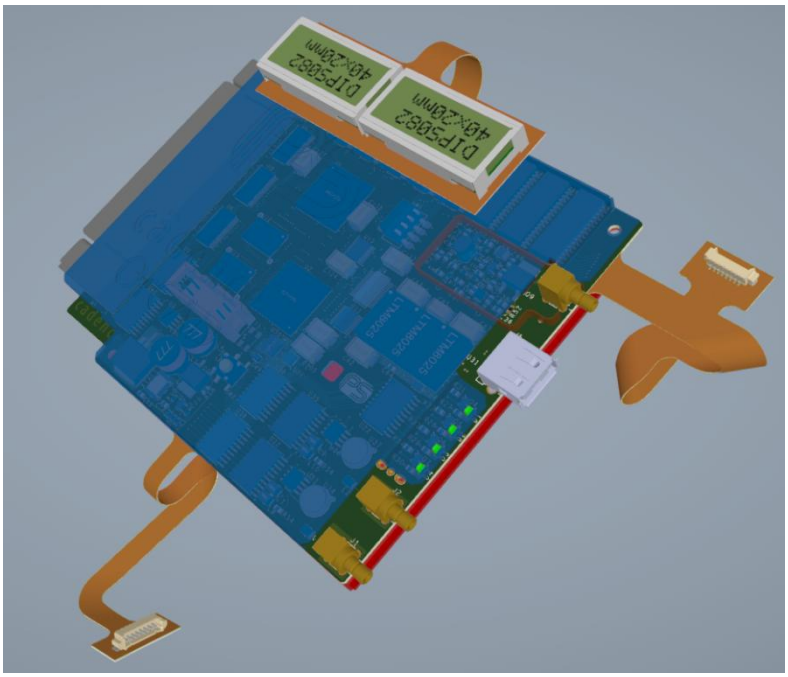
3DX Canvas with STEP models

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

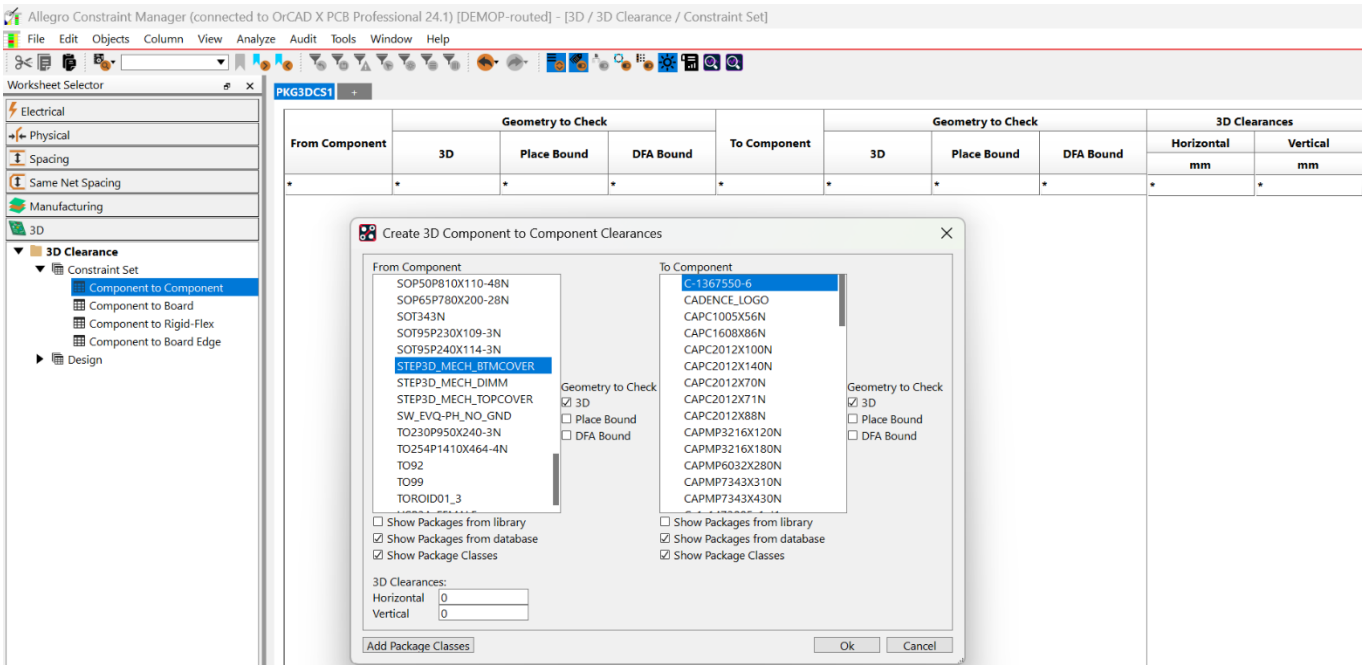
Select the 3DX viewer icon in the PCB Editor toolbar 

, or from the menu select Display > 3DX Canvas.

