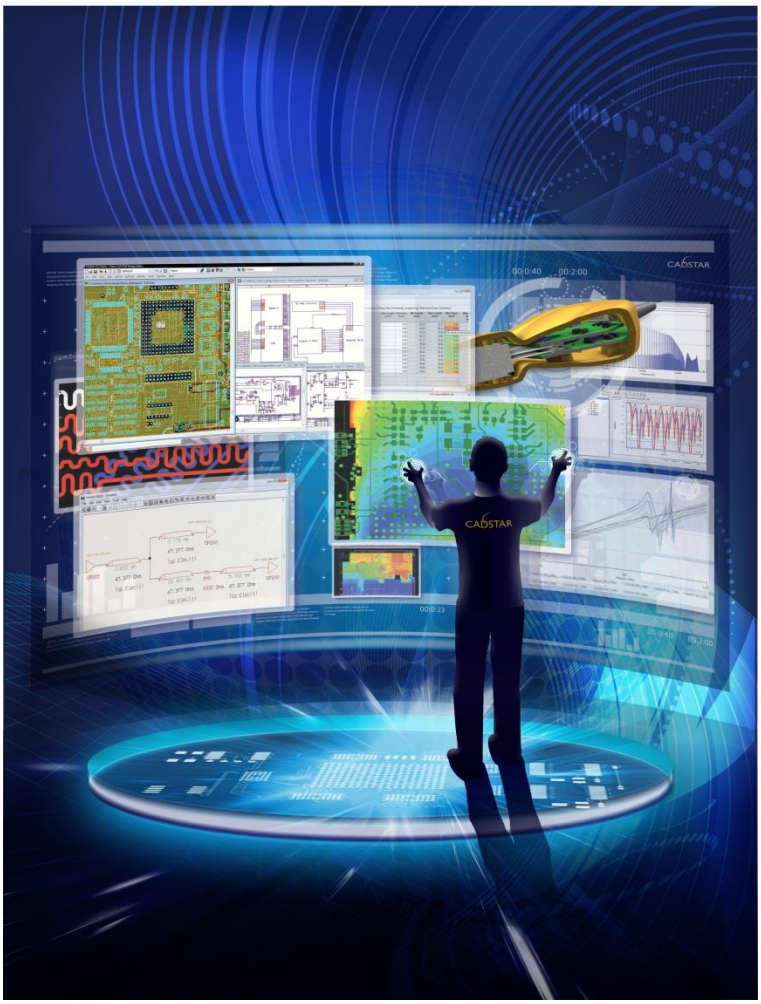
CADSTAR

BoardModeler Lite – Version 7.1



Self-Teach Training Guide

ZUKEN

Table of Contents

Document version: 7.1A Last updated: 26nd April, 2013

Introduction

Overview

This tutorial describes the basic operation and typical use of BoardModeler Lite. It is designed for first-time users to become familiar with the common commands and to gain experience of using BoardModeler Lite in a typical design flow with CADSTAR.

Further Information

For further detailed information on the topics covered in this tutorial, please refer to the on-line help provided within the application. This can be accessed via the ***Help / Help BoardModeler...*** menu option or by pressing the ***F1*** function key.

Conventions and Terminology Used In This Document

The following conventions and terminology are used throughout the tutorial document

- Commands, menu options or user interface operations are shown using bold italic text, for example, ***Tools / Options***
- Names of file system objects (file paths, file names, folder names etc.) are shown in italic text surrounded by single quotes, for example, '*C:\BML Training\BM_Project*'
- Any values or settings that must be entered into the BoardModeler Lite user interface by typing text are shown enclosed in double quotes, for example, "0.5". Note that the quotation marks should **not** be entered as part of the text input.
- Throughout this tutorial, the 'BoardModeler Lite' application name is usually abbreviated to 'BML'.

- This Symbol indicates an HTML link to a demonstration video. This icon will appear frequently throughout the tutorial. An internet connection is required to see the videos. They are best viewed with Internet Explorer (c) 9.0 or newer and Firefox (c) 3.0 or newer.

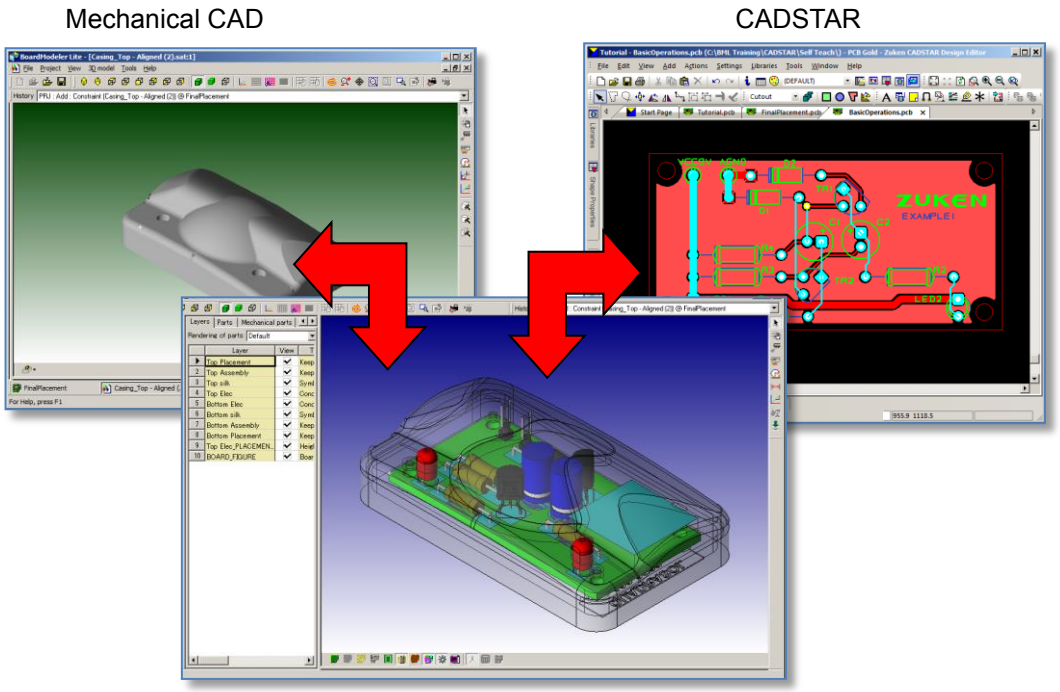
Software Tools Used By This Tutorial

This self-teach tutorial guide requires the use of the following tools:

- BoardModeler Lite 7.1
- CADSTAR 12.1 or later

What is BoardModeler Lite?

BoardModeler Lite (BML) provides a powerful design environment to bridge the gap between electronic (ECAD) and mechanical (MCAD) design. BML is a tool for use by the PCB designer, to combine the two-dimensional (2D) board design data and the three-dimensional (3D) mechanical design data in a single environment. It also allows for easy exchange of data between the electronic and mechanical domains.



BoardModeler Lite addresses the following main functional areas:

- 3D Representations for Electrical and Mechanical Parts

  BML provides the functionality to import the 2D CADSTAR PCB design data and 3D mechanical design data and to combine it in a single environment. It is able to display realistic 3D models to represent all aspects of the design, such as the board outline, PCB layers, electronic parts mounted on the board and a mechanical case or chassis. This is all presented in a single window and allows the user to view the data in 3D, from any required viewpoint.

- Clearance and Collision Checking

  BML provides the features to check a design (either interactively or in batch check mode) for collisions between components mounted on the board and between the components and board shape and any enclosing mechanical constraint (e.g. a case or chassis). It is also possible to measure sizes of design objects and measure the clearances between them.

- Back-annotation of Design Changes from 3D to 2D

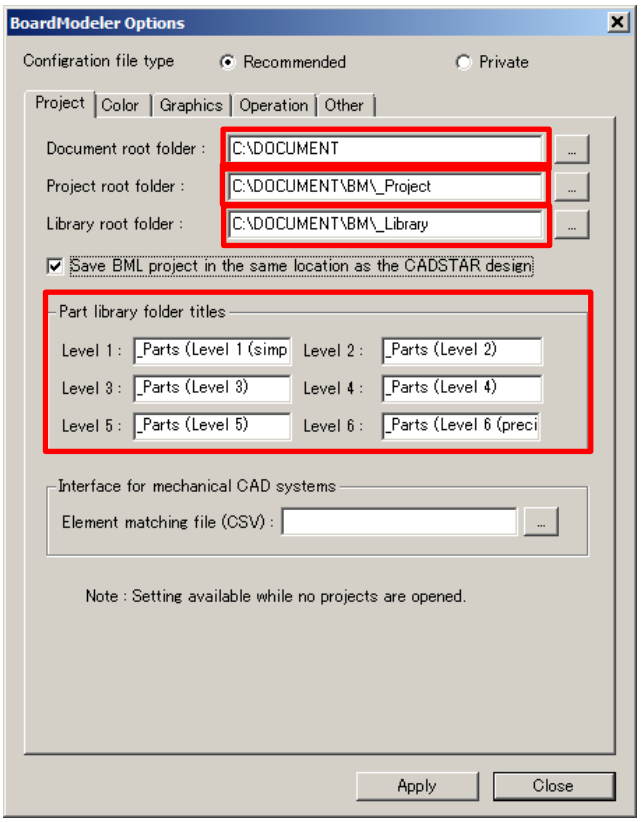
  BML allows changes made to the component placement in the 3D environment to be passed back to the 2D CADSTAR PCB design. This also covers the ability to pass initial board outline shape, critical component placement and keep-out or height limitation areas, supplied by the MCAD department, into the 2D CADSTAR PCB design.

All of these areas will be covered in this tutorial.

Data File Structure for BoardModeler Lite

The design data used by BoardModeler Lite is stored in a defined folder structure in the file system on disk. The location and naming of these folders is set up in BML on the ***Project*** tab of the ***Tools→Options*** dialog (as shown in the image below).

The location of the ***Document root folder*** is set during the software installation process (the default location is *'C:\DOCUMENT'*) but this can be changed later through the ***Tools→Options*** dialog. ***However, it is important to note that these locations can only be changed when no Project data is loaded into BML.*** Also note that if you do not have 'Administrator' privileges you will need to first select the ***Private*** option on the ***Tools → Options*** dialog in order to be able to modify the folder paths.



Document Root Folder

This is the main folder that contains all design data relating to BML. When this location is defined the following four sub-folders will be created automatically inside a folder called 'BM' in the Document Root Folder.

_Project
_Library
_Interface
_Resource

Project Root Folder

This folder is used to store data for individual projects created in BML. The default location for this folder is *'<Document Root Folder>\BM_Project'*, although it can be changed to a different location of your choice.

Library Root Folder

This is the folder that is used to store the 3D model data to be used to represent how the components (parts) will be viewed in BML. The default location for this folder is *'<Document Root Folder>\BM_Library'*, although it can be changed to a different location of your choice.

Underneath this folder are six separate folders (*'_Parts_1'* to *'_Parts_6'*), which allow 3D models of up to six different levels of detail to be used to represent each component. The required level of detail to use for display can be chosen from within BML. This will be covered further later in the tutorial.

Save BML project in the same location as the CADSTAR Design

If the option is enabled, then BML will attempt to load and save data into the same location as the corresponding CADSTAR PCB design file. BML will create a folder of the same name as the design and will write its design file <design_name>.bma alongside the CADSTAR PCB file. In this case, any setting for Project root folder is ignored.

The behaviour is different depending on whether BML is started from within CADSTAR (from the Tools menu) or from the Windows Start menu. Even if the new option has been selected, if BML is started from the Windows Start menu, it will still be necessary to browse to the required location from which to load existing data or save a new project. If the option is not set, then BML will behave as before.

If you have existing BML project data which you wish to re-locate along-side the corresponding CADSTAR data, then it will be necessary to run BML from the Start menu, load the existing project data and then save it in the new location (using File / Save as / Project…) before the new option is enabled – otherwise BML will not be able to find the existing data and will create a new project from scratch.

Task 1 - Configuration Set-up for the Tutorial

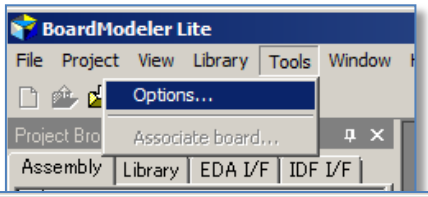
This tutorial expects the design data to be located in a folder called '*C:\BML Training'* and so we will now go through the steps necessary to configure BML to access this data. To begin, ensure that you have extracted the tutorial example data from the zip file ('*BML Training.zip'*) supplied with the tutorial into '*C:\BML Training'*.

Note: If you choose to put the example data in a different location, you will need to substitute all the file paths in the following tutorial to match the actual location of the data.

Now perform the following steps.

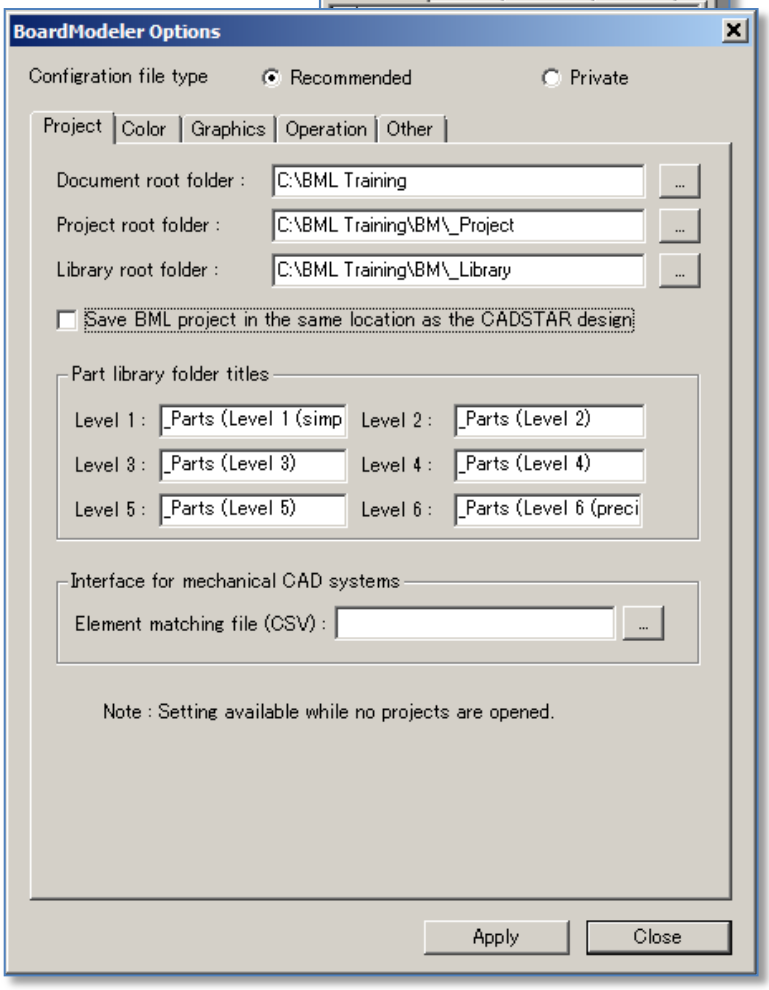
1. Select ***All Programs→BoardModeler Lite→BoardModeler*** from the Windows ***Start*** menu. BoardModeler Lite will run up.

2. From the main menu bar select ***Tools→Options*** to display the ***BoardModeler Options*** dialog and ensure the Project tab is active.

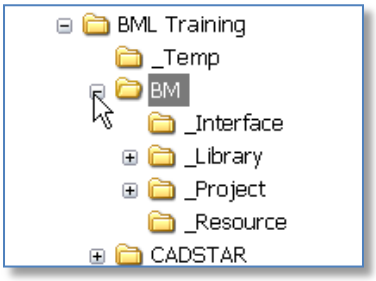


3. **Specify the folder locations as shown in the image.** Note that once you have entered the path for the Document root folder and clicked the [**Apply**], the paths for Project and Library root folders will be set automatically.

   If you do not have 'Administrator' account privileges on the computer, make sure that you select the *Private* option to enable the folder paths to be modified.

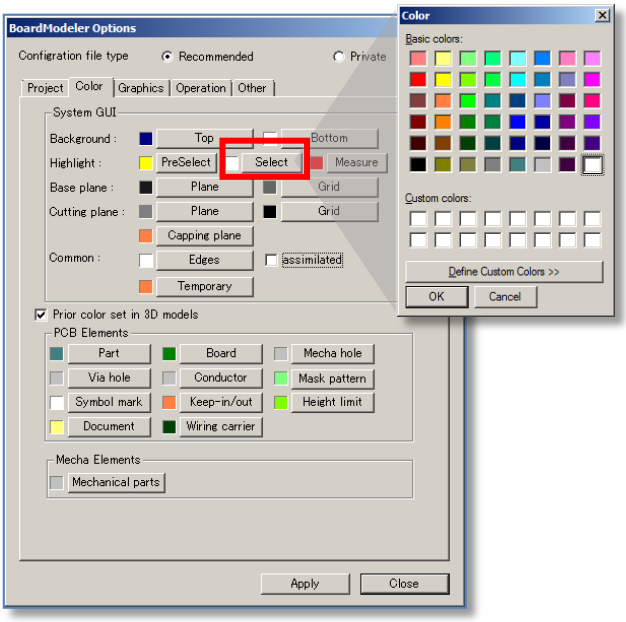


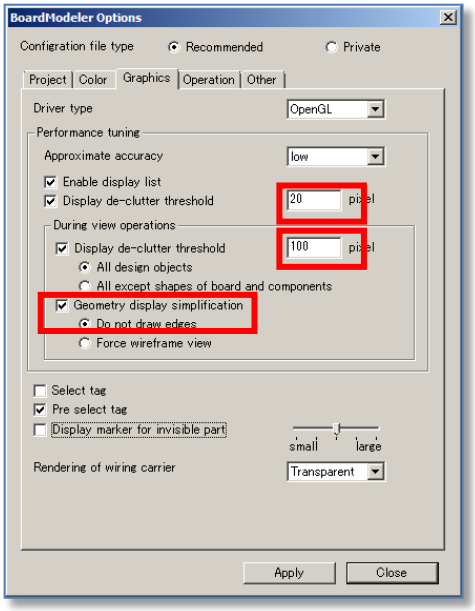
The above operations will direct BoardModeler Lite to access the folder structure as shown in the image below.

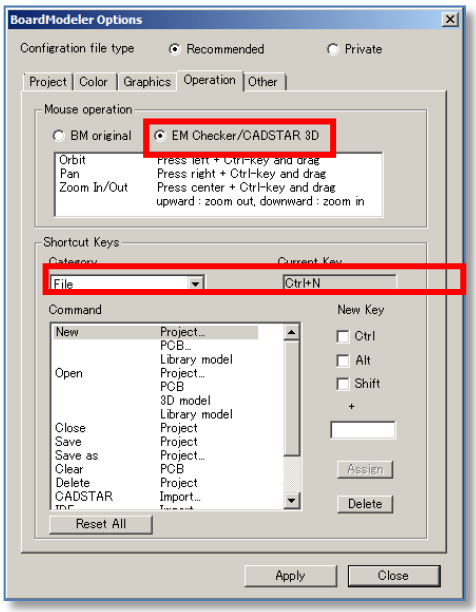


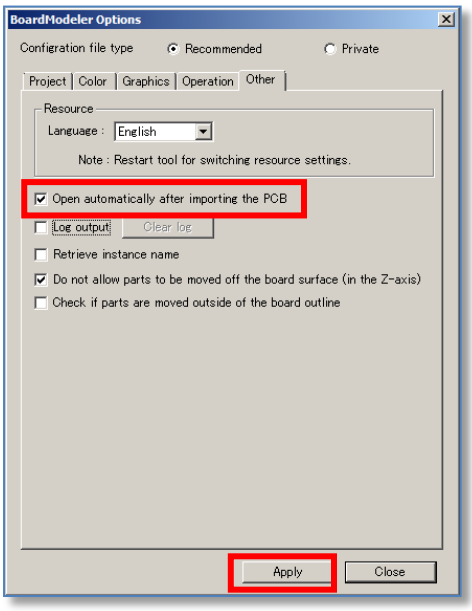
The example CADSTAR data which will be used during the rest of this tutorial is located in the 'CADSTAR' folder.

In order to make sure that you can match the operations described throughout the following tutorial, ensure that the settings on each tab of the Tools→Options dialog are set as shown in the following images. The most important settings are highlighted with a red border.









4. Click the ***Apply*** button followed by ***Close***.

5. Select ***File→Exit*** to close down BoardModeler Lite

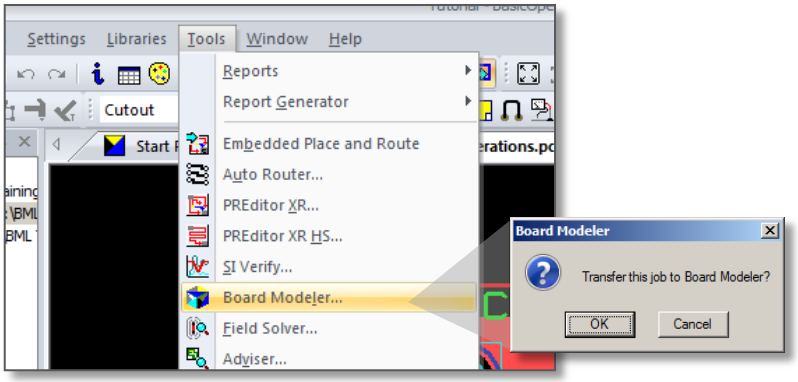
End of task

Task 2 - Basic Operations – Becoming Familiar with the U.I.

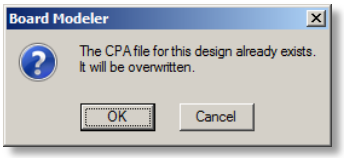
In this section we will take an initial look at BoardModeler Lite and become familiar with the different areas of the user interface and how to perform such common operations as selecting objects in the design or changing the view of the design (i.e. rotate, pan and zoom, setting colours etc.)

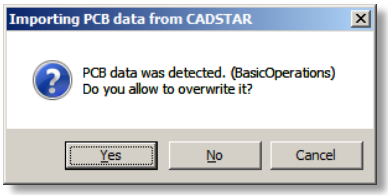
Start by running up CADSTAR and then load the Basic Operations workspace (*'BasicOperations.csw'*) from the *'C:\BML Training\CADSTAR\Self Teach'* folder.

1. From the CADSTAR Tools menu, select the **Board Modeler**... option. Select OK to confirm the dialogs that appear.



Note: If you have already run all or part of the tutorial before, you may see two further dialogs prompting for permission to overwrite the existing data. Select ***OK*** *and then* ***Yes*** in these dialogs.



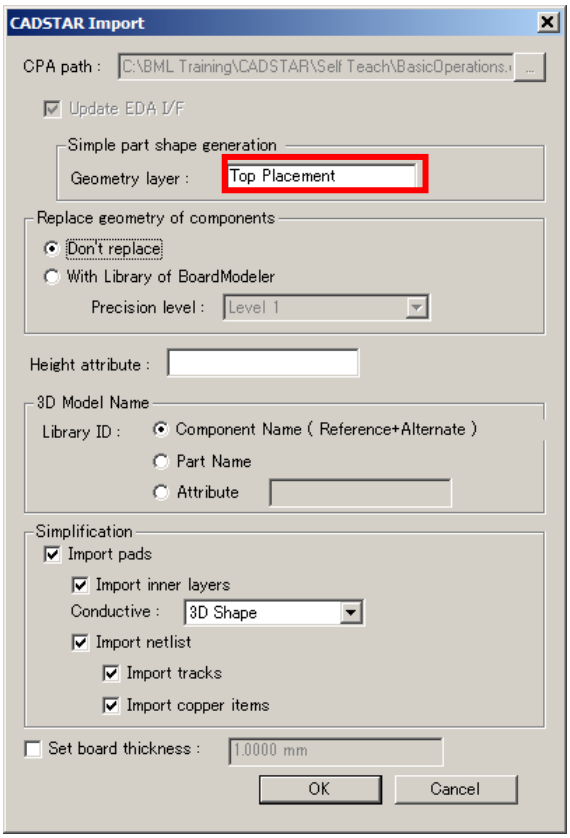


2. Set the options as shown →

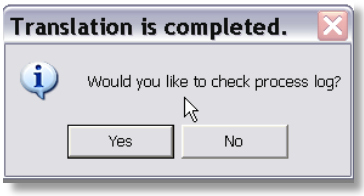
   Enter :**Top Placement**" as the name of the CADSTAR layer that contains the default 2D component geometries that will be displayed in 3D within BML.

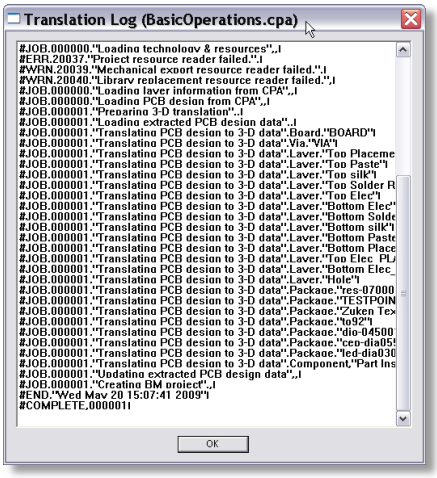
   Alternatively you can enter **Top Silk** or **Top Assembly.**

3. Select [**OK**]



BML will first convert the CADSTAR CPA file into the necessary files for BML to read. It will generate a log of operations as it does this and should there be any Warnings or Errors reported, it will offer the chance to view this log file.