When the 3DX 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.

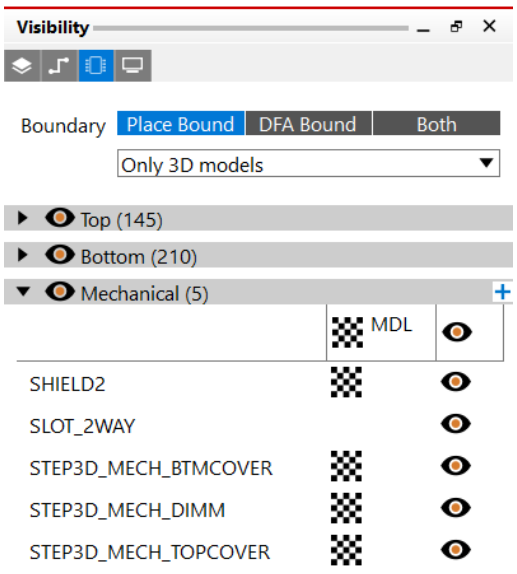


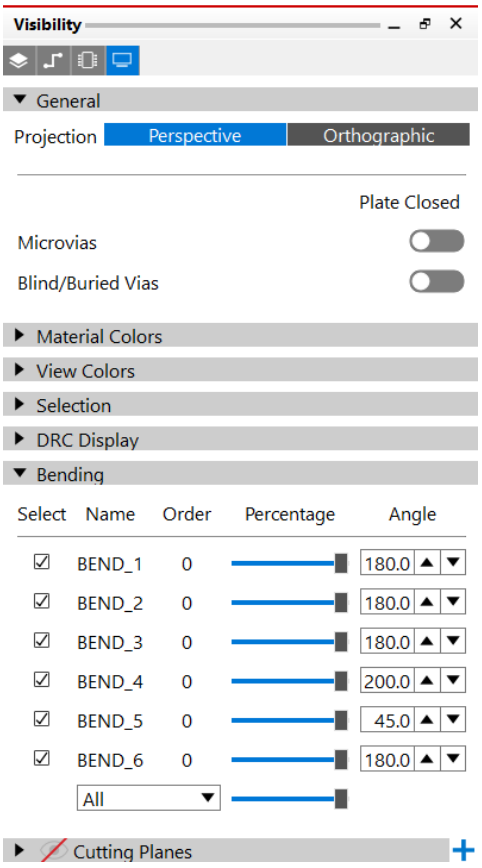
This design has mechanical models (top and bottom covers and a dimm module PCB that is mated into the connector on the back side of the PCB). The Cadence Constraint Manager has a 3D constraint section where rules