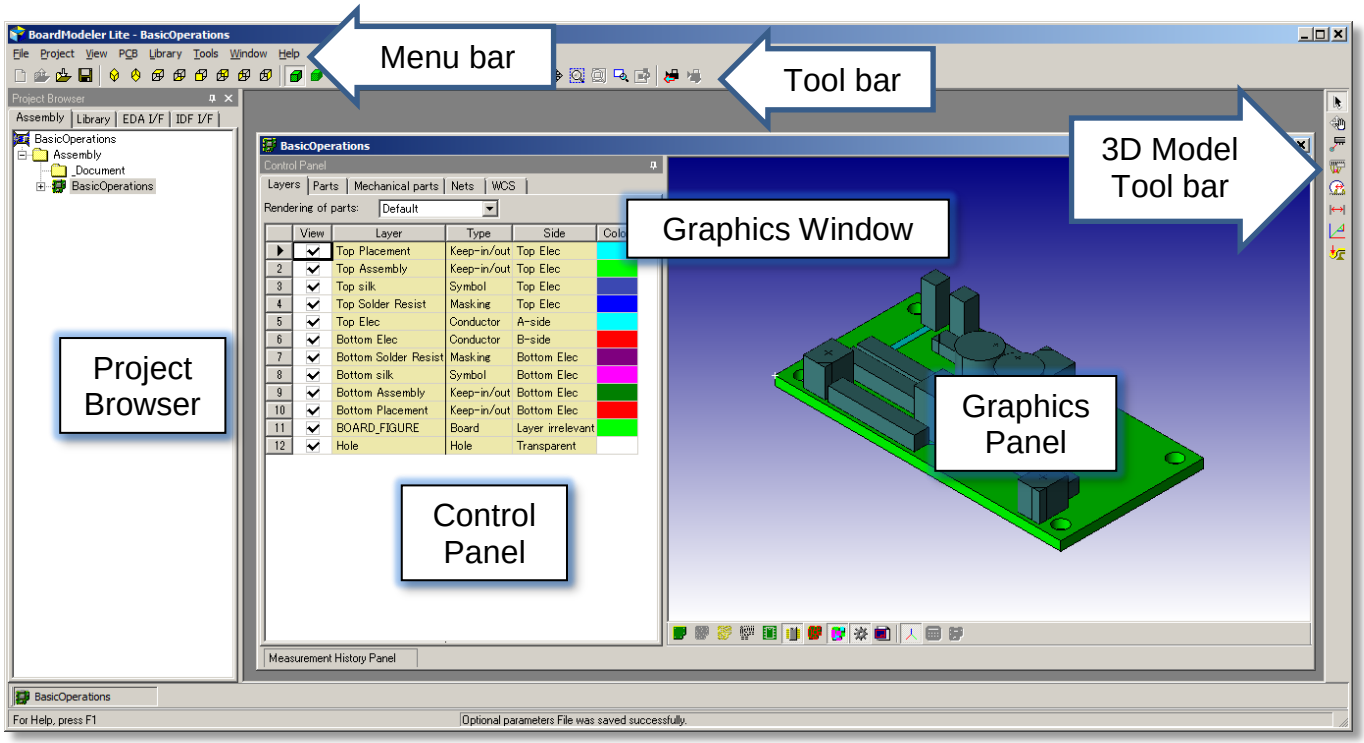
You can select No to continue without viewing the log file. Selecting Yes will show an output similar to that shown here. If you display this report, select OK to dismiss it and continue to load the design into BML.

BML will then continue to load the design and create the necessary graphics data to be displayed by the application.

The PCB design will be automatically opened for display in BML and should look similar to the image on the next page.

The User Interface

The different sections of the BML user interface are shown labelled in the image below and you should spend a few minutes to acquaint yourself with the different areas and their names. They will be referred to throughout the rest of this tutorial.



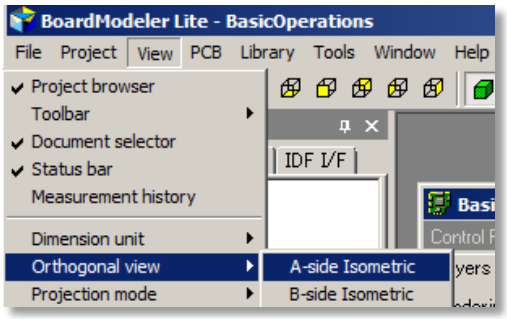
Manipulating the Display

Using the Standard Views

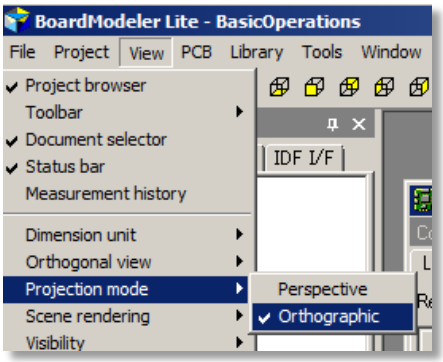
BML provides a number of toolbar icons to quickly switch between different, pre-set views on the design. These cover the standard 2D views (top, bottom, front, back and both sides) in addition to 3D 'Isometric' views of the top and bottom of the board.

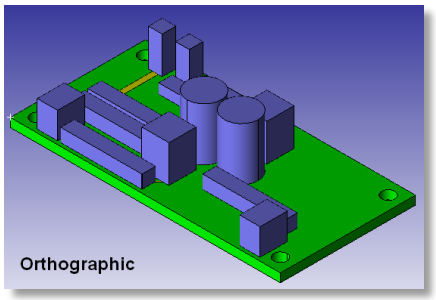
4. Hover your mouse cursor over each icon to see the tooltip which describes its function. Experiment with the different toolbar options to see the effect on the graphics display. Select the top (A-side) Isometric option when you have finished, return to the standard 3D view.

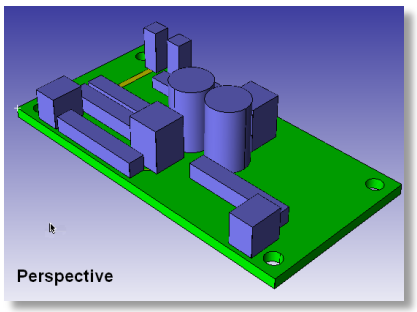
Note: The same functions are also available from the ***View → Orthogonal view*** menu if preferred.



5. If you would rather see the 3D view as a perspective projection (rather than orthographic) you may select this option from the **View→Projection** mode menu.





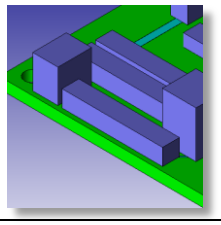
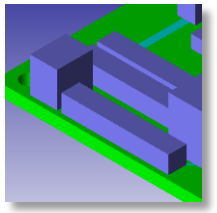
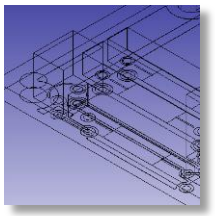


Changing the Scene Rendering Mode

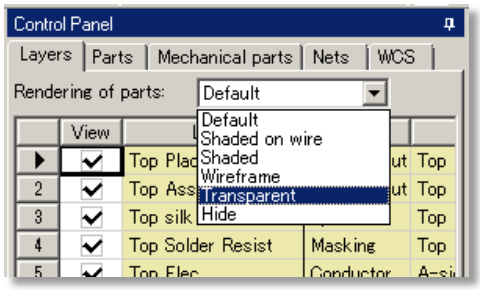
In a similar way, BML provides a set of toolbar icons to quickly switch between different, 'scene rendering' modes. These offer '*Shaded on Wire*', '*Shaded*' and simple '*Wireframe*' views.

6. Experiment with the different modes by selecting the icons from the toolbar or from the View / Scene rendering menu option.

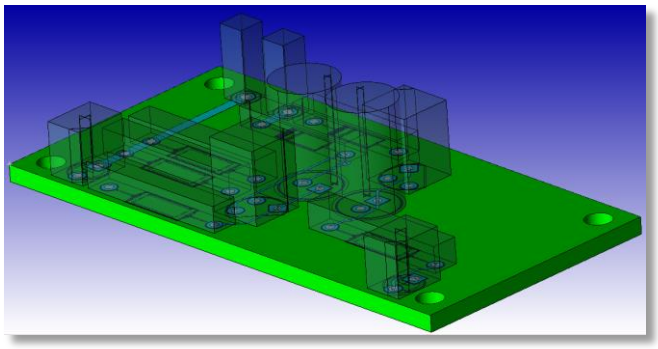
The scene rendering mode chosen will have an impact on how much CPU resource is required to manipulate the display (when rotating, panning or zooming, for example) and it will also affect which features of the design data can be selected. These differences are summarised in the following table.

Icon	Rendering Mode	Performance	Selection
	Shaded on wire 	Slow (especially with large designs)	All
	Shaded 	Medium	Surfaces are selectable, but not edges
	Wireframe 	Highest	No surfaces are selectable – only edges

It is also possible to control the rendering mode individually for each object in the design (e.g. parts, board, mechanical constraint, etc.), and in these cases it is possible to select a 'transparent' rendering mode and a 'hidden' mode (where the selected object is not displayed at all). We will return to this feature later in the tutorial.



You may change the rendering mode for all parts in the design with one operation. At the top of the Control Panel in the Graphics Window, there is a drop-down list box which allows the different rendering modes to be selected. The image to the right shows the effect of selecting the 'Transparent' mode.



7. See the effect of the different modes by selecting the options from the drop-down list in the Control Panel. When you have finished, restore the display by selecting the 'Default' option from the list.

Changing the view – Pan, Zoom and Rotate (Orbit)

BML provides a set of toolbar icons to invoke commands to change the view of the design in the graphics panel. These include standard 2D pan and zoom operations, but also a 3D rotate (or 'orbit') function. The full list of available commands is listed in the table below.

Note: selecting the Rotate, Zoom, Pan or Frame icons will set a 'mode' that will continue to be active until another command is selected. Pick the *Select* icon  from the top of the 3D Model Toolbar or press the 'Esc' key to cancel these commands.

If you use the Mouse Tool short-cuts instead, then these operations (Rotate, Zoom and Pan) can be used at any time (even during the execution of other operations – for example, the ***Move*** command) and the tools are only active while the mouse keys (in combination with the 'Ctrl' key) are pressed.

Icon	Meaning	Mouse Tool
	Rotate (Orbit)	Drag while pressing both 'Ctrl' key and the left mouse button.
	Zoom	Drag while pressing both 'Ctrl' key and the middle mouse button. Drag upwards to zoom out and downwards to zoom in.
	Pan	Drag while pressing both 'Ctrl' key and the right mouse button. Drag in the direction in which you want to pan the design.
	Zoom by area (select the area to be displayed)	
	Zoom Selection (zoom to show selected object)	
	Zoom to Extents (zoom out to display the entire design)	
	Re-center (pans the display so the selected object is at the centre of the screen)	

The ***Zoom Selection*** and ***Re-center*** icons are only available when an object is selected in the graphics panel.

8. Experiment with these icons to see how they operate and get used to manipulating the view of the design. You can use these operations in combination with the rendering mode commands (see above) to explore different views of the data.

Tip: Use the ***View All*** icon to bring the entire design back into view in the graphics panel at any stage if you get lost!

It is possible to cycle backwards and forwards through previous views of the design using the ***Back*** and ***Forward*** icons on the toolbar. 

9. When you are satisfied that you are familiar with these operations, perform a View All command then chose the Select icon from the 3D Model Toolbar.

End of Task.

Selecting Objects

Design items can be selected in BML in several ways. They may be selected individually by clicking on them in the graphics panel, multiple objects may be selected at once by drawing a selection rectangle around them and items can be selected in the Control Panel (on the left of the Graphics Window), which will then be highlighted in the Graphics Panel.

The type(s) of item which will be included in any selection is governed by the current setting of the Selection Filters and these are displayed as a row of icons at the bottom of the Graphics Panel.

The name and meaning of each filter is explained in the following table.

Icon	Filter	Description
	Board	Allows the PCB board itself to be selected
	Hole	Allows any holes (representing vias or component pins) to be selected
	Conductive Pattern	Allows routes/tracks and copper areas to be selected
	Mask Pattern	This option is not currently relevant for BML – please ignore.
	Symbol	Allows the silkscreen pattern from within the component footprint (if any) to be selected
	Component	Allows the components (parts) themselves to be selected
	Restriction Area	Allows restriction areas included within the component footprint to be selected (for example, placement shapes)
	Height Limitation Area	Allows any height limitation areas defined in the design to be selected
	Mechanical Part	Allows any mechanical parts (also known as 'constraints') imported into the design to be selected
	Document	This option is not relevant for BoardModeler Lite – please ignore.
	WCS	Allows any WCS origins (working coordinate system) to be selected
	Chassis	This option is not relevant for BoardModeler Lite – please ignore.

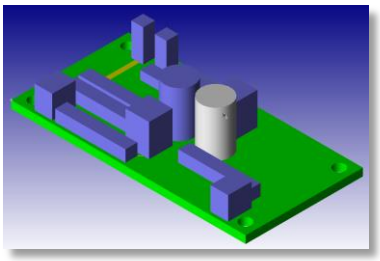
Task 3 – Selecting objects

Try the following examples to get used to the different ways of selecting items within the design.

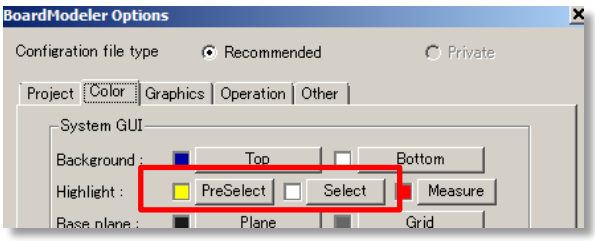
1. Ensure the Board and Component filters are active (click the icons so that they look as though they have been pressed in).

2. Hover a component in the Graphics Panel. The object changes to the ***pre-select*** color, in this case Yellow.

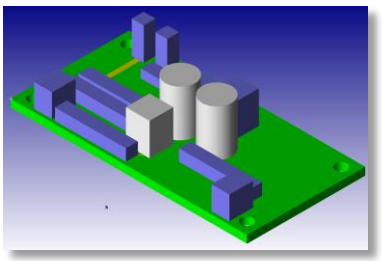
3. Select a component in the graphics panel using the left mouse button. The selected component will be displayed in the '***Select***' color



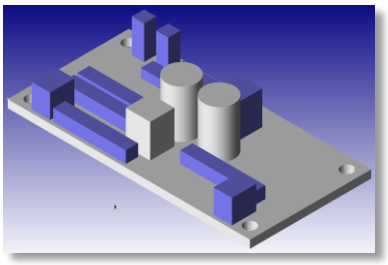
The ***Pre-select*** and ***Select*** colors can be changed in the ***Tools→Options*** [**Color**] tab dialog.



4. Hold down the <Shift> key on the keyboard and click on some different components. Notice that they are added to the selection set and also drawn in the '***Select***' color.

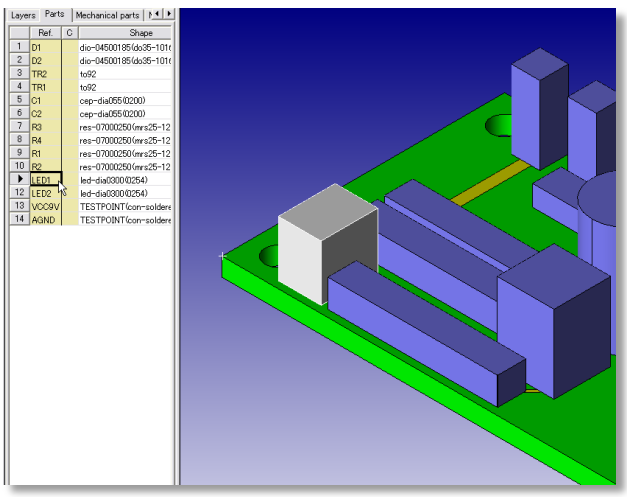


5. While still holding down the <Shift> key, click on the PCB board in an area where there are no components. The board is also added to the selection set.



6. Now click on the background of the graphics panel where there are no design items. This will clear the current selection.

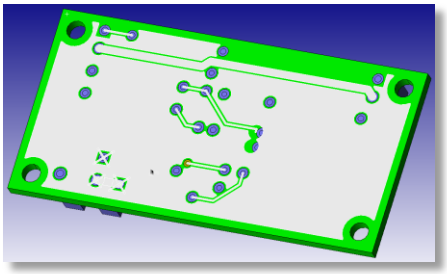
7. In the *Control Panel* (to the left of the Graphics Panel), make sure that the Parts tab is displayed and try selecting a row of the table (for example, the one corresponding to LED1). The corresponding component will be highlighted in the graphics panel in the '***Select***' color.



8. To select a range of components (e.g. from C1 down to R2), first click on the C1 row in the table then hold down the Shift key and click on R2. All the components in-between will also be selected and highlighted on the display.

Note: If you select components by clicking on the design in the graphics panel, the corresponding rows in the table on the ***Parts*** tab are also selected.

9. Flip the design (Camera View) over so that you are looking at the bottom of the board (using the Rotate command) and add the 'Conductive Pattern' filter to the active set.

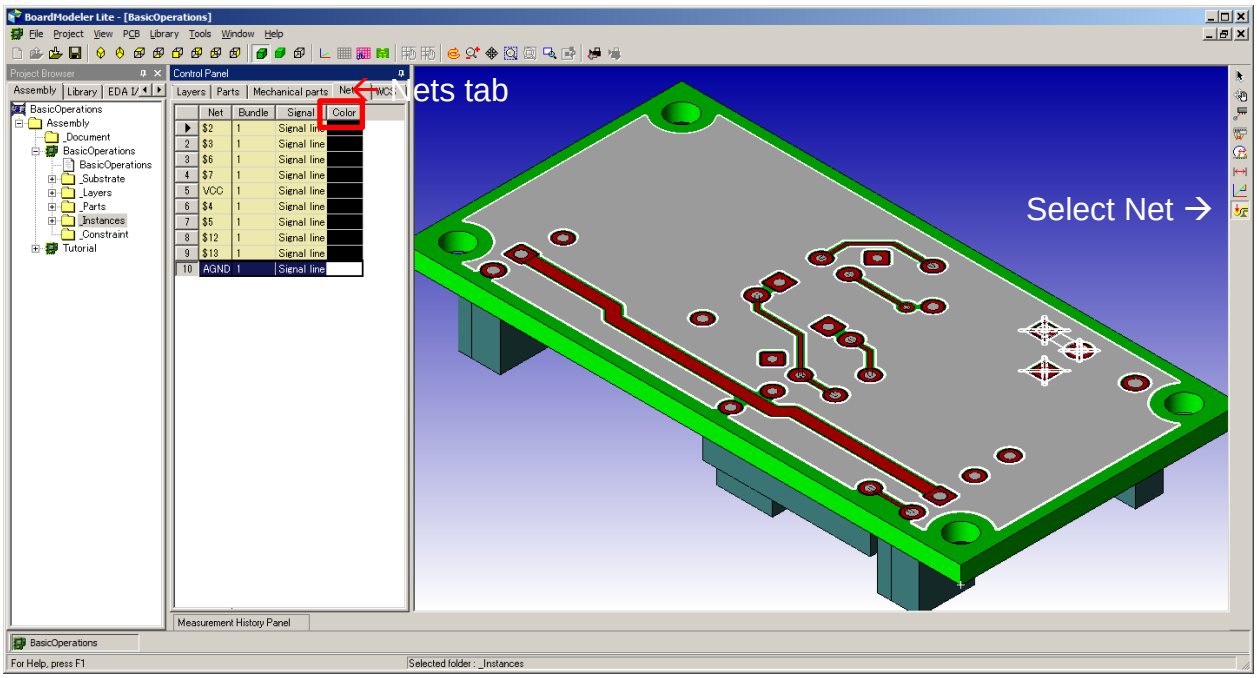


10. Select any part of the large copper area and notice that it is selected along with all the routing patterns.

11. Change the Camera view to *A-Side Isometric* using the tool bar icon.

12. Click the ***Select Ne***t tool bar icon on the right side of the graphics panel. This will change the **Control panel** to the [**Nets**] tab.

13. Select the net ***AGND***. Select nets ***$13*** and then ***$12***.



Tip: Change the ***Rendering of Parts*** to ***Transparent*** or ***Hide*** to make is easier to see net objects.

Data transferred from CADSTAR into the BML design, such as silkscreen and placement outlines, are actually part of the component footprint and are not separately selectable items themselves (i.e. they cannot be selected by clicking on them in the graphics panel). However, it is possible to view these items and control the visibility of them by turning on or off the relevant layer in BML. This is explained further in the following section.

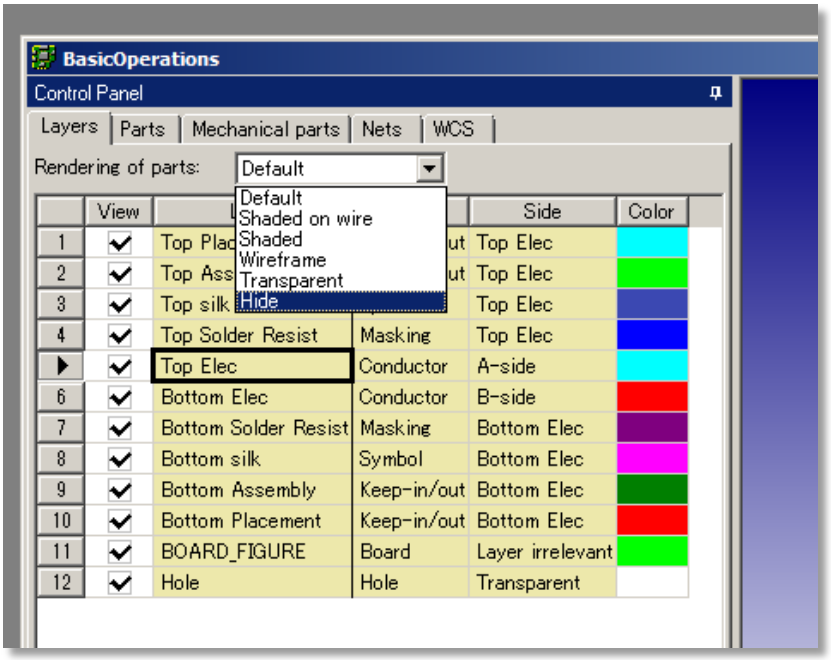
End of Task.

Viewing by Layer

BoardModeler is a layer based application with links to 3D objects. Each Layer is a 3D object in itself.

This section will help you to see the layers that are imported from the host CAD system. The CADSTAR Layer names are listed in the Control Panel – Layers tab. The Layers tab controls both the layers' visibility and color.

Note: All layer colors can be changed except BOARD_FIGURE.



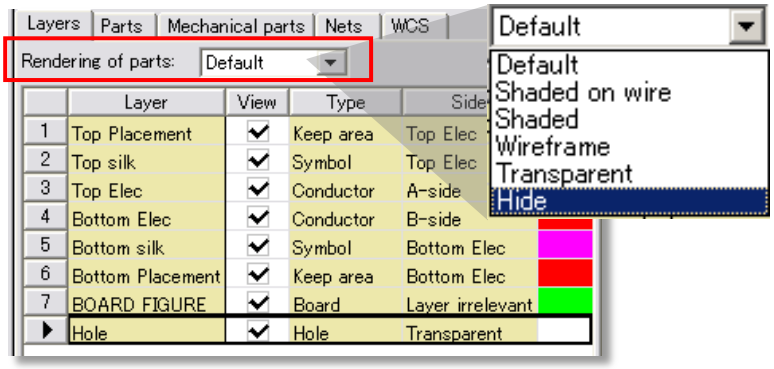
Task 4 – Viewing by Layer

1. Click the Layers tab in the Control Panel

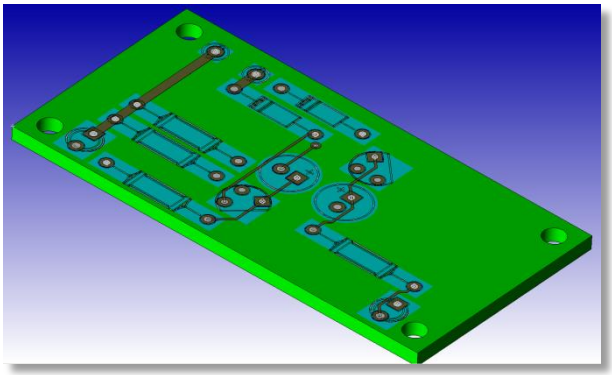
2. Change the Graphical view using the Isometric view icon.