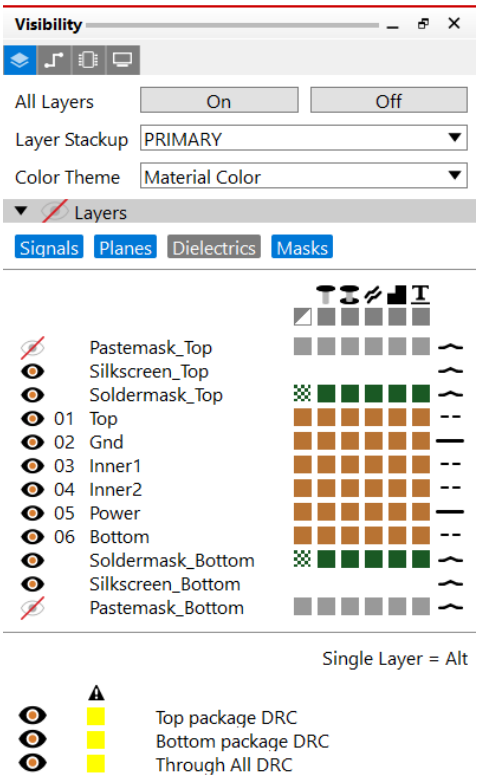


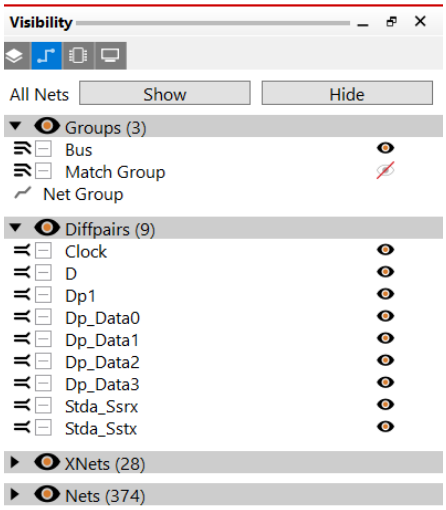
for the 3D objects can be defined. You can create classes for the different packages or pick a specific component to check to, like the cover for example. For further details on rule creation refer to the Constraint Manager User Guide.

The Visibility pane controls what is displayed and allows all objects to have their colours defined. This includes 3d bending for flexis.







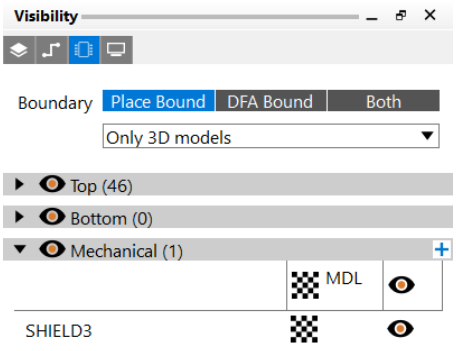


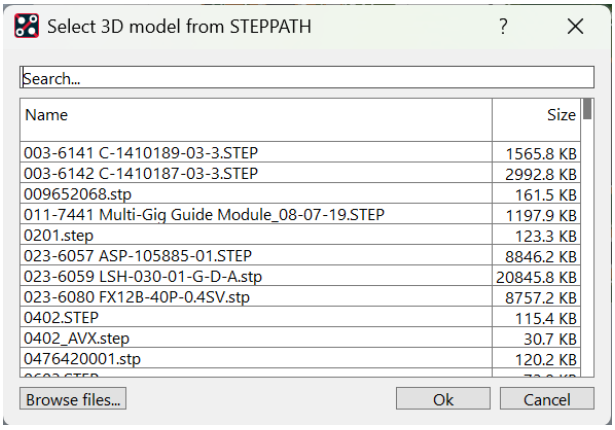
Mapping Mechanical STEP Models

Viewing STEP models in the PCB Editor 3DX Canvas provides a more accurate representation of on-board symbols. The user can view component placement and visually determine (or generate a Constraint rule) 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 3DX Canvas will display these models where the user can then run a DRC check to look for any DRC errors much the same way as a physical or spacing rule would work.

For mechanical parts that are placed on a board (using a mechanical symbol) are shown in the Visibility pane component tabs by default. You can add the step model to the actual mechanical symbol the same way you would to a package symbol. This is also where you would add parts manually for example a cover or a heatsink that doesn't have a mechanical symbol.

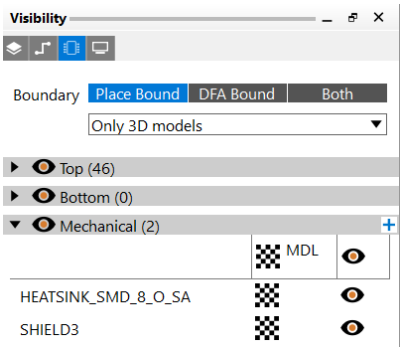
Launch the 3DX Canvas then in the Visibility tab click on the Components tab. You will see the Mechanical dropdown listing the mechanical parts. To add a new one, click on the + icon.

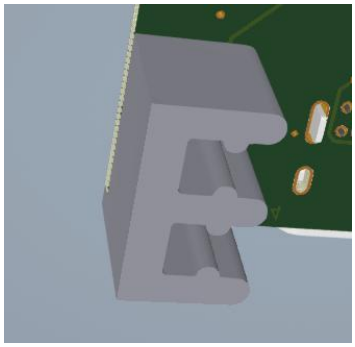




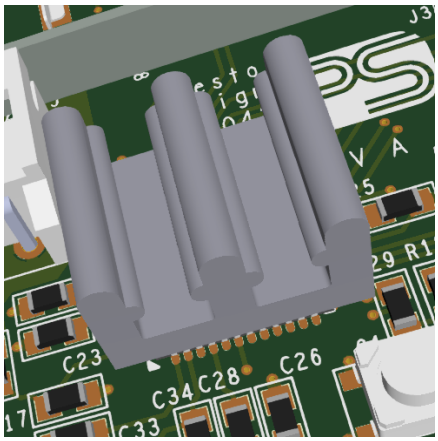
You can now browse the steppath for the required mechanical symbol, you also have the option to browse a different directory.