3. Tick each box in the View column to make the layers visible as in the image shown below,

Tip: Clicking the **View** column header button will turn all layers on and off simultaneously.



4. From the ***Rendering of parts:*** drop-down list box, select the ***Hide*** option to hide all the components in the design. You should end up with a screen similar to that shown →

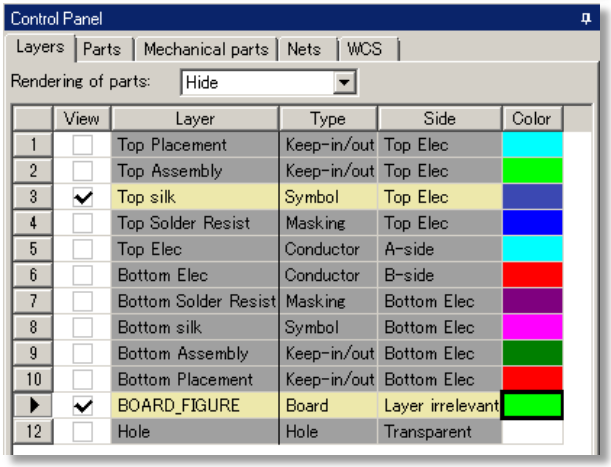


In the ***Layers*** tab of the Control Panel you should notice that there is a 'tick' in the ***View*** column for every layer listed as shown above.

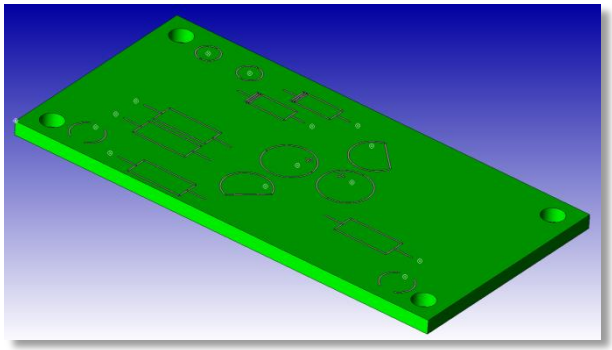
5. Experiment by turning or ticking layers on and off to see the appearance in the graphical window.

6. Clear all the ticks except for the 'BOARD FIGURE' and 'Top silk' layers.

7. Try changing the color of *Top silk* to white. This should be the result... →



Note: When creating closed component outlines in CADSTAR, BML will interpret them as filled.
To avoid this, change any closed shapes in your PCB components to ***Open.*** They will appear as shown.

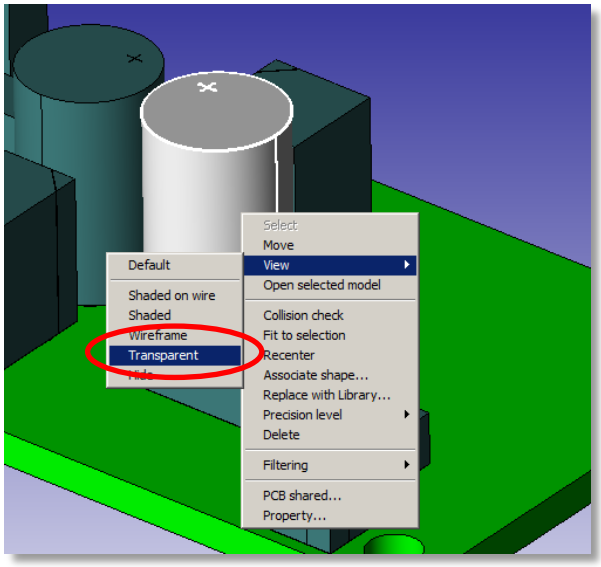


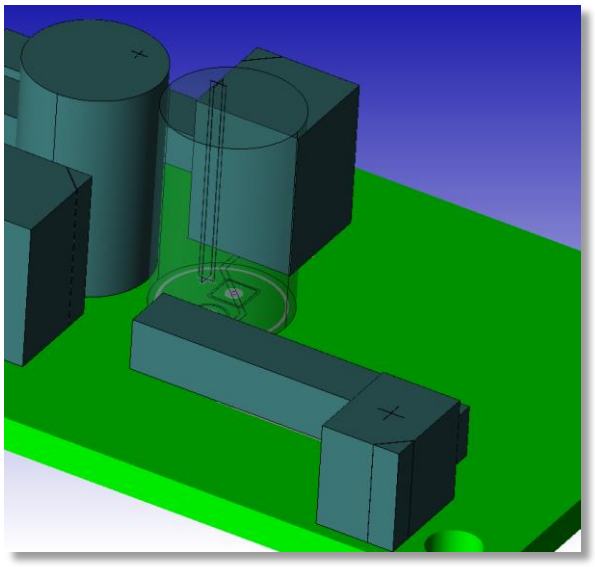
End of Task.

Controlling the Display of Individual Parts

By using a combination of the operations we have learned up to now, it is also possible to have different display settings for individual components (parts) within the design.

First set the display so that all components are visible again (from the drop-down list box at the top of the Control Panel – select the 'Default' option). Now select capacitor C2 in the design (using any of the methods from above) and press the right mouse button. A pop-up menu will appear.

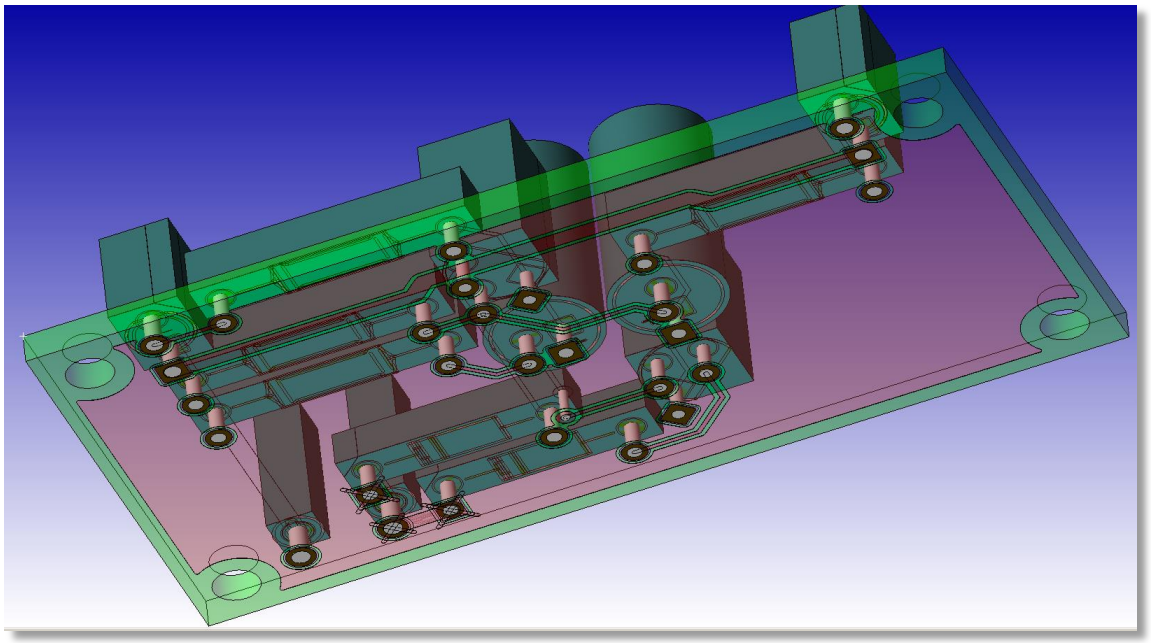




For the moment, ignore all the other options and select View to show the additional menu. Select the 'Transparent' option from the menu and observe the result. (As shown above)

Note that only component C2 is affected. Try assigning different options to other components and see that you can combine different viewing options (rendering modes) on a per component basis.

You can also apply individual rendering modes to the board and conductive patterns etc., allowing views of the design such as that shown below to be created.

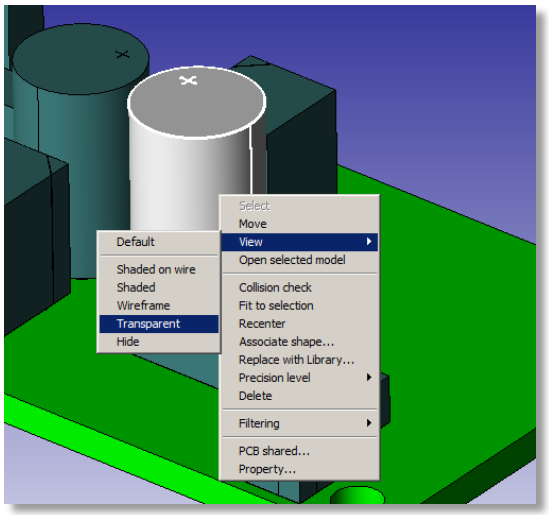


Task 5 – Controlling the Display of Individual Parts

1. From the ***Control Panel [Layers]*** tab, select the '***Default***' option for ***Rendering of parts.***

2. Select capacitor C2 in the design (using any of the methods from above) and press the right mouse button. A pop-up menu will appear.

3. For the moment, ignore all the other options. Using the Right Mouse Button, select the ***View →'Transparent*** option and observe the result.



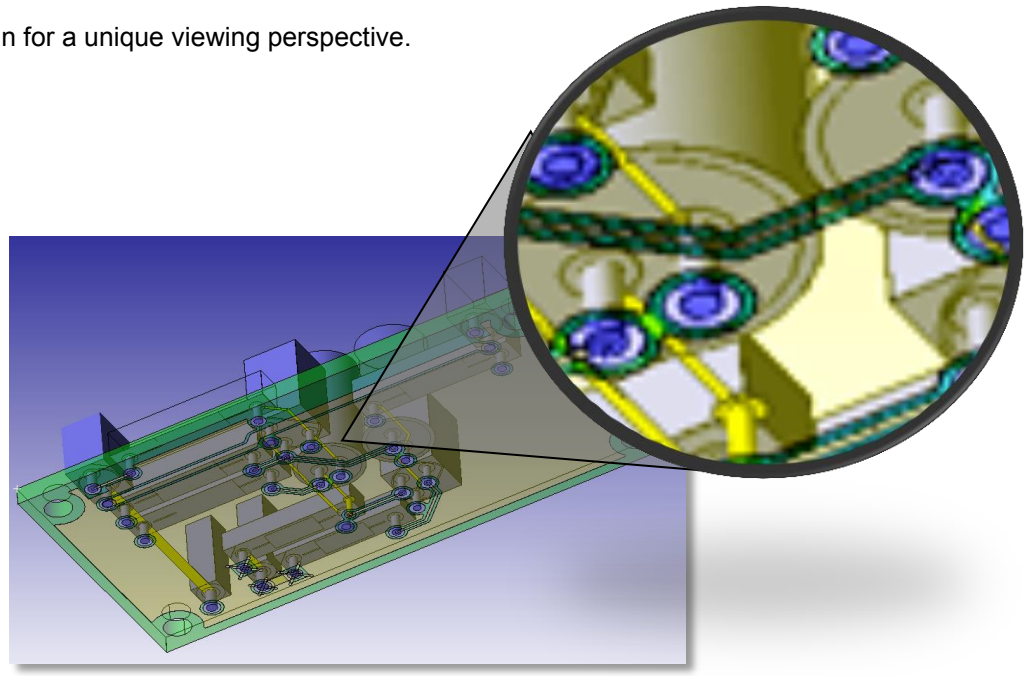
Note: only component C2 is affected.

4. Try assigning different options to other components and note that you can combine different viewing options (rendering modes) on a per component basis.

5. Select the Board shape and apply individual rendering mode such as transparent.

6. Select the conductive metal patterns,

7. Zoom in for a unique viewing perspective.



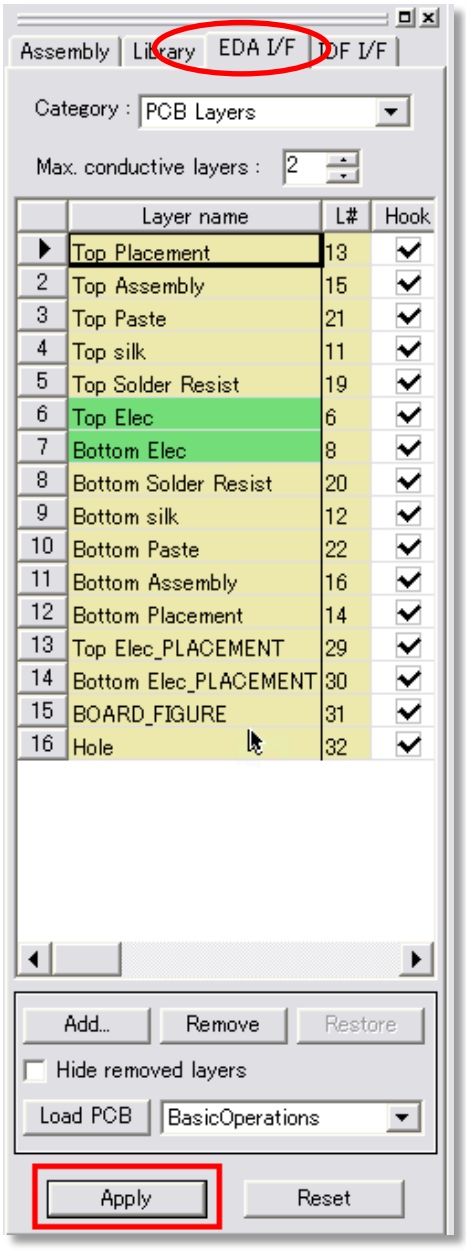
End of Task

Preparing BML for use with a CADSTAR Layer Stack

In order to setup a proper CADSTAR to BML relationship for seamless collaboration, we need to provide BML with some information about a default CADSTAR layer stack. This will allow us to start a design within BML, importing a board outline and a placement height limitation area which we can then transfer back to CADSTAR. To do this we need to know which layer should be associated with the height limitation area.

In the Project Browser, switch to the [***EDA I/F***] tab. You will see the layers defined which match the layer-stack for the current design.

Simply select ***Apply*** button. This will update the configuration for BML and allow it to use this layer data as a default when a new project is created locally within BML (without passing any design data from CADSTAR first). This process will become clear in the next section of the tutorial.



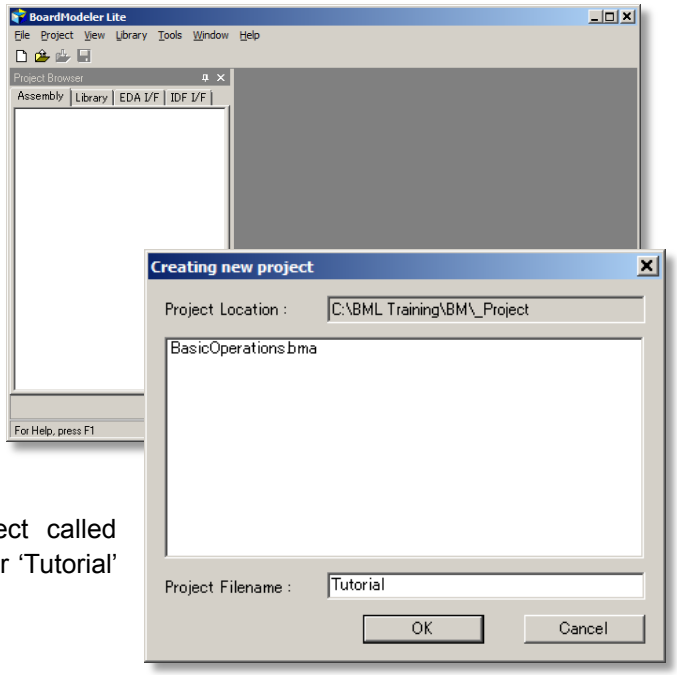
Task 6 - Working with Design Data

Now that we have become familiar with some of the basic operations of BML, we can move on to working with design data. In the next section we will import a board outline into BML – which we have theoretically received from the MCAD department, set up the origin of the board and finally load this data into a new CADSTAR design which only contains the unplaced components as a result of passing a design forward from CADSTAR schematics.

We will also see how placement changes can be made in BML and back-annotated into CADSTAR and how we can choose how components are modelled and displayed within BML (from simple extruded 'boxes' to complex, true 3D models).

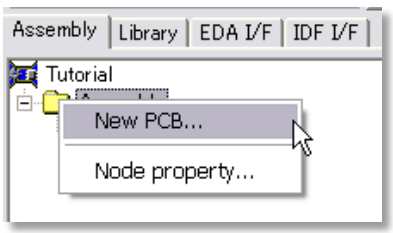
Starting a new PCB Design

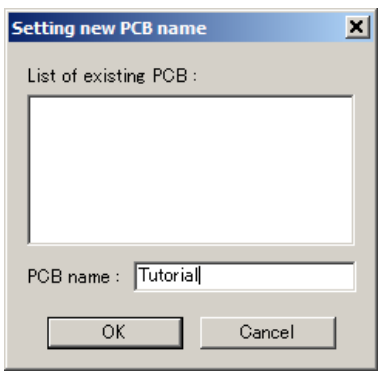
1. If you still have BML running then **select File→Close→Project**. (you don't need to save it), otherwise select **All Programs→ BoardModeler Lite→BoardModeler** from the Windows Start menu to run it up again.

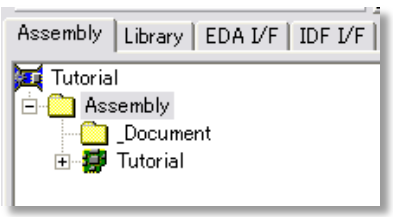


2. We are going to create a new BML project called 'Tutorial'. Select ***File→New→Project*** and enter 'Tutorial' in the *Creating new project* dialog.

3. In the Project Browser click the right mouse button on the Assembly folder and select New PCB from the popup menu. Enter 'Tutorial' again as the name of the PCB.



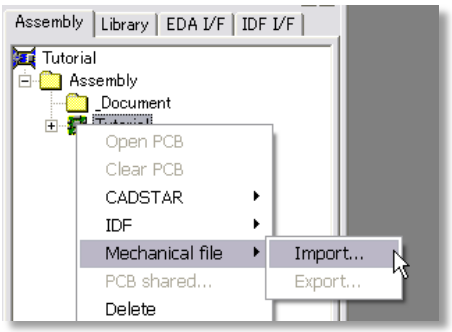




The Project Browser tree will update to show the new contents of the project.

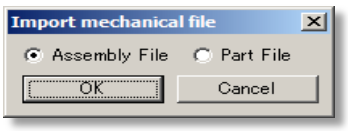
Importing a Mechanical file

4. We will now import the board outline to be used for this design from a STEP format file supplied to us by the MCAD department.



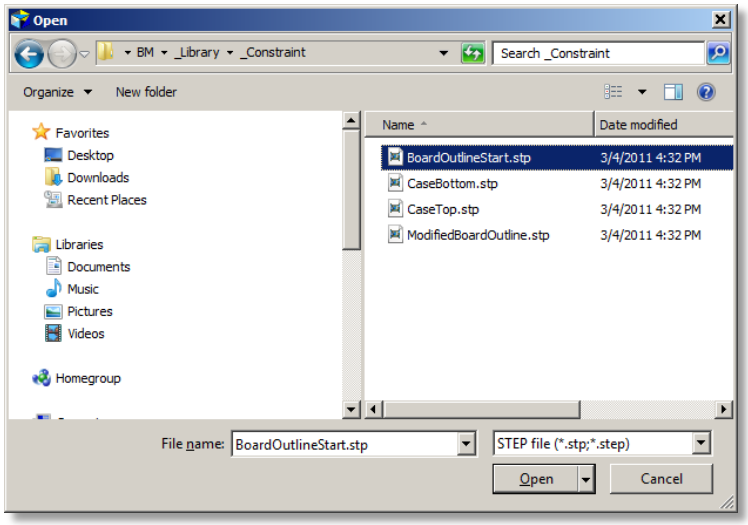
5. Right-click on the *Tutorial* PCB folder in the *Project Browser* and select **Mechanical file→Import**... from the popup menus. [Alternatively, ensure the Tutorial PCB folder is selected in the Project Browser and then use the ***File →Mechanical file→Import***... menu].

   For the next dialog you must tell BM how to support the mechanical file that is about to be imported. It is either an Assembly File or a Part File. Since the file contains more than one part it is an Assembly file.

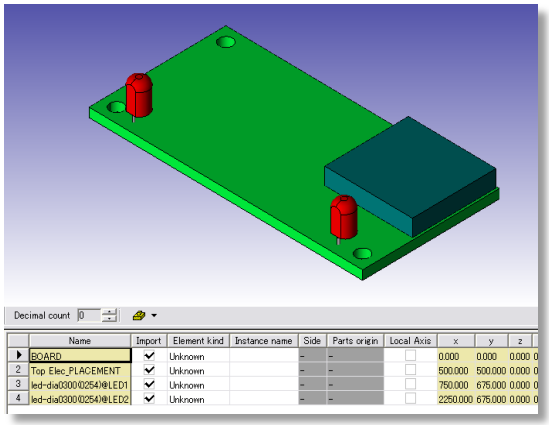


   Note: it is important to request this type of data from your MCAD system. 'Assembly' is a STEP structuring level and should be understood by most/all MCAD systems that support STEP import/export.

6. Browse to the ***C:\BML Training\BM_Library_Constraint*** folder in the *Open* dialog and select the 'BoardOutlineStart.stp' file.



   The STP file is converted to an ACIS file which is native to the 3D kernel being used. Upon completion, a report dialog will be displayed. If the dialog contains any output you can open the report to review it or else select **[Close]**

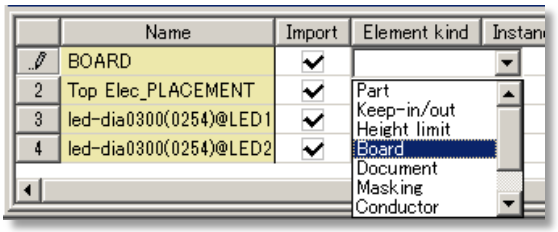


   The file will be imported and the display should look similar to the image shown.

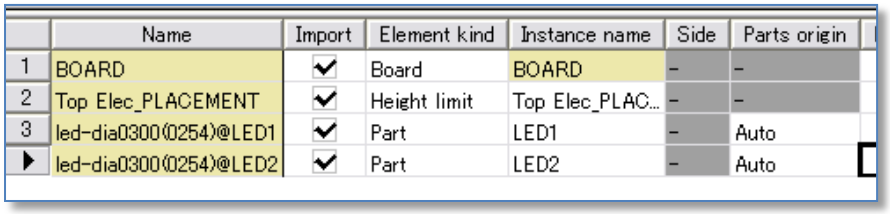
   As you can see, we have not only imported the board outline shape, but two pre-placed components and an additional shape that represents a height limitation area.

We now need to do a small amount of set-up so that BML knows how to interpret these different items of data.

7. In the table at the bottom of the Graphics Panel, for the first item, set the Element kind to be 'Board'. The Instance name will be automatically set to BOARD. [Note that you may need to either press the Enter key or click in the Instance name field after selecting the *Element kind* from the drop-down menu



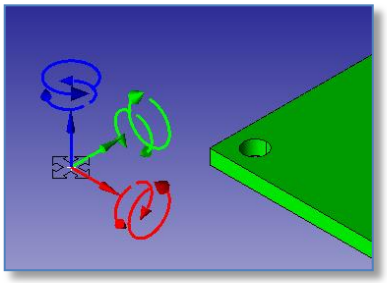
8. Set the remaining entries in the table to match those shown in the image below.



	Name	Import	Element kind	Instance name	Side	Parts origin
1	BOARD	✓	Board	BOARD	-	-
2	Top Elec_PLACEMENT	✓	Height limit	Top Elec_PLAC...	-	-
3	led-dia0300(0254)@LED1	✓	Part	LED1	-	Auto
▶	led-dia0300(0254)@LED2	✓	Part	LED2	-	Auto

Setting the Origin

Now all that remains to be done is to set the origin for the board. We are going to locate it at the bottom left-hand corner (seen when looking down on the board from directly above).

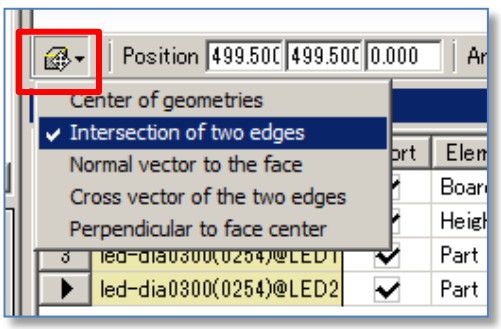


9. From the 3D Model Toolbar select the Move Origin icon and  this will display the standard move 'manipulator' in the Graphics Panel.

This could be used to move the origin interactively by dragging it. In this case, we will specify the exact location where we want the origin to be as the intersection of two sides of the board.

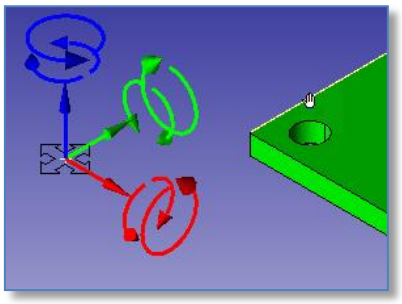
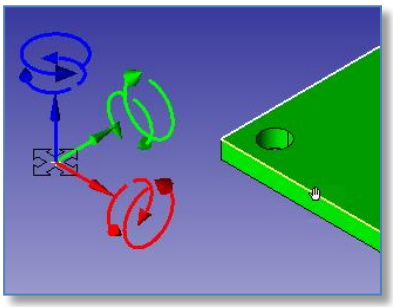
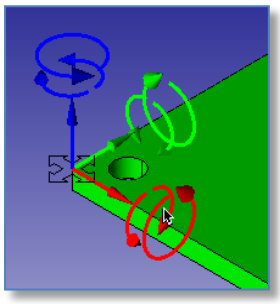
Note: the pre-placed components and height restriction area will not be displayed during this process).

10. From the tool bar icon at the bottom of the Graphics Panel(tool tip: ***Ways to specify the origin***, select the ***Intersection of two edges*** option. →

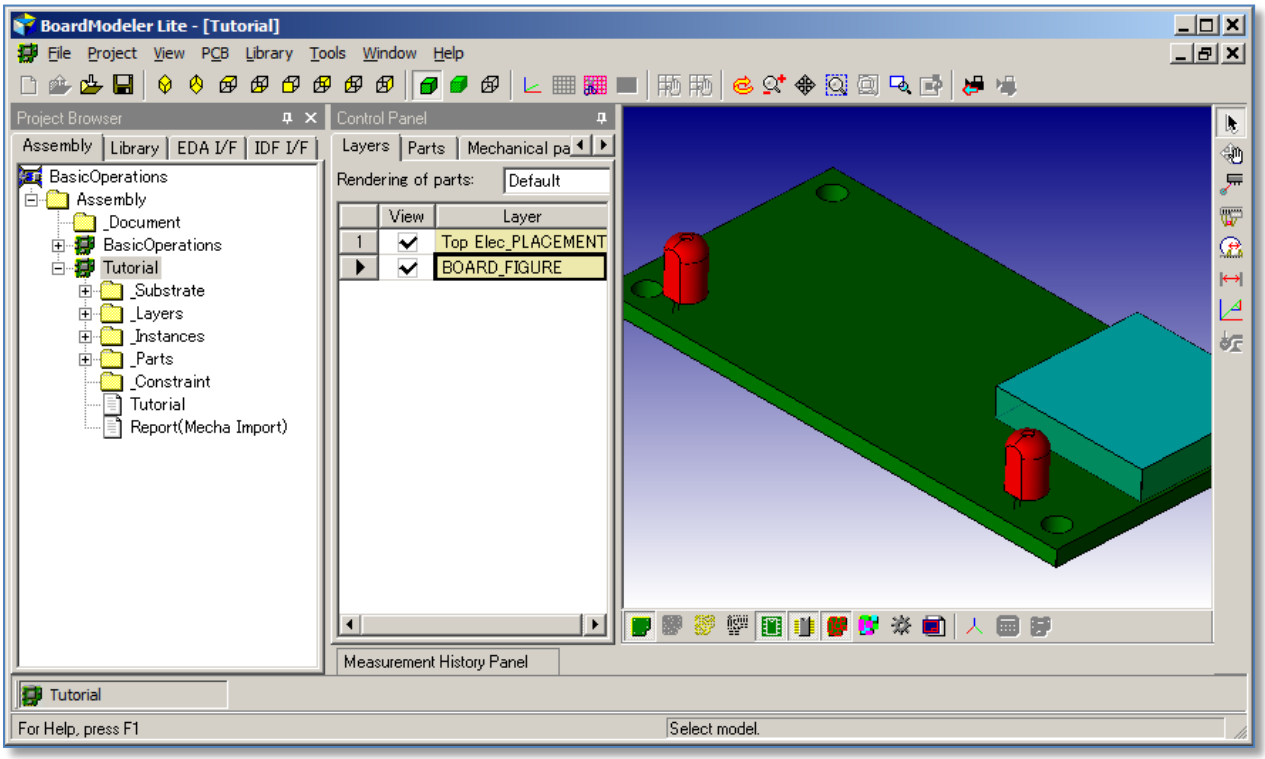


11. In turn, select the two edges of the board which intersect at the bottom left-hand corner.

Once the second edge has been selected the origin will be moved to the intersection point.

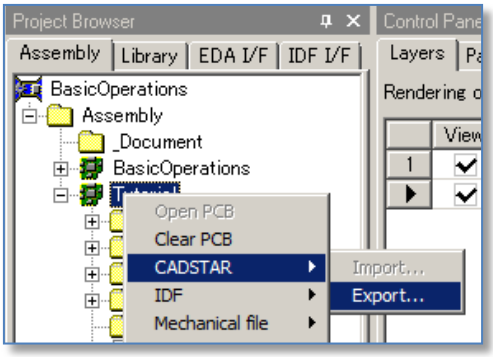




12. Finally, select the bottom icon from the 3D Model Toolbar to accept all the changes we have made and 'register' this information in the project.



Importing BML data into CADSTAR

We have now prepared the board outline shape with the required origin and are ready to export this data for use in CADSTAR Design Editor.

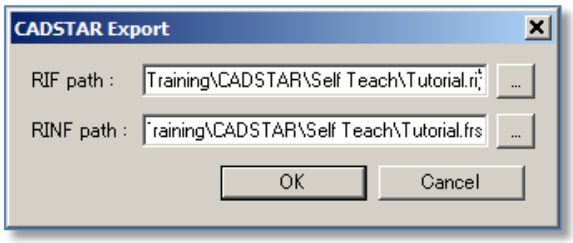


1. Right-click on the Tutorial PCB folder and select ***CADSTAR→ Export...*** from the pop-up menu.

2. Set the following paths for the RIF and RINF files in the CADSTAR Export dialog and then select [OK]

   C:\BML Training\CADSTAR\Self Teach\Tutorial.rif
   C:\BML Training\CADSTAR\Self Teach\Tutorial.frs

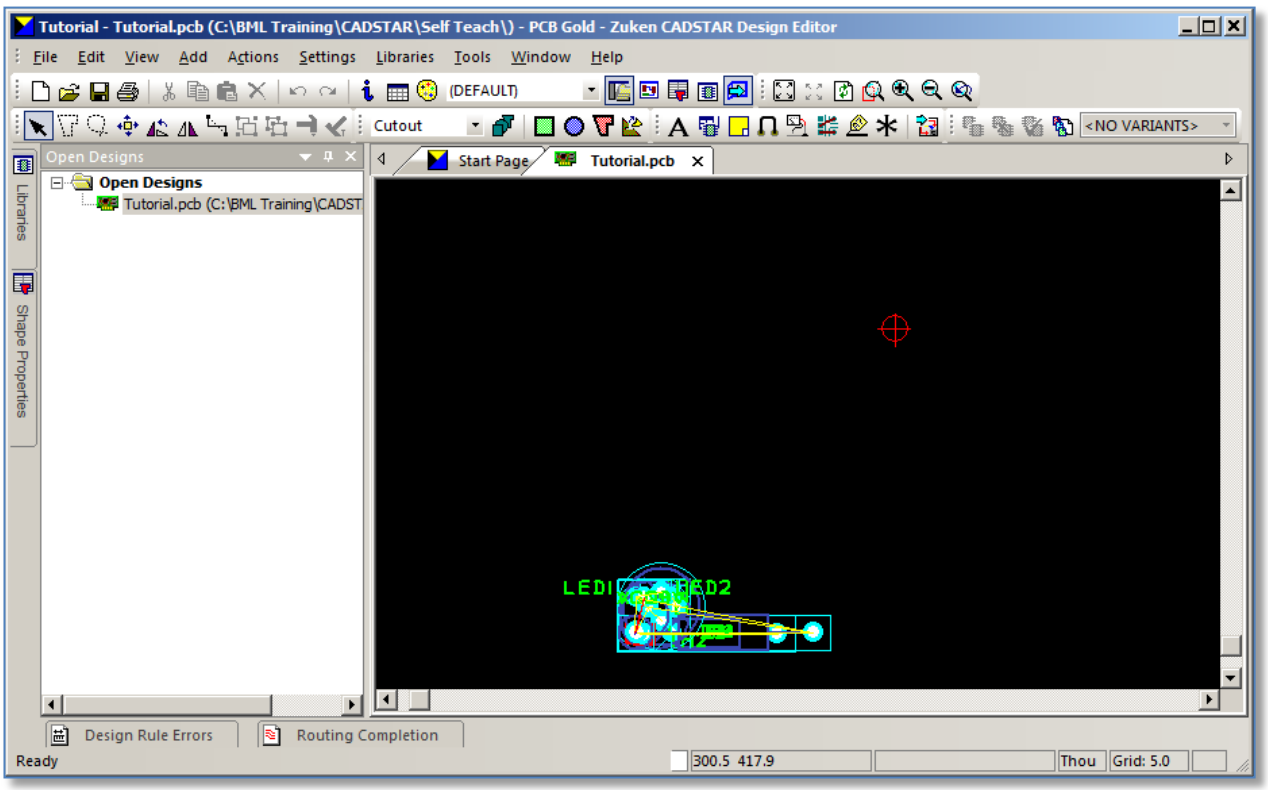
   Make sure you include the correct file extensions (.rif and .frs) in the names.



3. Select ***File→Exit*** and then choose **Yes** when prompted to save the data and quit BML.

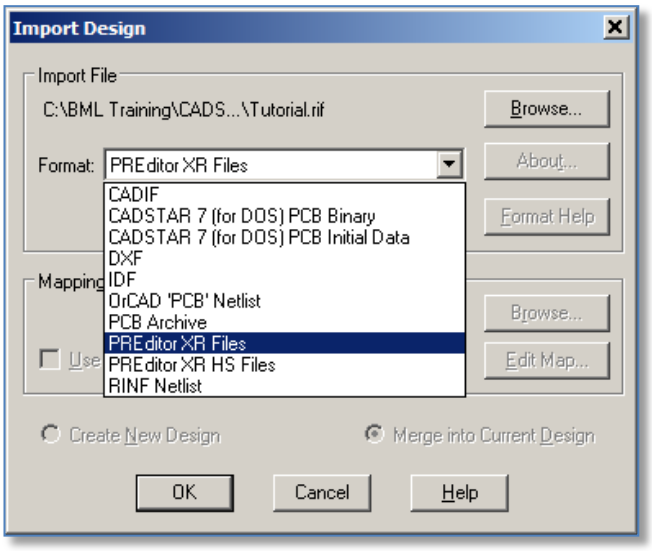
NOTE: If BML was run up from the **Tools** menu in CADSTAR you will be now be prompted to reload changes into the CADSTAR design. Select ***No***, and close the design without saving.

4. If it is not already running, start CADSTAR and load the 'Tutorial' workspace **('C:\BML Training-\CADSTAR\Self Teach\Tutorial.csw')**. The display should look similar to that shown below.



5. Select the ***File→Import***... menu option and set the format to be ***P.R.Editor XR Files.*** Use the [Browse] button to select the 'Tutorial.rif' file from the 'C:\BML Training\CADSTAR\Self Teach' folder.

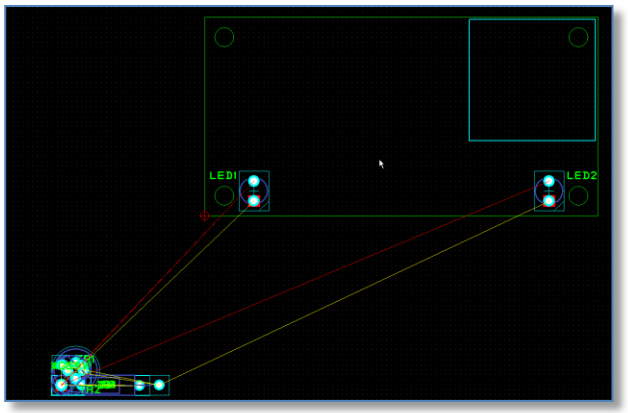
   Do not change any other settings and simply select [OK].



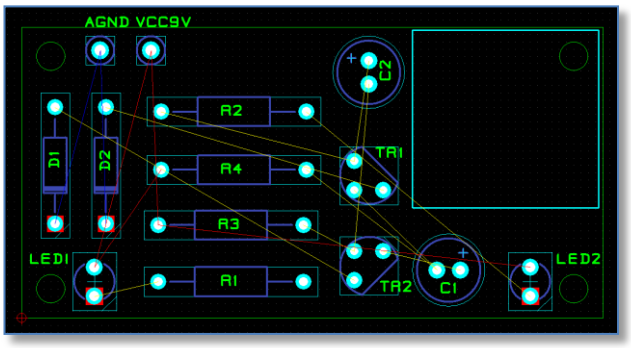
   The board outline and height limitation area shapes from BML will be imported into the CADSTAR PCB design.

6. Repeat the same ***File→Import***. process but this time set the Format to be RINF Netlist and use the Browse button to select the 'Tutorial.frs' file from the 'C:\BML Training\CADSTAR\Self Teach' folder.

   The two pre-placed components (LED1 and LED2) will be positioned in the correct location on the board.



7. To quickly place the remaining components on the board and for the purposes of this tutorial, import the RINF file ***StartPlacement.frs*** from the ***C:\BML Training\CADSTAR\Self Teach*** location.

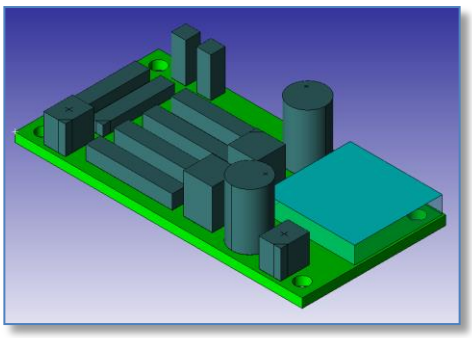


8. Using the same process which we used earlier in the 'Basic Operations' section of the tutorial, transfer this design to BoardModeler Lite. This will be used in the next Tutorial section.

Note: When creating a new PCB design using Mechanical data files, the CADSTAR PCB design must exist containing the parts. This process will not import P.R.Editor XR file into a new CADSTAR PCB Design unless a netlist is present.

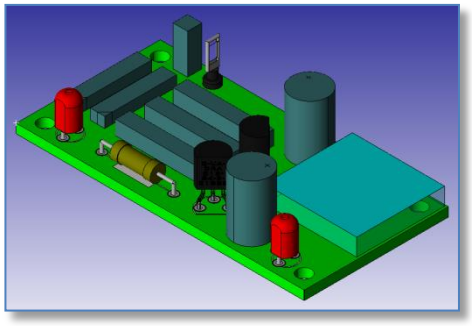
End of Task

Changing How Parts are Displayed in 3D



Your design in BML may look similar to the image shown if you chose to use the Top Placement layer component shapes. The default 3D view of the components on the board is a very simple representation, created by extruding the shape drawn on the placement layer (in the PCB footprint) in CADSTAR to a defined height. The height of each component can be specified either by adding an attribute to the part definition in the CADSTAR library, or by setting the height directly in the component footprint. In this tutorial, we have used the second of these two methods.

However, this simple representation may not be accurate enough for our needs (for example to check for collisions between components and a system case or chassis) and we may have more detailed 3D models available for some or all the components we are using in our design.



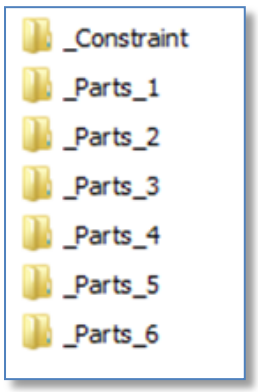
With BML it is very easy to replace the simple representation with a more detailed 3D model and also to switch between the different representations as required.

Many parts vendors, such as manufacturers of connectors,, will supply their 3D CAD data on their website in either STEP or ACIS file format.

From BML 5.2 onward you will have access to the 3D Parts Library Wizard.

An objective at this stage is to collect the critical 3D part geometries and store them in a safe place. This is commonly located in the BM Installed DOCUMENT folder.

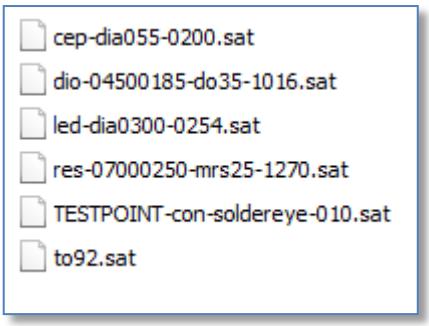
For this Tutorial we have provided you with some higher details shapes in a project structure in place. See "**C:\BM Training\BM\Library"**



Mechanical enclosures – Items used for collision checks – BML Tutorial data

Lowest level of parts detail

BML Tutorial 3D Parts are stored here.

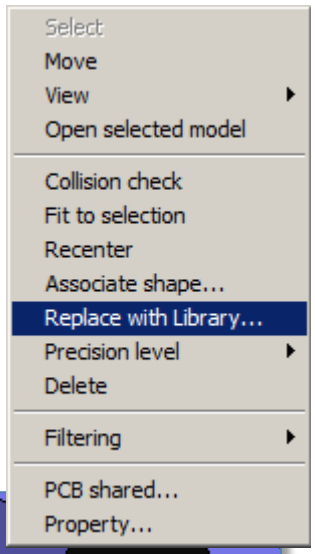


Highest level of Parts detail

.

Task 7 - Changing How Parts are Displayed in 3D

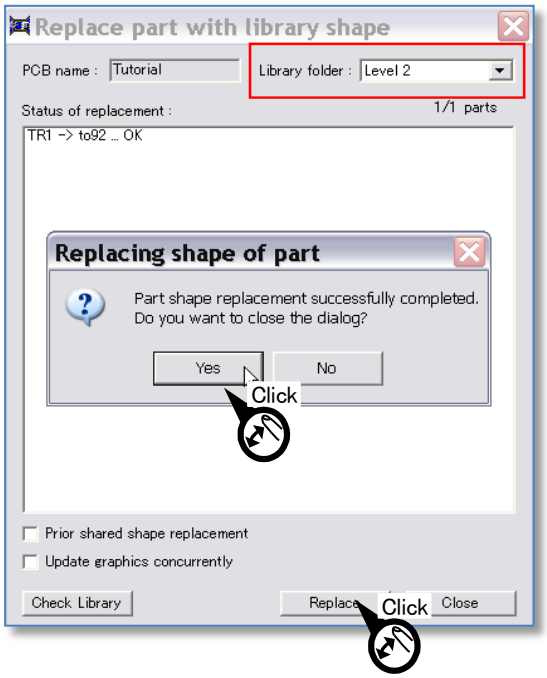
1. Select one of the transistor components (TR1 or TR2) in the design and click the right mouse button. From the assist menu select the ***Replace with Library...*** option.

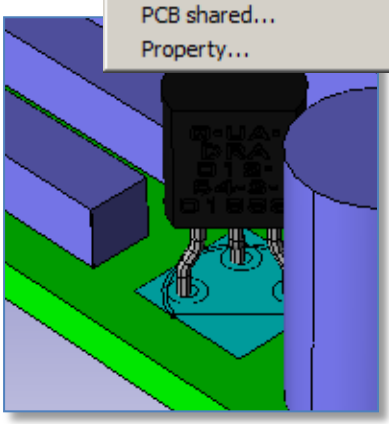


2. In the dialog that now appears, ensure that the Library Path field is set to "Level 2" and select the ***Replace*** button.

   Click [Yes] to confirm that the dialog should be closed.

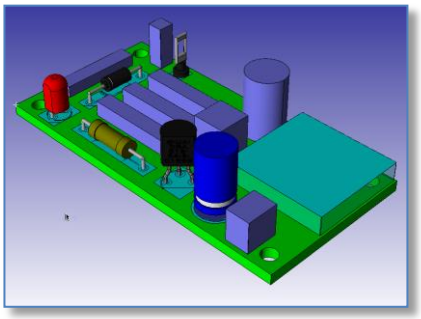
   You should now notice that the simple representation for the selected transistor has been replaced by a more detailed model.

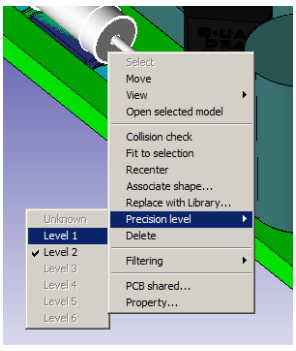


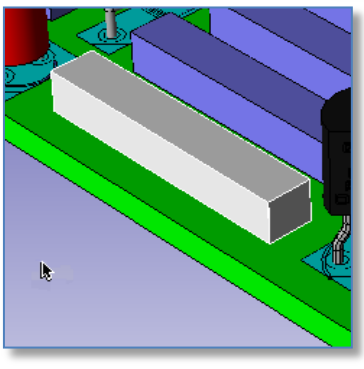


3. Now repeat the same process with a few more of the components in the design.

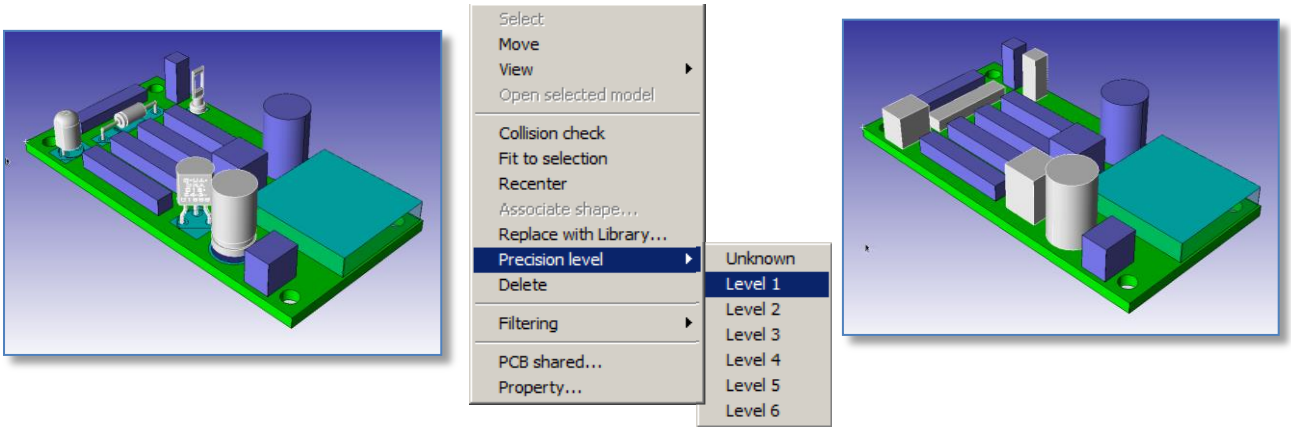
   Once you have done that, select one of the components with a detailed 3D model and then click the right mouse button. From the resulting pop-up menu select Precision level and then Level You will notice that the displayed representation for the component has now returned to the default, simple representation.







You can also select multiple components (using <Shift >+ click or by drawing a selection box round the components) and set the precision level for all selected components in one operation.



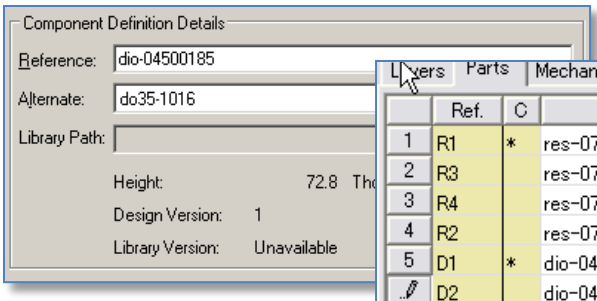
It is possible to make the display of the detailed 3D models the default option when the design is passed from CADSTAR to BML and we will see that in operation very shortly.

The relationship between the components used in the design and the detailed 3D models is handled automatically based on 1 of 4 methods

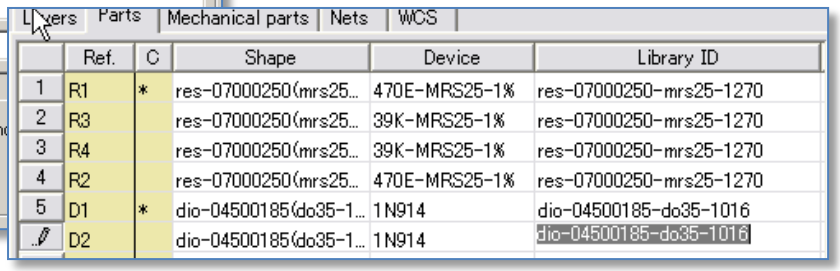
1. the 'Reference' and 'Alternate' names defined for the parts used within CADSTAR matches the name of the 3D model.
2. The Part name in CADSTAR matches the name of the 3D Model
3. A user attributed such as "3D_model" is added in the CADSTAR Part library or just with in the local design attributes.
4. Using an XLS file that can be referenced by BML

If we take the diodes (D1, D2) as an example, in CADSTAR (using Item Properties) you can see that the reference is 'dio-04500185 ' and the alternate is 'do35-1016 '. Based on this, BML will look for a 3D model to represent this part with the following name: dio-04500185-do35-1016.sat.