Once you click OK the step model is added to the mechanical list and is also placed at the design origin location.





Using the method you used to position package symbols, you would the adjust the position of the step model in the model mapper.

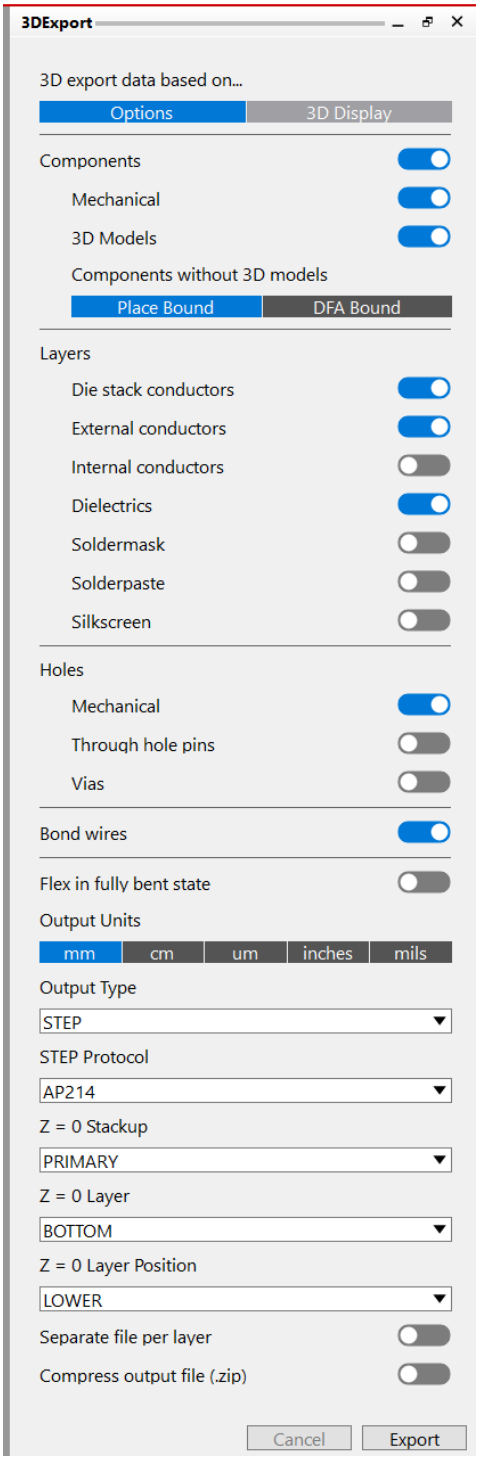


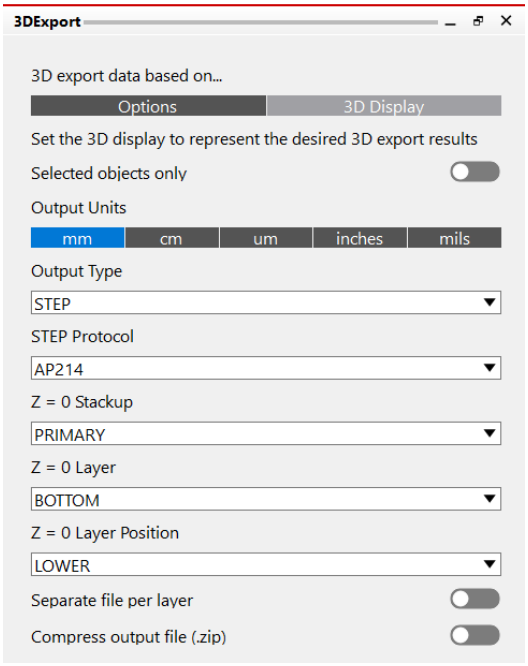
To delete a mechanical part right-click the mechanical part name in the visibility pane and choose Delete.

Exporting a Board to a STEP Model.

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.**

From the 3DX Canvas look for the 3DExport tab. If this is not shown use View > 3DExport to display the tab.





Choose the required settings and export the 3d Step Model. You can also invoke this command from with PCB Editor File > Export – 3D (Allegro) and Export > MCAD > 3D (OrCAD). This shows the same menu options as above.