This mapping can be seen on the ***Parts*** tab of the Control Panel in BML, in the ***Library ID*** column – as shown below.



Item properties in CADSTAR



	Ref.	C	Shape	Device	Library ID
1	R1	*	res-07000250(mrs25...	470E-MRS25-1%	res-07000250-mrs25-1270
2	R3		res-07000250(mrs25...	39K-MRS25-1%	res-07000250-mrs25-1270
3	R4		res-07000250(mrs25...	39K-MRS25-1%	res-07000250-mrs25-1270
4	R2		res-07000250(mrs25...	470E-MRS25-1%	res-07000250-mrs25-1270
5	D1	*	dio-04500185(do35-1...	1N914	dio-04500185-do35-1016
	D2		dio-04500185(do35-1...	1N914	dio-04500185-do35-1016

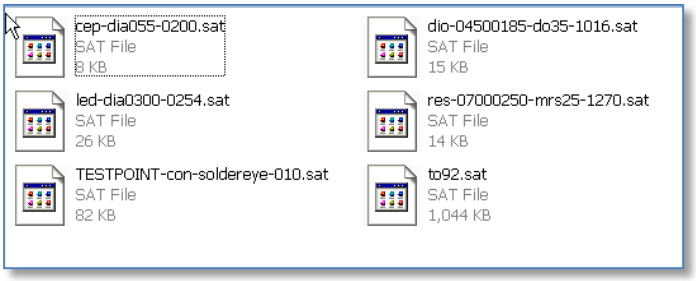
Control Panel in BML

The file name for the 3D model is built by combining the reference name with the alternate name, separated by a hyphen. The '.sat' file extension is added automatically. If, as in the case for the transistors, the alternate name is blank, the name of the 3D model simply becomes the CADSTAR reference name (to92.sat).

If you use Windows Explorer and navigate to the folder at

{x}:\BML Training\BM_Library_Parts_2

You can see the 3D model files used for this tutorial design.



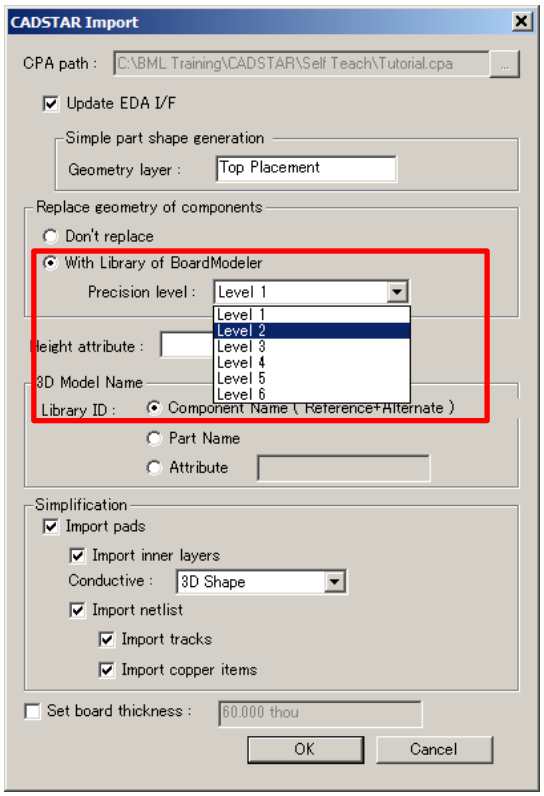
Note: For existing CADSTAR 3D users, BML provides a very simple and straightforward way of converting any existing 3D model libraries into a form suitable for use with BML]. See Appendix - A

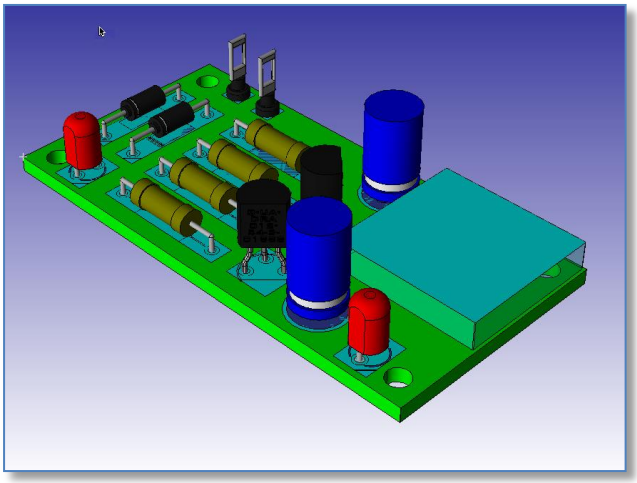
Now that we have seen how to change the representation for individual components interactively within BML, let's look at how to have the detailed 3D models displayed by default when the design is passed from CADSTAR to BML.

4. Exit from BML (***File →Exit***) and return to CADSTAR – there is no need to save the project.

5. Select ***Tools→BoardModeler***... to pass the design to BML again in the same way as previously – but we now need to change the settings on the CADSTAR Import dialog.

   Make sure the settings are as shown in the image – specifying that we want to use the representation from the BoardModeler library at precision level 'Level 2'.

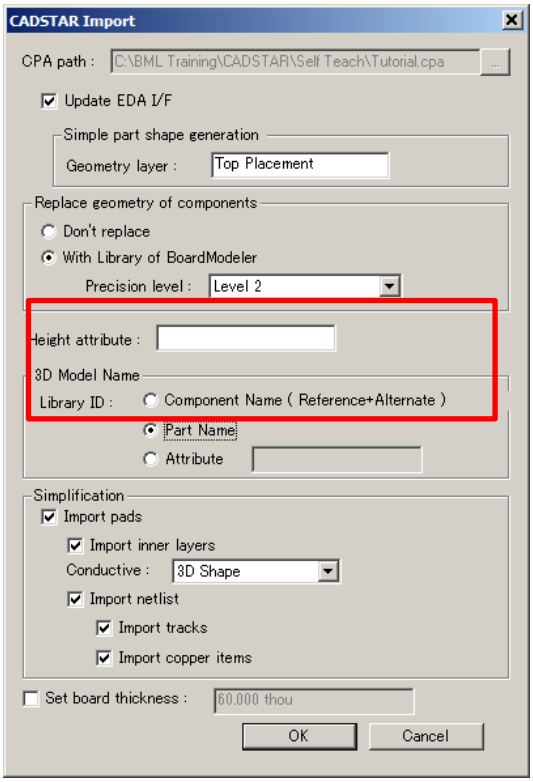




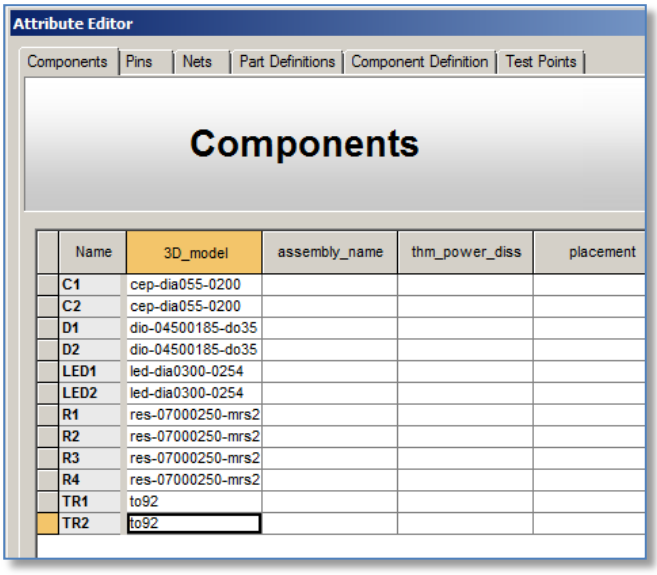
When the design is loaded into BML the detailed 3D models will be used immediately.

You may switch back to the simple precision for individual components in exactly the same way as described above.

As an alternative to using the combination of the reference name and alternate name to form the name of the detailed 3D models, it is possible to configure BML to use the part name instead. To use this option, ensure that the relevant 3D models are named correctly based on the CADSTAR part names (e.g. *'1N914.sat'*) and select the ***Part name*** option for the ***Library ID:*** field on the ***CADSTAR Import*** dialog.



Another alternative is to create an Attribute of Components in the CADSTAR Parts Library or within the local attributes of the PCB design (Shown below). For *those* 3D models specified in the attribute value they will automatically be used when the **3D Model Name – Library ID:** is set to *Attribute*.



Name	3D_model	assembly_name	thm_power_diss	placement
C1	cep-dia055-0200			
C2	cep-dia055-0200			
D1	dio-04500185-do35			
D2	dio-04500185-do35			
LED1	led-dia0300-0254			
LED2	led-dia0300-0254			
R1	res-07000250-mrs2			
R2	res-07000250-mrs2			
R3	res-07000250-mrs2			
R4	res-07000250-mrs2			
TR1	to92			
TR2	to92			

End of task

Task 8 - Moving Parts in 3D and Collision Checking

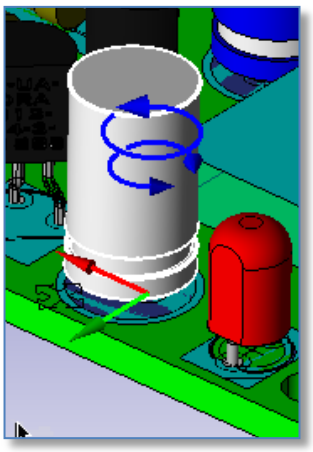
Now that we have our design in 3D, we can investigate how to change the component placement by moving components in the 3D environment. During this process we can also perform on-line 'collision checks' to make sure that we are not causing errors in the design.

We will also investigate the use of the '***measure***' functionality to measure sizes and distances and the 'clearance' tool which allows us to identify all objects which lie within a specified clearance distance of a selected component or other design item.

Finally we will see that having changed the component placement, the new component positions are back-annotated to CADSTAR to update the 2D PCB design.

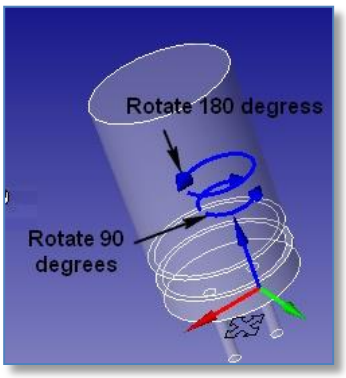
1. Start by selecting capacitor C1 and then chose the '**Move'** icon from the 3D Model Toolbar. 

   You will see movement 'manipulator' appear, at the components origin. →

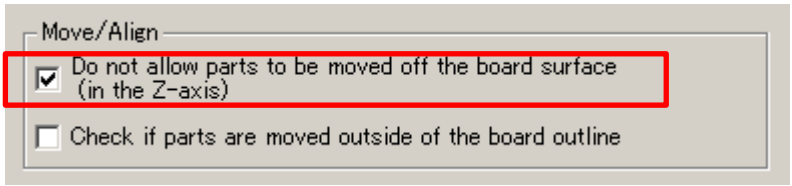


Selecting and dragging on the red, green or blue-coloured arrows (if visible) will move the component, but it will be constrained to move only in the direction of the axis you have chosen (corresponding to the arrow).

The component can be rotated by clicking on the blue circular-shaped arrows located at the top of the manipulator. The top control (with only an arrow at one end of the arc) will rotate the component through 180 degrees, whereas the lower control (separate arcs with an arrow on each one) will rotate through 90 degrees.



It is possible to disable the grab handles in the Z-axis. (i.e to lift items off the board). This setting is located in the ***Tools→Options[Other]*** tab. If you did not select this option, then you will also see a third, blue arrow forming the Z-axis part of the manipulator

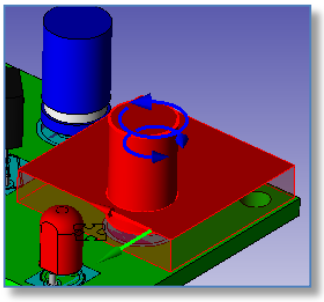


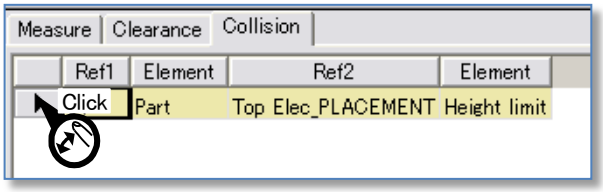
Note: Z-axis movement is not back annotated to CADSTAR as a height change. However it will affect the clearance measurements in BML which may not be valid.

2. Before we move the component, select the icon at the bottom of the graphics panel to Synchronize Collision Check.

Now click and drag on the small black cross with 4 arrows and move the component so that it lies somewhere within the height limitation area and then release it.

Notice that the height limitation area and the component has turned red and that an entry has appeared in the 'Collision' table. This shows that component C1 is now in error against the height limit area associated with the Top_Elec_PLACEMENT layer.

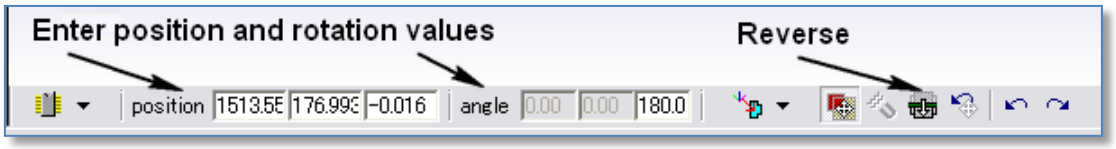




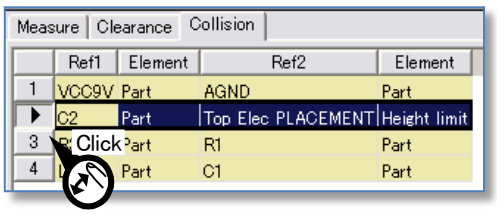
3. Correct this error by moving the component (in the same way as before) until it is clear of the height limit area, or we could simply use the Reset icon (from the bottom of the graphics panel) to immediately return it  to its original location.

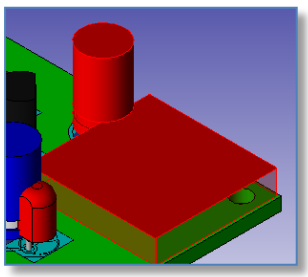
4. Experiment with these controls to see how they work until you are comfortable with their operation. You can always Reset the component position at any time or use the Undo/Redo icons to step backwards and forwards through individual move operations. Note that you can select multiple components (using 'Shift' click etc.) and move them together in one operation.

   Components may be swapped to the other side of the board using the Reverse icon (located at the bottom of the graphics panel) and component positions and rotations can be set by typing in exact values in the input fields (again, at the bottom of the graphics panel).

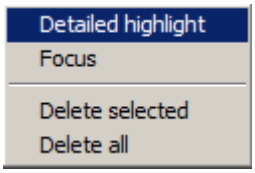


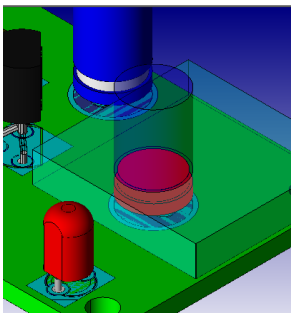
As you move the components, you should notice that for every 'collision' error that you cause, a line will be added into the Collision table below the graphics panel. If you click on any of these table rows, the corresponding items, which are involved in the collision, will be highlighted on the display.





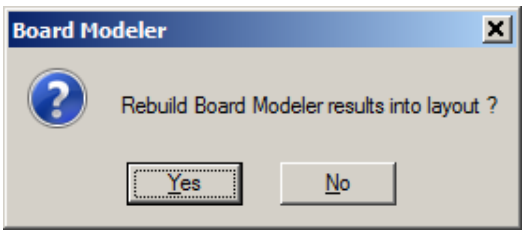
5. To make a finer detailed identification of the colliding surfaces, click the Collision item row in question, then click the <R.M.B.> and select Detailed Highlight





6. Once you have finished experimenting and you have a modified placement, save the project and exit from BML to go back to CADSTAR Design Editor.

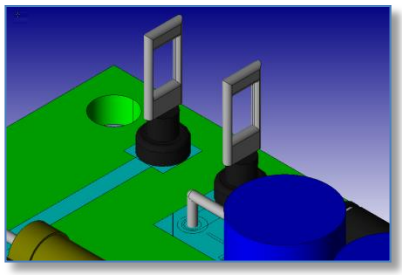
7. You will be asked to verify that you want to re-build the results of the changes made in BML, back into CADSTAR. Select ***Yes***, and notice that the placement of the components in CADSTAR has changed to match that set up in BML.



End of Task

Task 9 - Measuring Distances and Checking Clearances

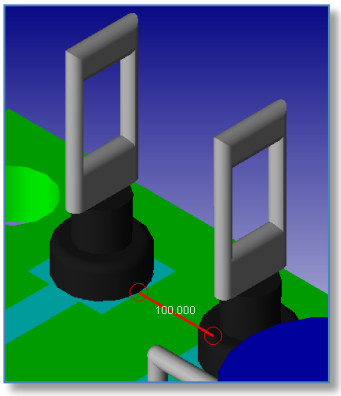
In this section we will investigate the use of the 'measure' functionality to measure sizes and distances and the 'clearance' tool which allows us to identify all objects which lie within a specified clearance distance of a selected component or other design item.



1. Load the '3D Enclosure' workspace into CADSTAR Design Editor (('*3D Enclosure.csw'* from the *'C:\BML Training\CADSTAR\Self Teach'* folder).

2. Transfer this design to BML in the usual way – select the option to use the detailed 3D models in the CADSTAR Import dialog (set the ***Precision level*** option to "Level 2").

3. Zoom/rotate the display onto the two testpoint components.



4. Select the ***Measure*** tool from the 3D model toolbar and select the two testpoints, one after the other.

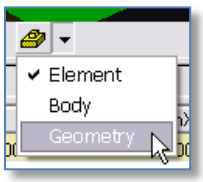
The minimum distance between the two components is measured, an entry is added to the Measure table (below the graphics panel) and a red line is drawn on the display to indicate this minimum distance.

Measure | Clearance | Collision

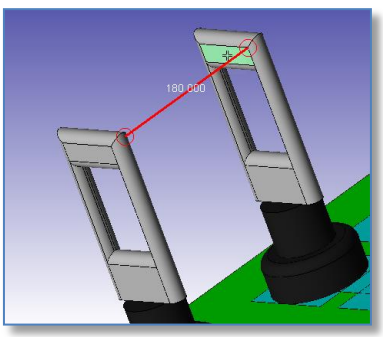
	Distance	Ref1	Ref2	x1	y1	z1	x2	y2	z2	LenX	LenY	LenZ	Time
▶	100.000	VCC9V	AGND	300.000	875.000	0.000	400.000	875.000	0.000	100.000	0.000	0.000	10:58:48

If you change the view of the display (zoom, rotate, etc.) then simply clicking on the entry in the table will draw the measurement line again. If you have multiples entries in the Measure table, then clicking on the required table row will show the corresponding measurement line.

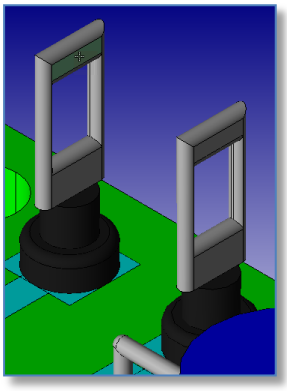
If we do not want to simply know the minimum distance between objects, but perhaps the distance between two specific points or surfaces on the components, then we can also use the Measure tool to do this.



5. Select the ***Level for shape*** icon (at the bottom of the graphicsl panel) and set the value to "Geometry"

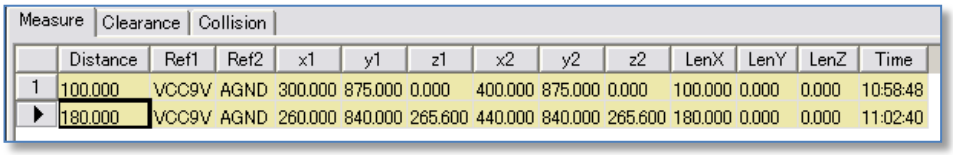


6. Now select the face shown, towards the top of the first testpoint component.



7. Rotate the display so that you can see the parallel face on the second testpoint and select the face (notice that it will highlight individually).

The distance between these two faces is measured and shown on the display and in an entry in the Measure table.



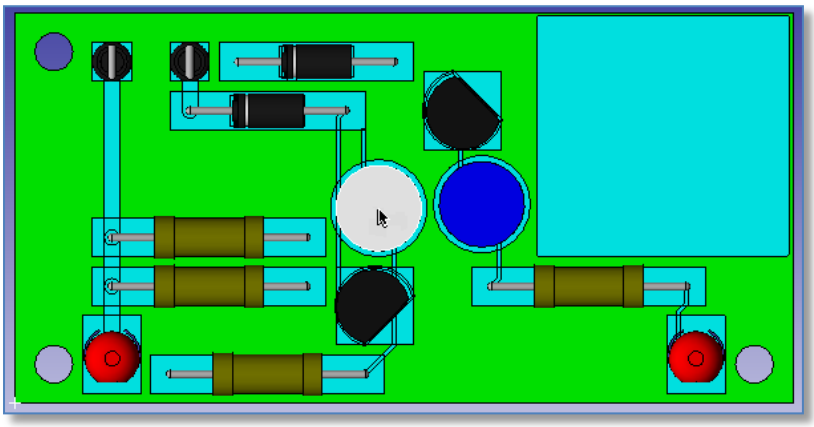
	Distance	Ref1	Ref2	x1	y1	z1	x2	y2	z2	LenX	LenY	LenZ	Time
1	100.000	VCC9V	AGND	300.000	875.000	0.000	400.000	875.000	0.000	100.000	0.000	0.000	10:58:48
▶	180.000	VCC9V	AGND	260.000	840.000	265.600	440.000	840.000	265.600	180.000	0.000	0.000	11:02:40

With the ***Level for shape*** option set to "Geometry", it is possible to measure distances between different points on the same component in addition to distances between components.

8. Continue to Experiment with the Measure tool to become familiar with the operation.

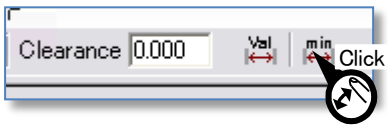
Let's now move on to look at the ***Clearance*** tool.

9. Switch to a **top** view of the board so it is easy to see the measurements.

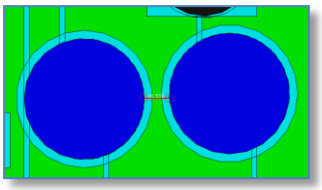


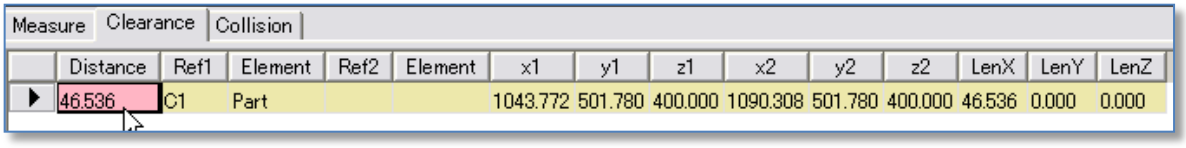
10. Select the Clearance icon from the 3D model toolbar and select component C1 in the design.

11. From the toolbar at the bottom of the graphics window, select the 'min' icon. This will measure the distance from C1 to the next nearest component – or in other words, the minimum clearance. In this case, that is component C2.



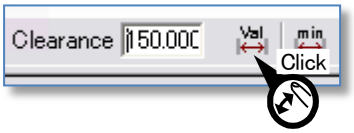
The distance is shown as before with the red measure line on the graphics display and an entry is added into the Clearance table.



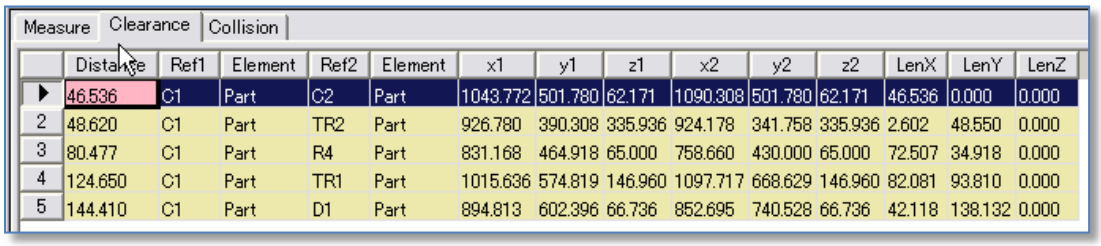


	Distance	Ref1	Element	Ref2	Element	x1	y1	z1	x2	y2	z2	LenX	LenY	LenZ
▶	46.536	C1	Part			1043.772	501.780	400.000	1090.308	501.780	400.000	46.536	0.000	0.000

12. Now, ensure that C1 is still selected, and enter a value of “150.0” into the Clearance field. This time select the ‘Val’ icon and BML will find all components (or other objects) which are closer than this specified clearance distance to component C1.



For each object identified, a row will be added to the Clearance table. In this example case there should be 5 rows in the table. Click on each row in turn to see the corresponding red measure line displayed on the design.



Measure | Clearance | Collision

	Distance	Ref1	Element	Ref2	Element	x1	y1	z1	x2	y2	z2	LenX	LenY	LenZ
▶	46.536	C1	Part	C2	Part	1043.772	501.780	62.171	1090.308	501.780	62.171	46.536	0.000	0.000
2	48.620	C1	Part	TR2	Part	926.780	390.308	335.936	924.178	341.758	335.936	2.602	48.550	0.000
3	80.477	C1	Part	R4	Part	831.168	464.918	65.000	758.660	430.000	65.000	72.507	34.918	0.000
4	124.650	C1	Part	TR1	Part	1015.636	574.819	146.960	1097.717	668.629	146.960	82.081	93.810	0.000
5	144.410	C1	Part	D1	Part	894.813	602.396	66.736	852.695	740.528	66.736	42.118	138.132	0.000

13. Continue to experiment the Measure and Clearance tools until you are comfortable with their operation.

14. Choose the Select icon from the 3D model toolbar, or press the Esc key on the keyboard to cancel the Measure or Clearance tool and switch back to the 3D isometric view.

End of Task

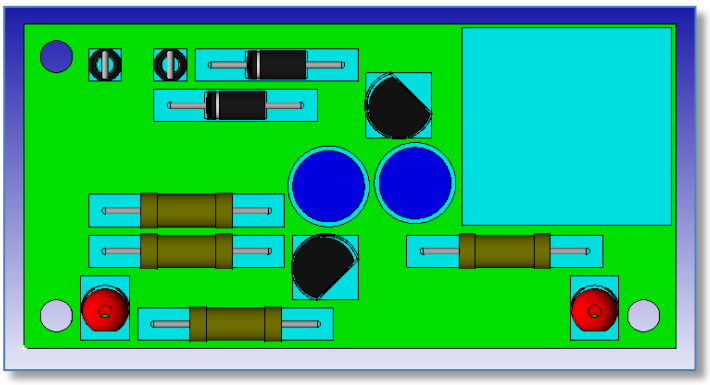
Task 10 - Replacing the Board Outline

Even after the PCB design has progressed to an advanced stage (e.g. placed and routed), it may still be found necessary to change the shape of the board (for example, to accommodate more components or to fit the PCB into a modified casing, etc.).

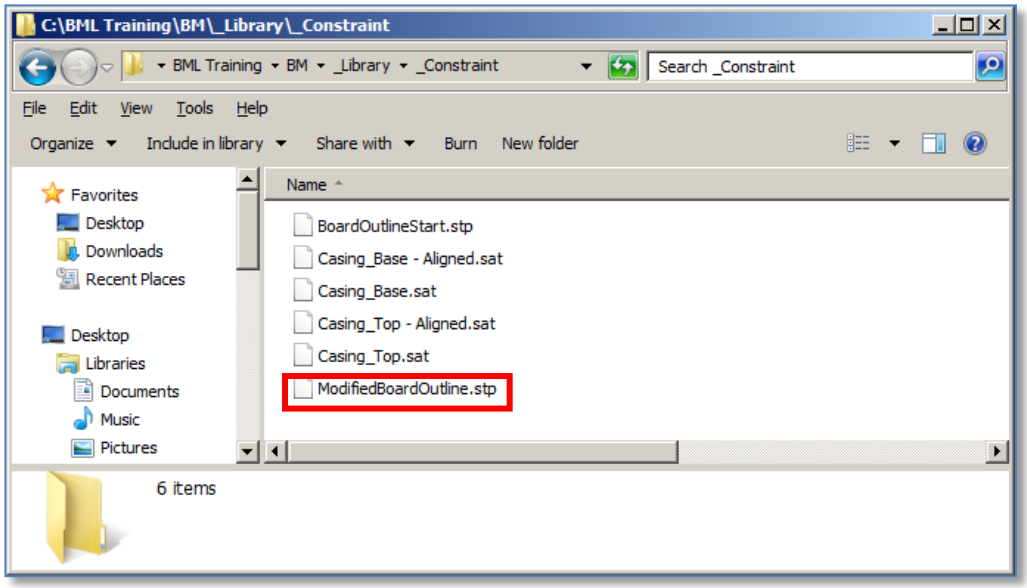
This can be achieved very simply within BML and the new board outline shape can be back-annotated to CADSTAR Design Editor to update the 2D design.

We can see how that is done in the following operations.

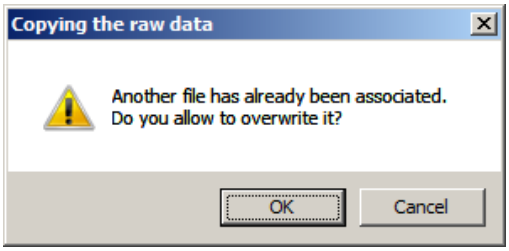
1. Adjust the display of the design to show it as a Top view.



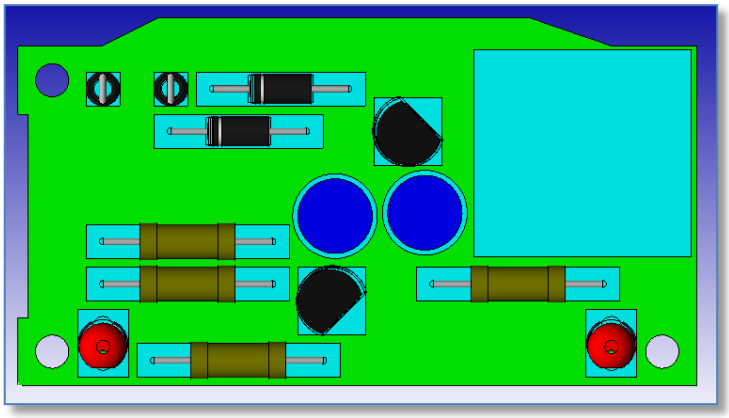
2. From the Main menu, select the PCB →Substrate →Associate board... option. In the file window, browse to the '**C:\BML Training\BM_Library_Constraint**' folder and select the file called '**ModifiedBoardOutline.stp'**.



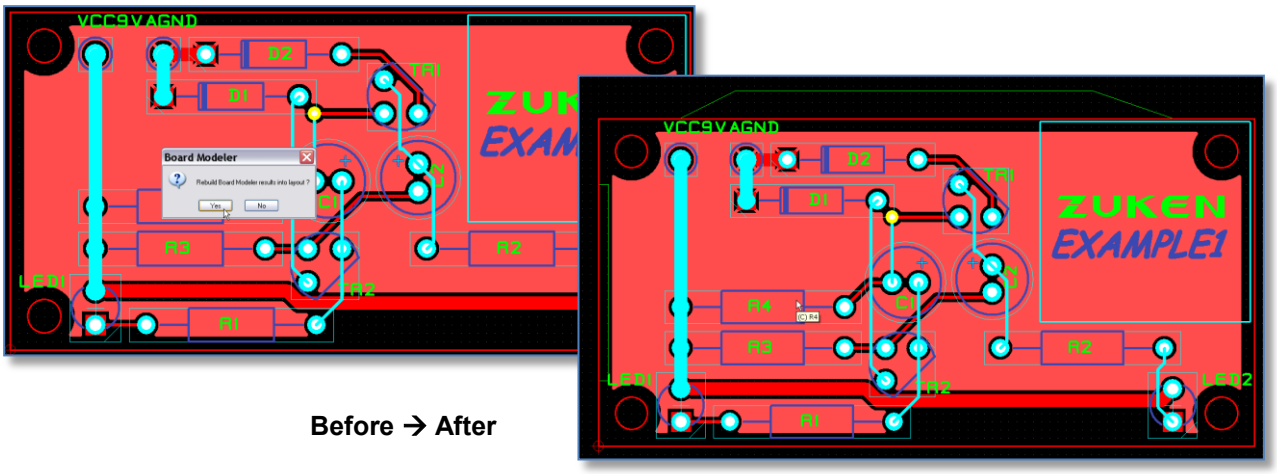
BML will warn that a board outline is already associated with the design and ask for confirmation that it should be replaced. Select ***OK*** from this dialog.



3. The new board shape will be imported and will replace the original one. The display should now look like this.



4. Save the project and exit from BML, returning to CADSTAR Design Editor. You will be prompted to import the design changes from BML. Select OK from the dialog. The modified board outline is imported into the PCB design.

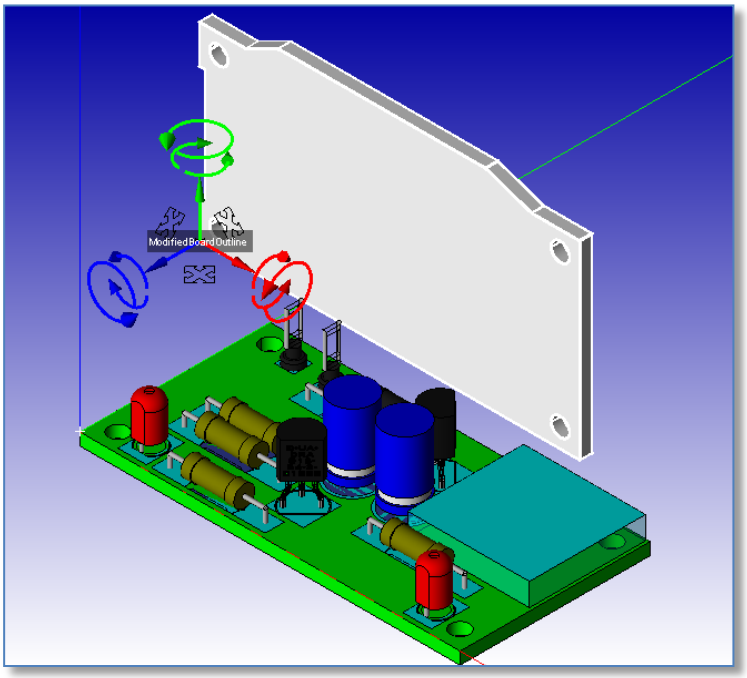


Before → After

Note: As simple as this task is, it is not always easy to re-align a new board shape. In most real world cases the board shape is coming from the master MCAD design where the origin is placed somewhere other than what was translated from CADSTAR. In this tutorial it is easy to visualize that the original Board shape we imported may been modified by the same user so we would not expect any extra effort.

Tip: Import the replacement board shape as a temporary constraint shape for initial inspection. If the New MCAD board shape does not align with the one imported from CADSTAR, it is recommended that it be returned to the MCAD user for adjustment.

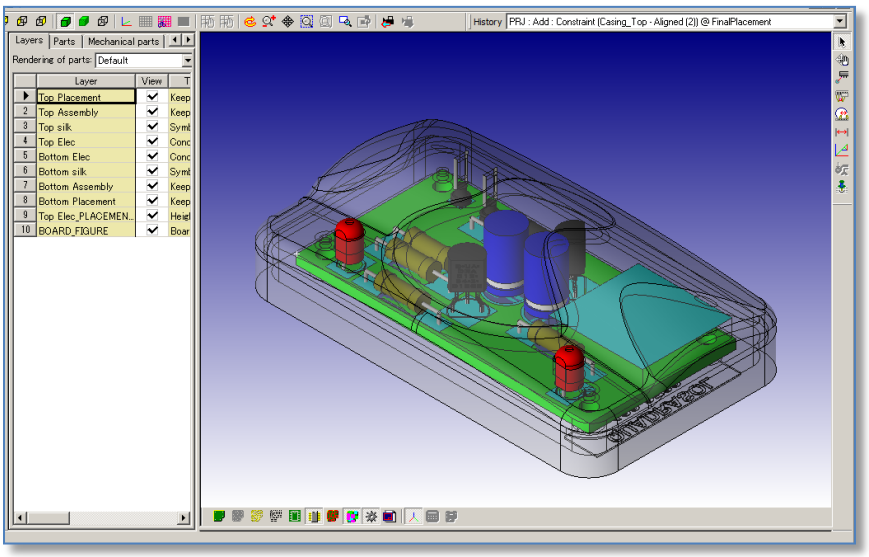
Tip: BML will allow you to create a 3D Library part where the new board shape can be imported and aligned in the desired orientation. Once aligned it can be exported as .STP file and then associated to replace the original Board shape.



End of Task

Adding Mechanical Enclosures

The ability to import a true 3D enclosure is the ultimate comparison using the PCB Designers placement scheme



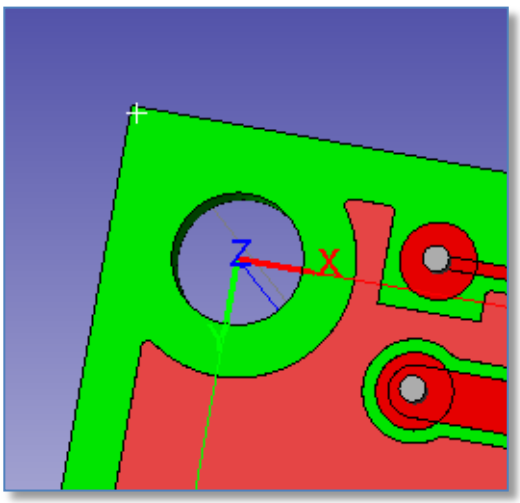
In this section we will go through the process of importing a mechanical enclosure (or 'constraint' in BML terminology) into the design, correctly positioning it relative to the electronic data (i.e. the PCB) and running collision checks to make sure the board and the components fit in the case correctly.

In an earlier section we used the on-line collision checking function which is provided as part of the ***Move*** command. That approach could be used just as well here, but to explore more of BML's functionality, we will use the batch ***Collision check*** option this time.

We will use the ***Align*** tool to locate the imported mechanical data in the correct position in relation to the PCB data.

Unfortunately, it is not common for PCB Design and MCAD design to have a coincident origin. This is one area in the world of advanced EM collaboration that is still evolving. However with BML we can stream line the process and still save a lot time.

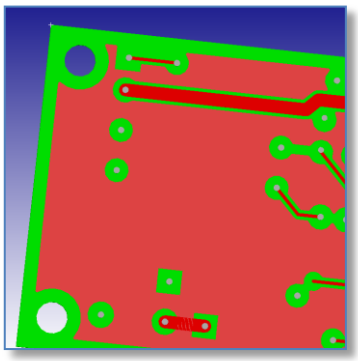
Importing an enclosure can be made easier by first placing an additional 'local' origin (a WCS or 'working co-ordinate system') on the BML PCB design. This is not strictly necessary in this case and we could achieve the alignment operation without the additional WCS, but in a more complex situation it is a method that may simplify things considerably.



Task 11 – Adding Mechanical Enclosures

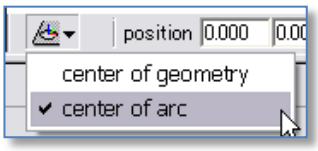
Before we actually import the enclosure, we are going to add an additional, 'local' origin to the board, which will help us in the alignment process. Make sure that the 'Board' and 'WCS' selection filters are active before we start.

1. Identify the hole in the board nearest to component LED1 (in the bottom left corner when looking from the top of the board). Flip the board over so that you are looking at this hole in the bottom of the board.

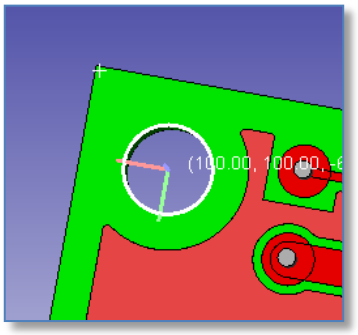


2. Either select the ***Add WCS*** icon from the 3D Model Toolbar or select the ***PCB→Add WCS*** menu option.

3. From the icon at the bottom left of the graphics panel make sure that the option is set to 'center of geometry' and then select the edge of the hole on the bottom surface of the board.



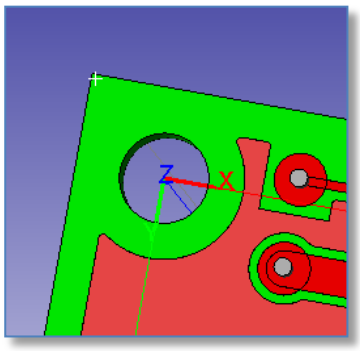
A pale-coloured orgin marker will be drawn at the position where the WCS will be added. In this case, it should be at the centre of the hole. However, the z-axis in pointing in the opposite direction to the one we want.



4. Select the icon at the bottom of the graphics panel to reverse the direction of the z-axis.

5. Complete the operation by confirming that the WCS should be added at this position by selecting the Do operation icon (at the bottom of the graphics panel).

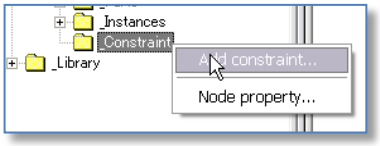
The WCS will then be drawn in the normal, solid red, blue and green colours to show that it has been successfully added.



6. You can now flip the PCB back again so that it is the correct way up. Choose the ***Select*** icon from the 3D model toolbar or press the ***'Esc'*** key to cancel the ***Add WCS*** command.

Now we can add the mechanical case to the design and align it in the correct position.

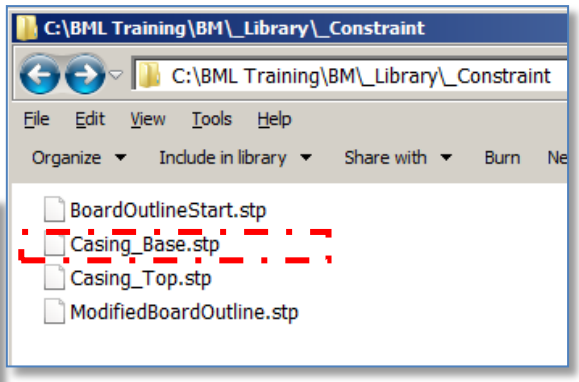
7. Either select the ***PCB / Constraint / Add constraint...*** menu option or right-click on the '_Constraint' folder in the Project Browser and select ***Add constraint***....

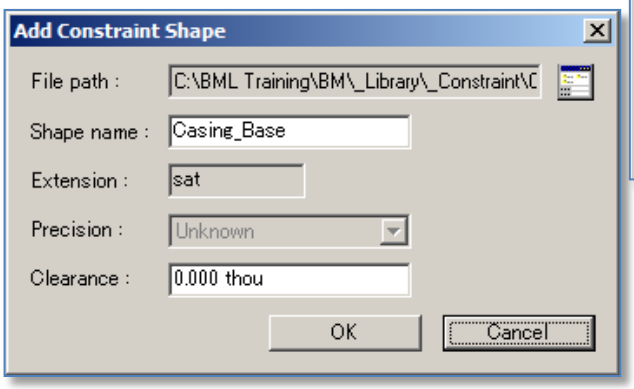


8. Cpen the file browser in the **Add Constraint Shape** dialog, navigate to the 'C:\BML Training\BM_Library_Constraint' folder and select '**Casing_Base.stp**'.

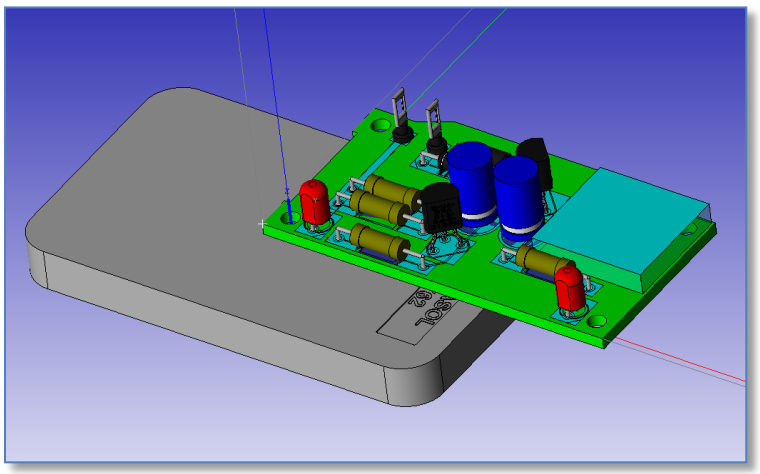
   BML will then convert this STEP format file into a, ACIS '.sat' file ('*Casing_Base.sat*') and the dialog will remain displayed allowing you to confirm that this file should be imported.

   Select ***OK*** from the dialog.

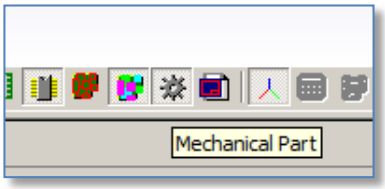




   The display should now look similar to this (zoom out a little if necessary).

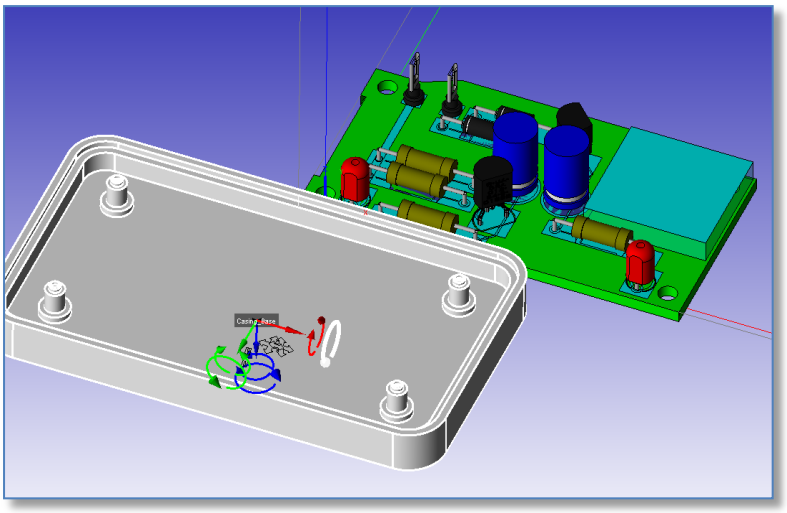


9. Select the tool bar button for **Mechanical part**.

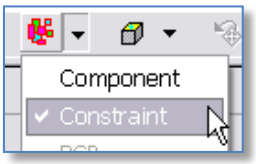


10. Use the Move function to flip the part 180 degrees and slightly away from the PCB data as shown.

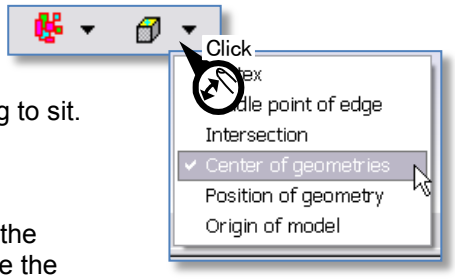
    Hint:use the manipulator and click the red 180 arrow.



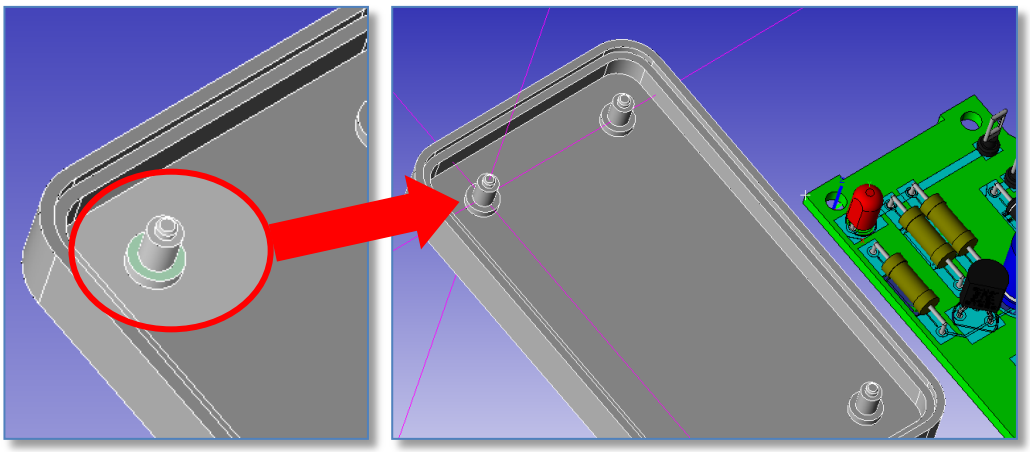
11. Select the Align tool from the 3D model toolbar and ensure that the Element kind option (at the bottom of the graphics panel) is set to “Constraint”.



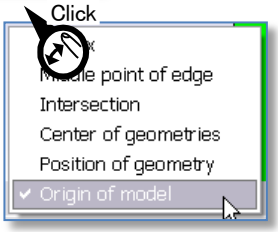
12. Set the ***Coordinate system*** option to be ‘*Center of geometries*’ and rotate the design so that you can see inside the case and can see the locating pins where the board is going to sit.



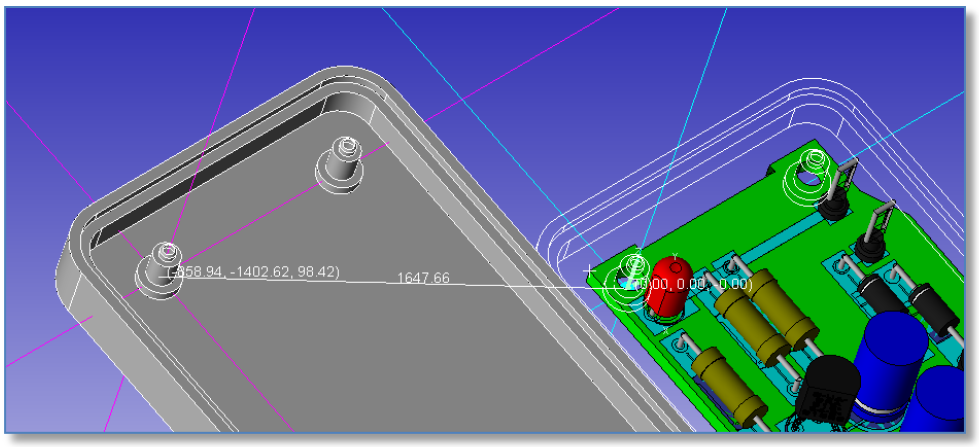
13. Click on the case once to select it and then click on the face of the locating pin that the bottom of the board will locate against. See the imagea below to make sure you get the correct surface.



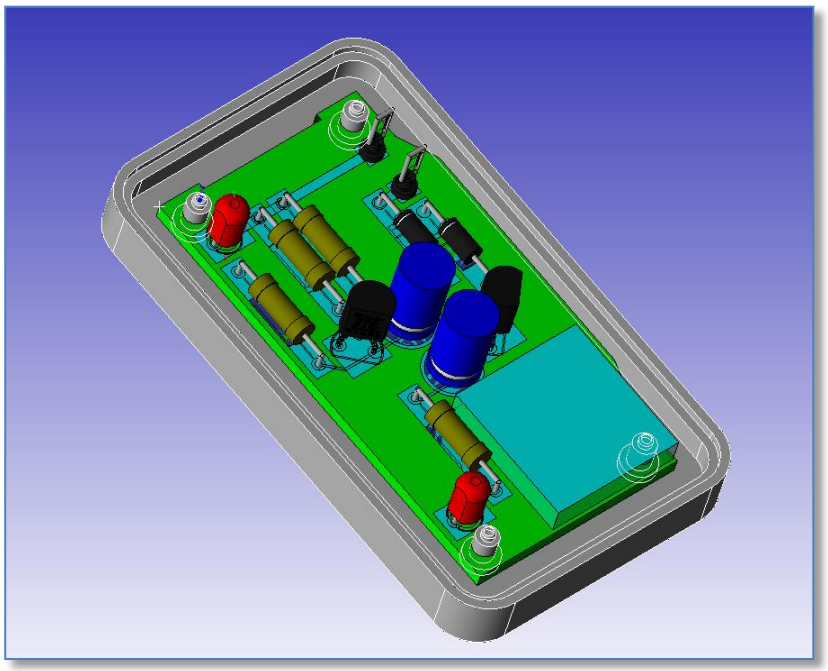
14. Now change the Coordinate system option to ‘Origin of model’ and select the WCS orgin, in the middle of the locating hole in the board, that we added earlier.



A white ‘ghost’ image will now be shown to indicate where the case will be moved to and a straight line is drawn joining the origin point selected on the case with the WCS origin selected – i.e. where the select point on the case will end up after the Align operation is complete.

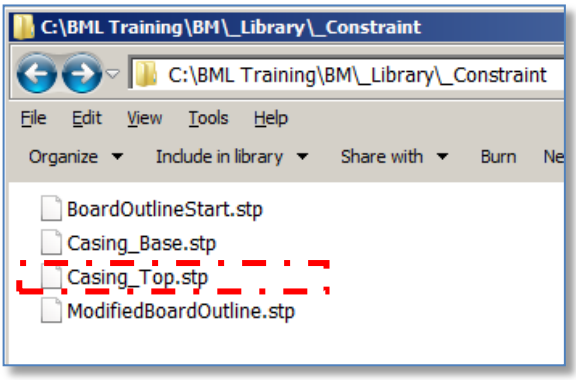


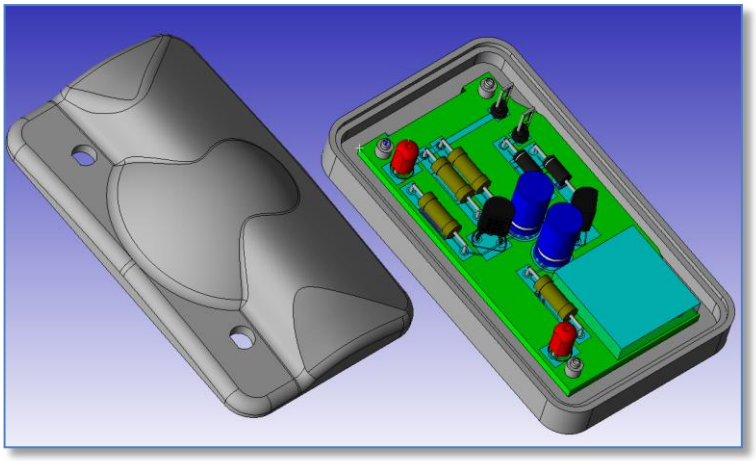
15. Select the **Do operation**  icon (at the bottom of the graphics panel) to confirm and execute the operation.



16. You can now try to add the top of the case into the design yourself. The data can be found in the file called ‘Casing_Top.stp’ which is located in the ‘C:\BML Training\BM_Library_Constraint’ folder.

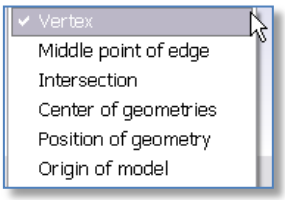
As a hint, set the ***Coordinate system*** option to ‘Vertex’ and select appropriate corners on the top and bottom of the case to align them together.





Casing_Top added as a constraint and moved to avoid 3D overlapping.

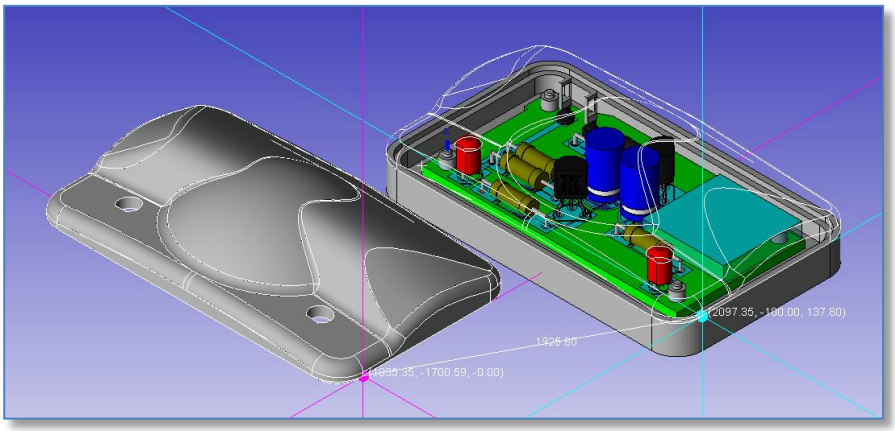
17. Select the ***Align*** tool from the 3D model toolbar and ensure that the ***Element kind*** option (at the bottom of the graphics panel) is set to "***Constraint***".

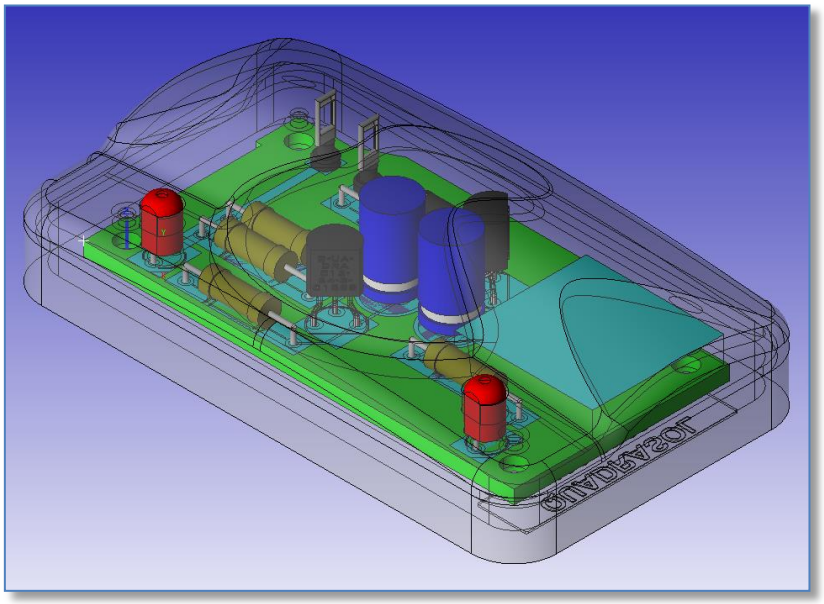


18. Set the ***Coordinate system*** option to be '*Vertex*' and click on the case lid once to select it and then select the corner as shown (a small purple spot will appear over the corner when the cursor is in the correct postion).

19. Select the corresponding corner on the bottom of the case (a small blue dot will be shown) and confirm the alignment with the **Do operation** icon.

    The top of the case will be positioned correctly on top of the bottom section





Casing_Top aligned with Casing_Base with view of constraints set to transparent.

Reminder – You can use <CTRL>-+ <Left, Middle and Right> mouse buttons when performing *Align* or measurement functions to zoom, pan, orbit

20. At this point it would be a good idea to save the project (File→Save→Project). Once the project has been saved with the imported mechanical data in place, then it will remain part of the project even if the design is modified within CADSTAR Design Editor and passed forward again to BML.

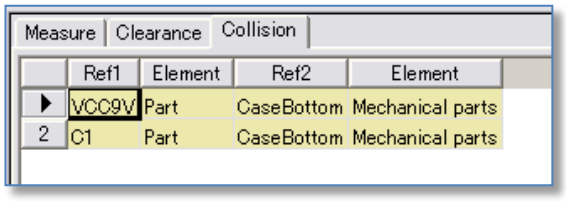
    However, for the operations we are now going to perform in the tutorial, the top of the case will only get in the way. So, either hide the top, by selecting it and from the ***View*** option on the right-mouse pop-up menu choose '*No display',* or open the '*_Constraint'* folder in the Project Browser, right-click the '*Case Top'* entry (the one with the 'gearwheel' icon) and select ***Delete*** from the pop-up menu.

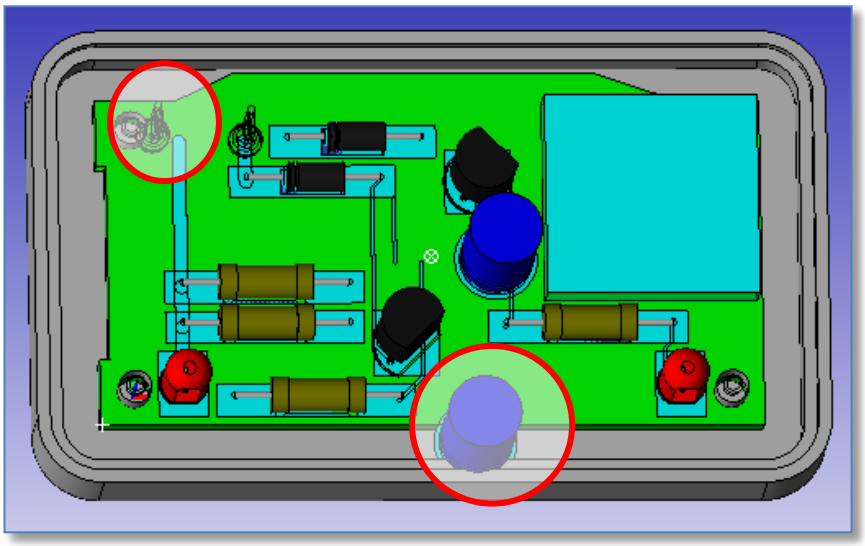
End of Task

Task 12 - Running Batch Collision Checks

So that we can look at the operation of the batch collision check functions, for the purposes of the tutorial, let's start by causing some deliberate errors between the components on the board and the mechanical case.

1. Move component C1 so that it overlaps into the side of the case and move the VCC9V Solder terminal so that it clashes with its nearest support pin on the enclosure base. This should result in a layout similar to the following (the errors that have been created are circled in red).



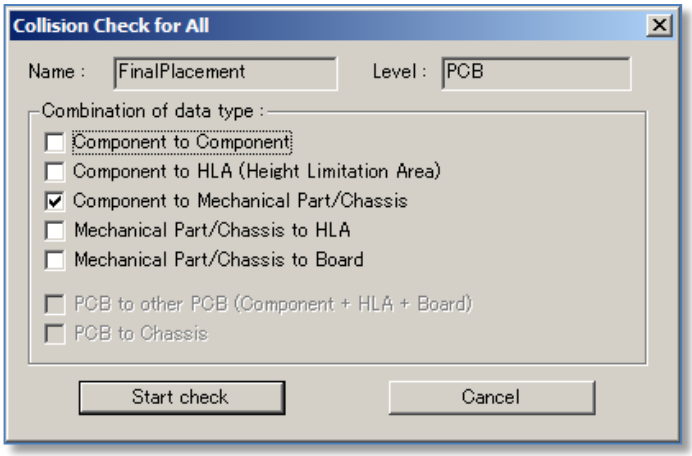


Now we can see if BML will detect these problems.

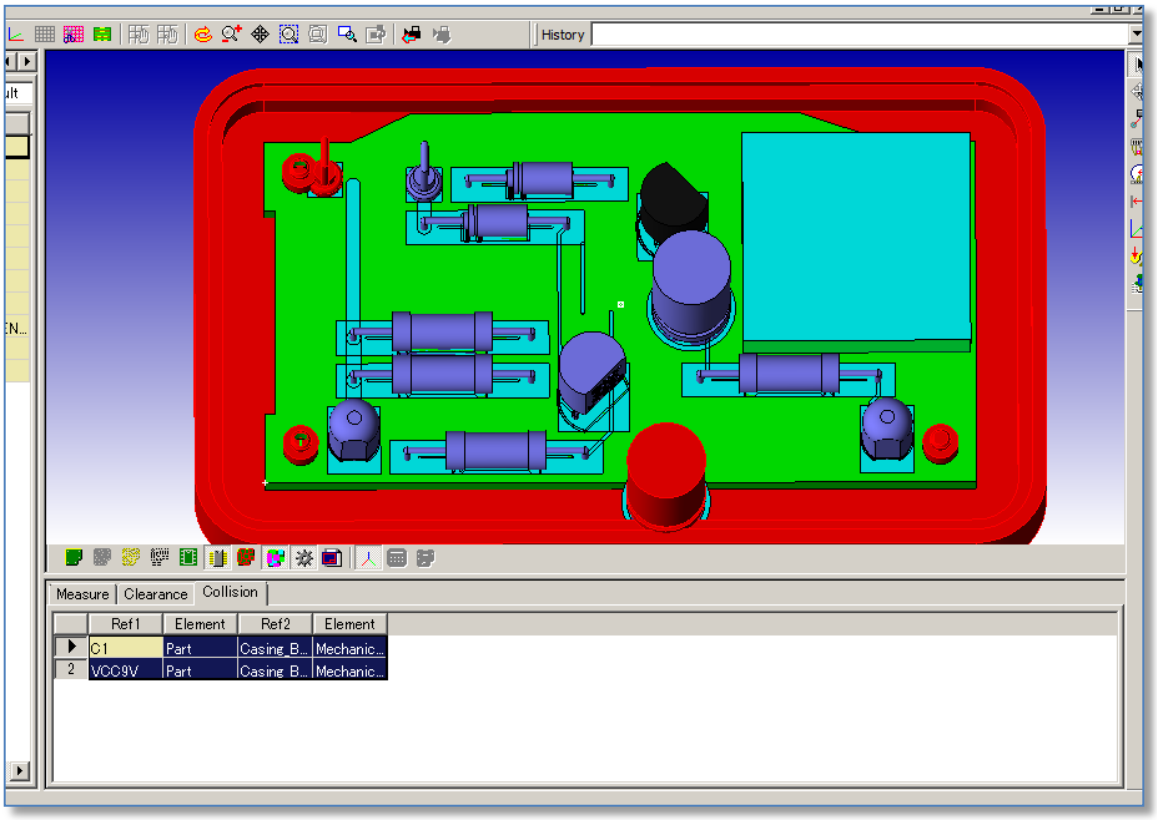
2. Click the Select icon from the 3D model toolbar to cancel any existing command.

3. Select PCB→Collision check→All... from the menu and set the 'Collision Check for All' dialog options as shown.

   For now we are just going to look for collisions between the components and the casing.



4. Select the [Start check] button. All the items which are in error (the two components and the case itself) will be highlighted in red on the display and there will be an entry added to the Collision table for each individual error.



The bottom casing, C1 and VCC5V are in error. The LED models are red by design, they are not in error.

It is possible to combine all or any subset of the checks at the same time, as required. You can also use the ***Measure*** and ***Clearance*** tools to check distances between the components or board and the mechanical case.

The batch collision check can also be limited to selected objects – rather than the whole design.

5. Simply select the objects which you wish to check and use the PCB / Collision check / Selected Data command.

6. Having tried out the commands, move the components back to their original positions to clear any collisions.

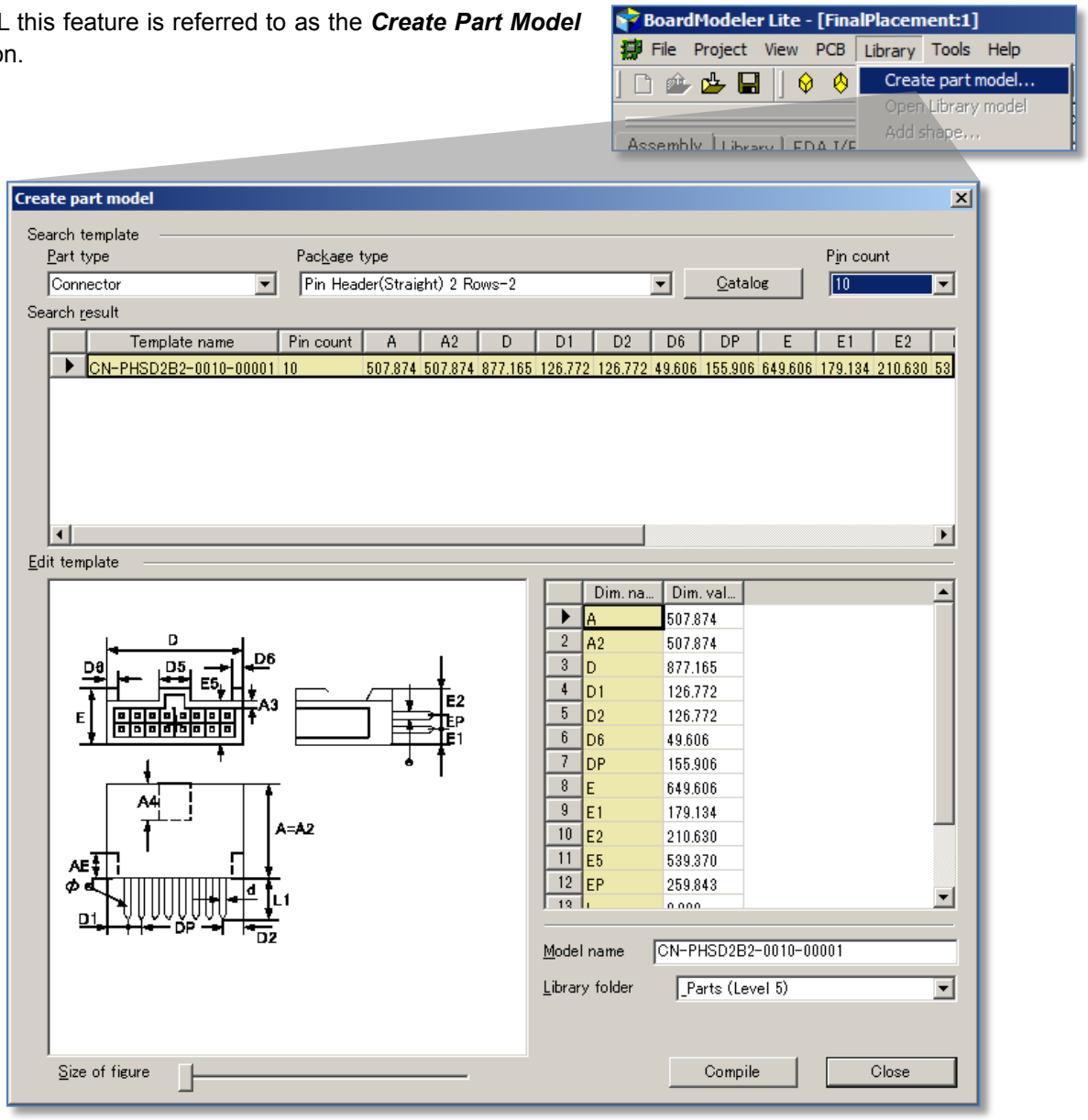
End of Task

Creating Detailed 3D Part Models

Acquiring 3D part models can be very time consuming, especially when electronic parts manufactures do not provide 3D models for download via their web sites. However it is worth the search and even to contact the Vendor for the actually 3D CAD model.

In the event of a failed WEB search, BoardModeler Lite provides a 3D component wizard which allows detailed 3D representations of the components in your designs to be created quickly and easily – there is no need to rely on a separate M-CAD department to supply this data. In fact this application can be used by the M-CAD department to help with the critical parts selection process.

In BML this feature is referred to as the ***Create Part Model*** function.



Note: This is a user selectable option during the installation of BoardModeler Lite and must be installed for this section.

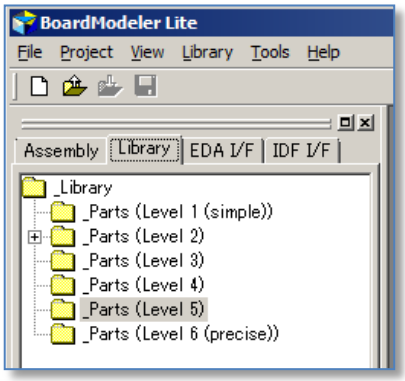
Task 13 - Creating Detailed 3D Part Models

In this task you will learn how to use the **Create Part Model** function to generate detailed 3D models and we will also consider the additional steps you may need to perform to prepare the models for use with your CADSTAR designs inside BML.

This operation is part of the library management for BML and as such can be performed without any other project data being loaded in the application.

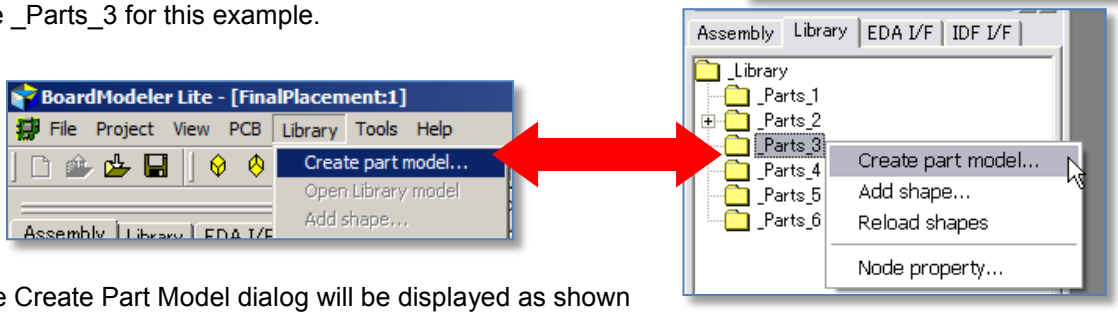
So to begin this section, if you already have BML running close any open projects (***File / Close / Project***)

1. Start BML from the Windows ***Start*** menu or desktop icon

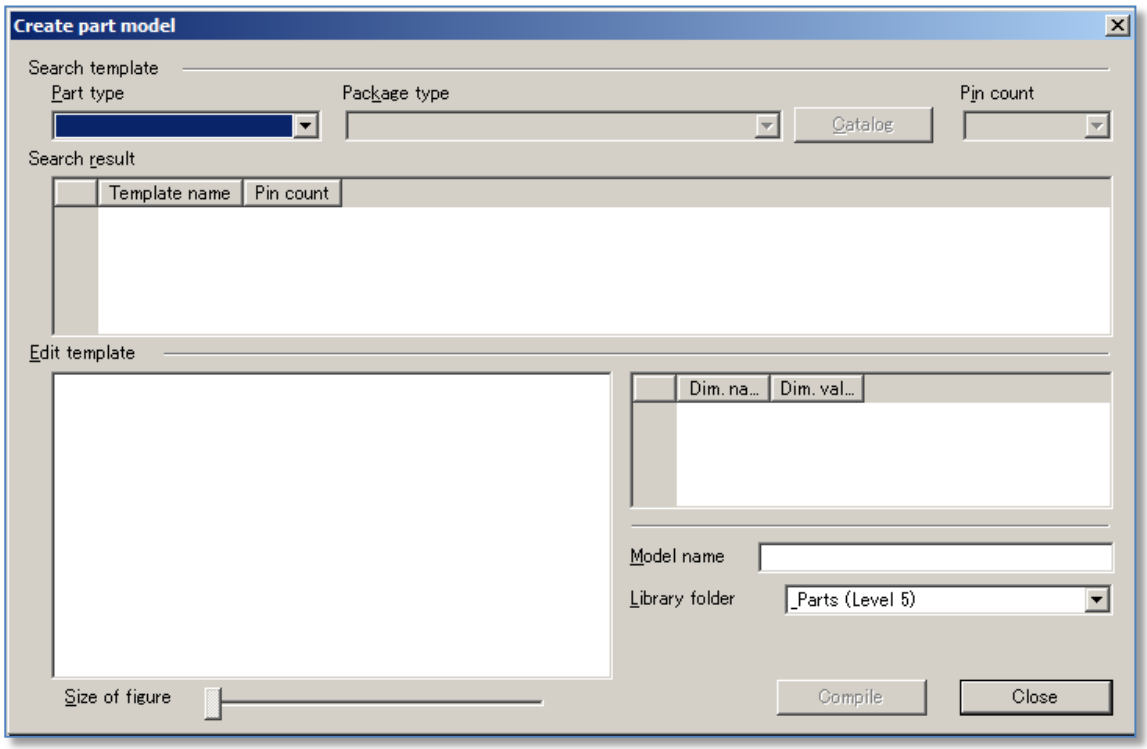


2. In the project browser, switch to the ***Library*** tab. You should see a display similar to that shown →

3. Start the ***Create Part Model*** function either by selecting the ***Library / Create part model…*** menu option or by selecting the ***Create part model…*** option from the RMB menu on one of the six library folders (_Parts_1 to _Parts_6). We have chosen to use _Parts_3 for this example.



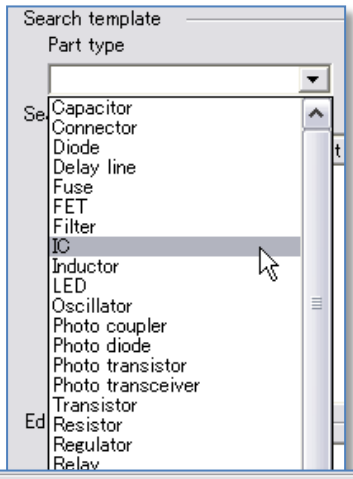
4. The Create Part Model dialog will be displayed as shown .



5. First select the drop-down list in the top left of the dialog labelled Part type. This allows us to select the type of part we wish to create a detailed model for (e.g. capacitor, resistor, IC, diode, etc., etc.).

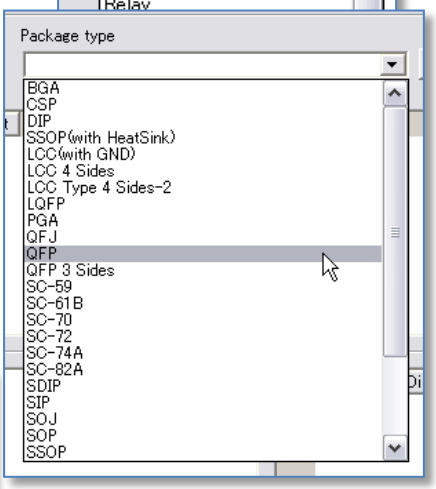
   For this example, select 'IC' from the list.

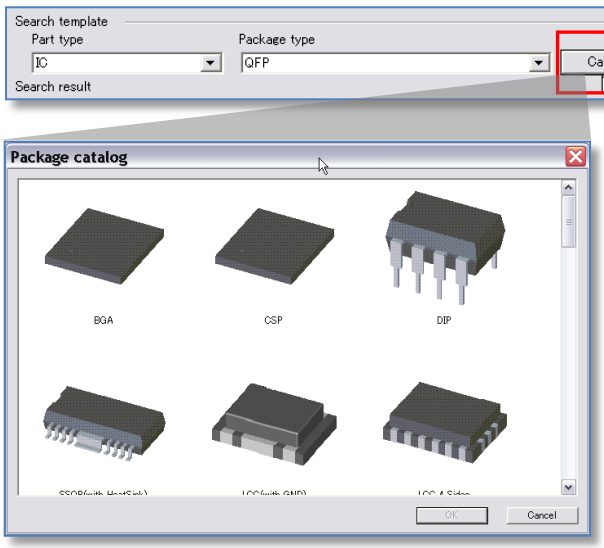
   Having selected the part type, we can now go on and specify the particular package type that we require.

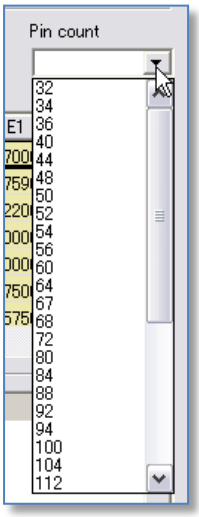


6. Select the Package type drop-down list and select 'QFP' from the list of package types.

   As an alternative to choosing the package type from this list, select the Catalog button and choose the required package type from the images shown.

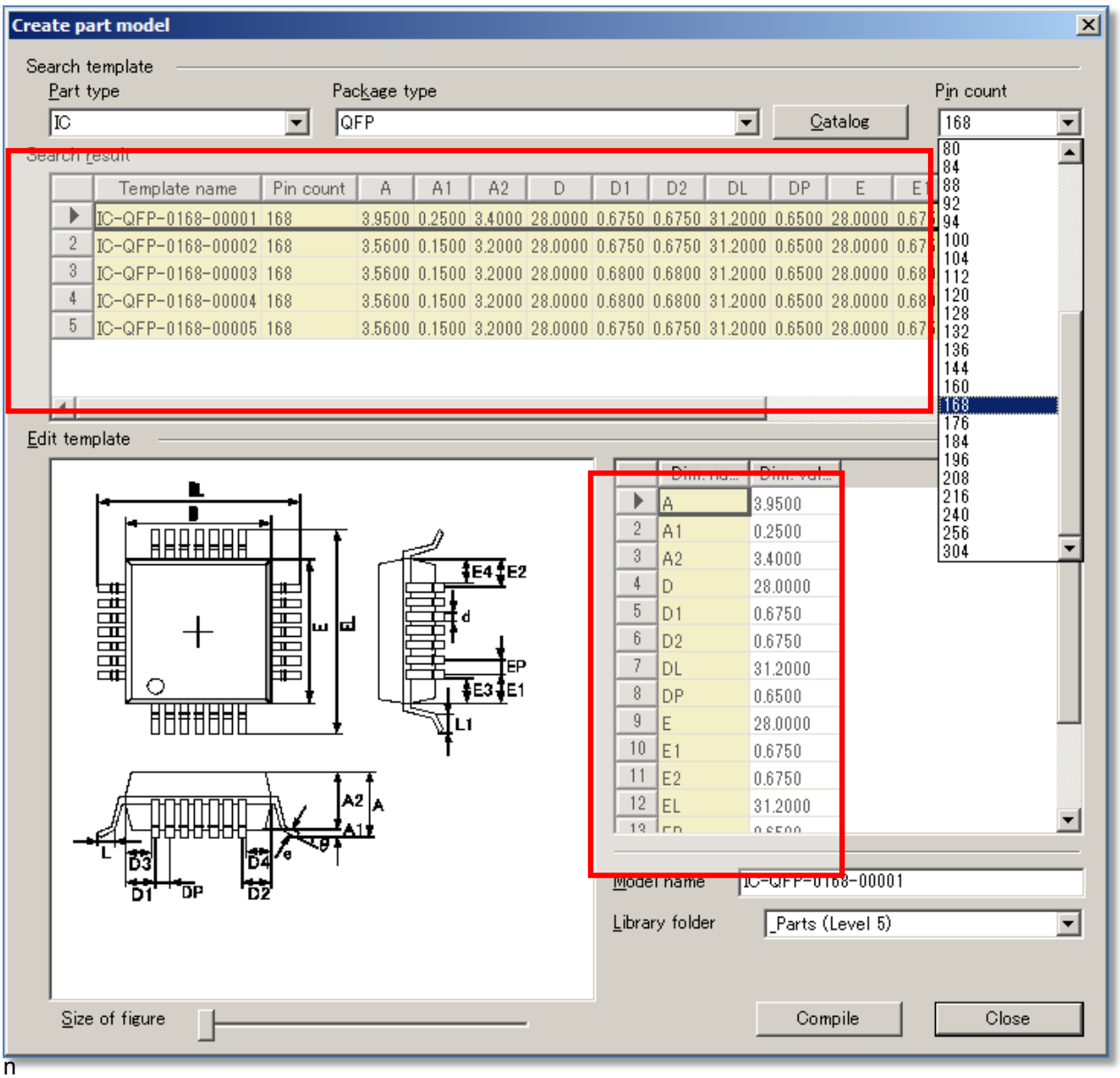






It is possible to filter the search results further by selecting a specific number of pins from the Pin count option.

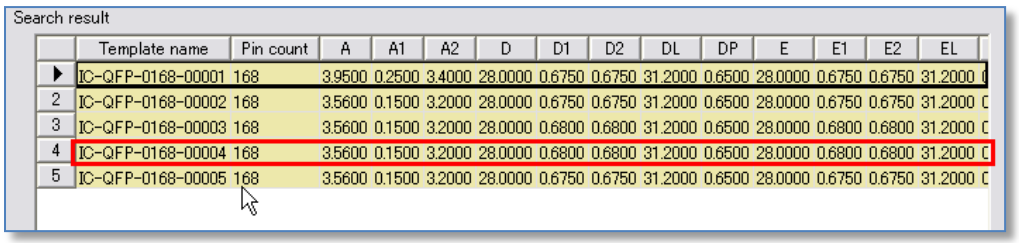
Whichever method is used, after making a selection the remainder of the dialog will be populated with results of the available templates for the selected part and package types. For example, if we select '168' from the Pin Count list, the number of different options presented in the table will be limited to only 5 entries.



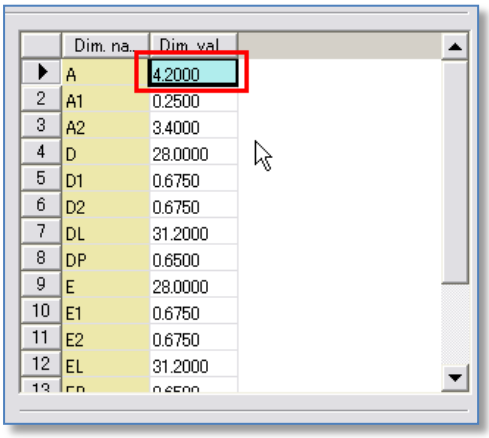
n

The search results presented correspond to the number of standard templates for the selected package that the ***Create Part Model*** wizard has defined.

The columns in the dimension associations table (with headings A, A1, A2, D etc.) correspond to the values for the specific dimensions shown in the graphics window in the bottom left of the dialog. If one of the rows in this tables gives the correct sizes required for the package, then you can simply select the desired row.

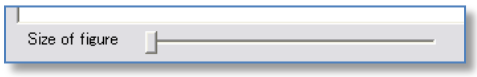


However, if none of these standard sizes matches your requirement, it is possible to specify custom sizes for the dimensions in the panel at the bottom right of the dialog.



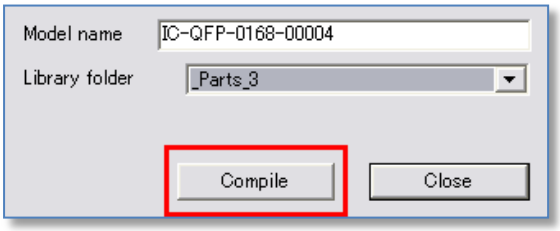
Simply enter the required sizes in the table. Note that you will need to make sure all the dimensions for the package are consistent and do not conflict with each other.

If you need to enlarge the image in the graphics panel to see all the dimensions clearly, use the '***Size of figure***' slider at the bottom of the window.

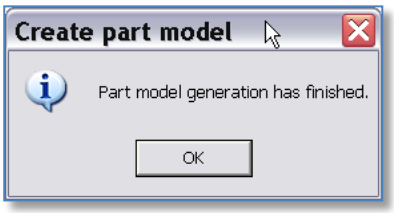


7. For the purpose of this example, select one of the templates from the Search result table.

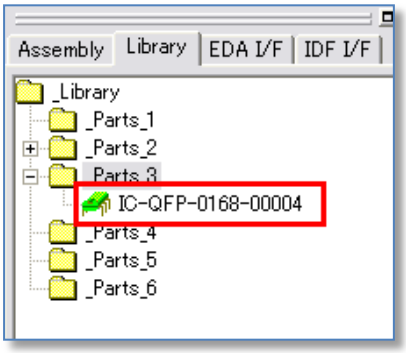
8. Accept the default model name and libray folder name shown in the dialog and select the Compile button.



BML will compile the 3D model for the selected package, using any customized dimension values and pin count, and confirm successful completion.

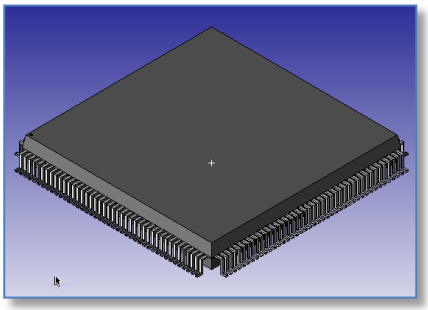


9. Exit the Create part model dialog with the Close button and open the relevant folder in the Library tab of the Project Browser. You will see that the 3D model has been created and added to the library.



10. Select the new model in the _Parts_3 folder in the Library tab and from the RMB menu select the Open Library model option. The new model will be opened and displayed in the Library Model editor graphics window.

    As you can see, it is very easy to quickly create complex 3D models and add them to the BML library!



11. Finally, select **File/Close/Library** model from the main menu to exit from the 3D Library model editor.

End of Task

Task 14 - Modifying Detailed 3D Part Models

In This task you will repeat the steps of task 12 to explore the Create Part Model wizard in more detail and try generating detailed 3D models for different types of component packages.

Now that we know how to create detailed 3D models, we can look at the extra steps we may need to perform to make those models ready for use in BML to represent the components in your CADSTAR designs.

In order for the models to be useful, we need to make sure that the position of the origin and the orientation of the component match between the definition of the 2D footprint in the CADSTAR library and the definition of the 3D model in the BML library.

So, let's create an alternative representation for one of the components in our tutorial design and see how to set up the required configuration (the 1N914 part).

By referencing the datasheet information below for this part and the existing CADSTAR PCB design, we can determine that we need to create a model with the following basic dimensions:

Body diameter	**=**	**80 thou**
Body length	**=**	**200 thou**
Pin pitch	**=**	**400 thou**
Lead diameter	**=**	**20 thou**

1. Start off by making sure that you have BML running with the BasicOperations project loaded (transfer the design from CADSTAR to BML if necessary).

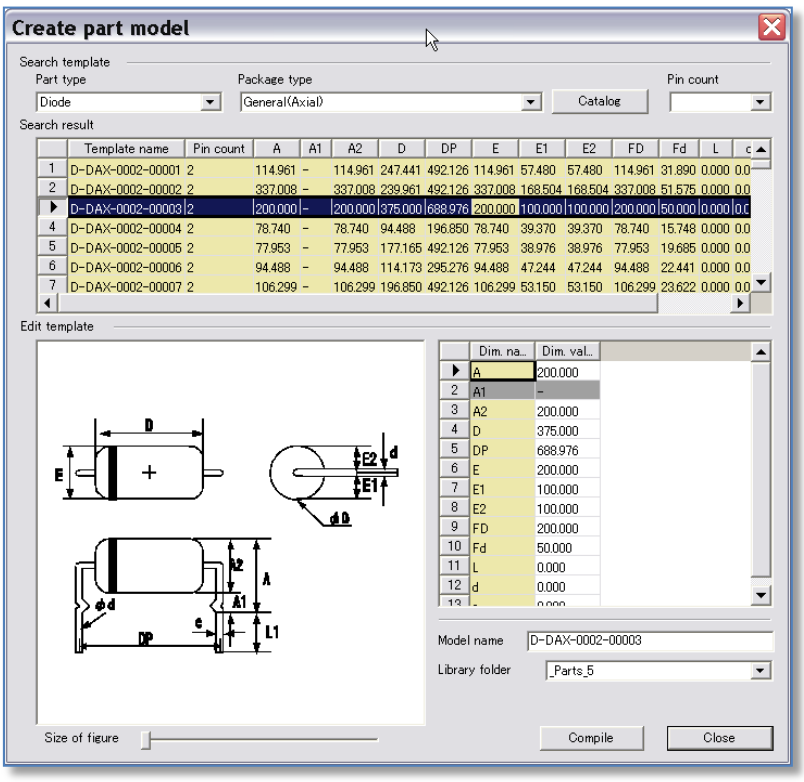
2. Switch to the Library tab in the Project Browser in BML, select the _Parts_5 folder and from the RMB menu select the Create Part model… menu option to launch the wizard as before.

3. Select a part type of 'Diode' and a package type of 'General(Axial)'.

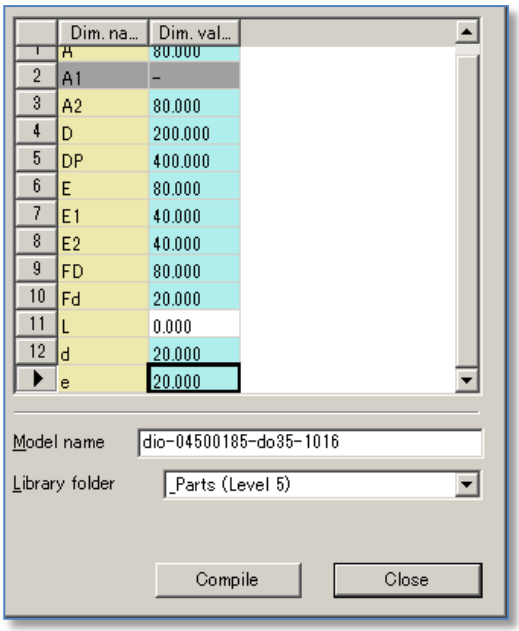
   The Search result table will be populated with a range of possible package templates.

   Since there is no template matching the specification, we can select a model template where the diode body diameter is 200 thou.

   We can quickly see that the rest of the dimensions do not exactly match the sizes we want to use, so we need to edit the template and create a custom package.

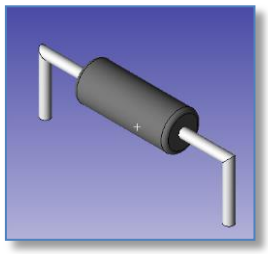


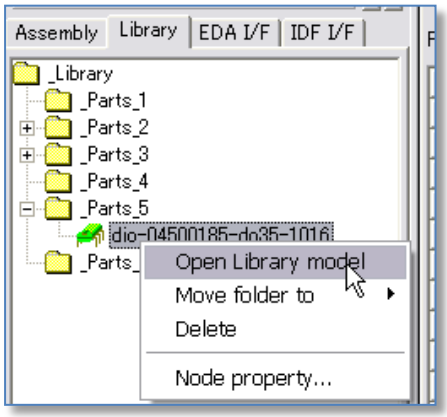
4. Enter the custom dimensions in the dialog so that they match the values shown in the adjacent image. →



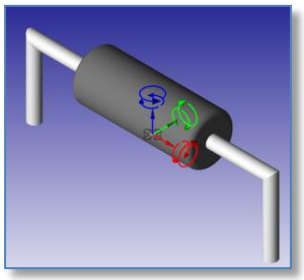
5. Set the ***Model name*** and ***Library folder*** so that they are also as shown in the image, then select the ***Compile*** option.

6. In the same way as before, open the newly created 3D model in the 3D Library editor window.
The result should be similar to that shown below.





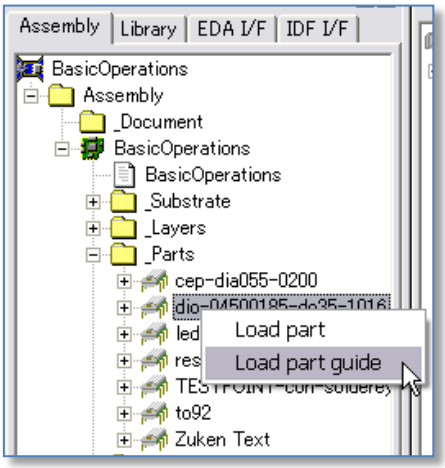
7. Now select the **Move origin**  icon from the 3D model toolbar to see where the origin is currently positioned. As you can see, it is in the centre of the component body, not at pin 1 where we need it to be.



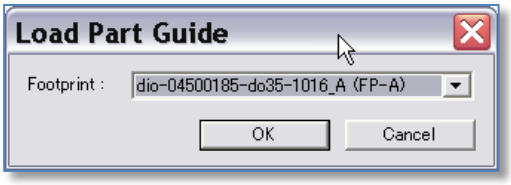
We now know where to align the origin to make it match with the 2D footprint origin defined in the CADSTAR library.

To help with this alignment we will superimpose a 'part guide' onto the 3D model window.

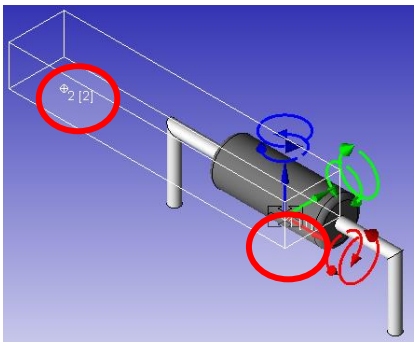
8. Switch to the Assembly tab in the Project Browser. Make sure the _Parts folder is expanded and select the diode (dio-04500185-do35-1016) from the list.



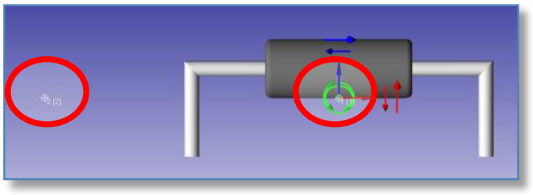
9. Now, from the RMB menu select Load part guide. Select OK to accept the default footprint from the next dialog that is displayed.



Notice that two numbered, origin symbols have been added to the display, marking the position of the component terminal positions as defined in the CADSTAR footprint. →



10. Use the Front display icon to switch the view to a 2D view of the front of the component. →

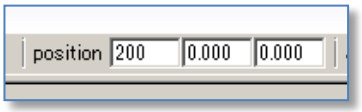


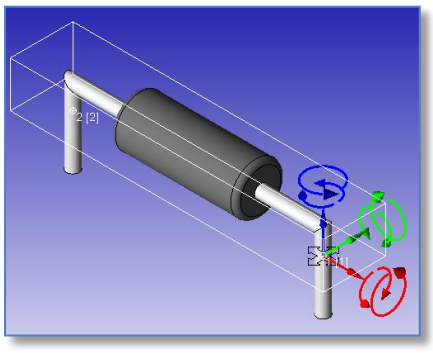
11. Select the **Move origin**  icon again so that we can re-position the origin in the correct location.

Since we know that the pin-pitch of the component in our design is 400 thou and the orign is currently positioned at the centre of the component, we know we need to move the origin by 200 thou in the x-axis direction.

We could simply repositon the origin by dragging on the red arrow in the Move 'manipulator', but to position it in exactly the correct location, the easiest way is to type the required shift into the number field at the bottom of the graphics window.

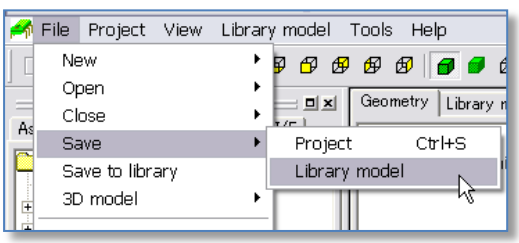
12. Enter '200' in x-axis box and press the Enter key. The display will be updated to show the new origin location.



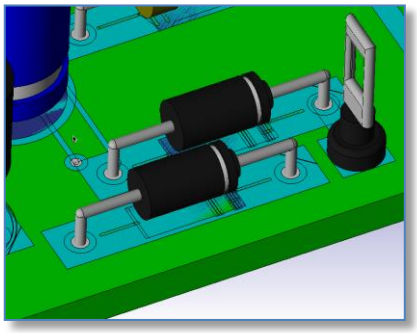


(Isometric view shown)

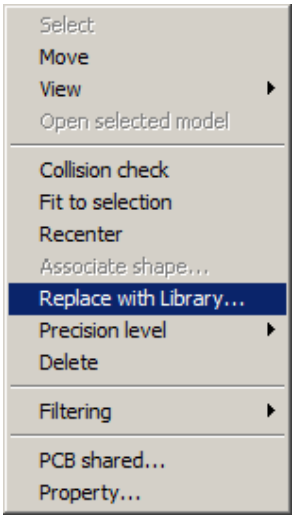
13. Save the modified 3D model (File / Save / Library model) and exit from the 3D Library editor back to the main project window.



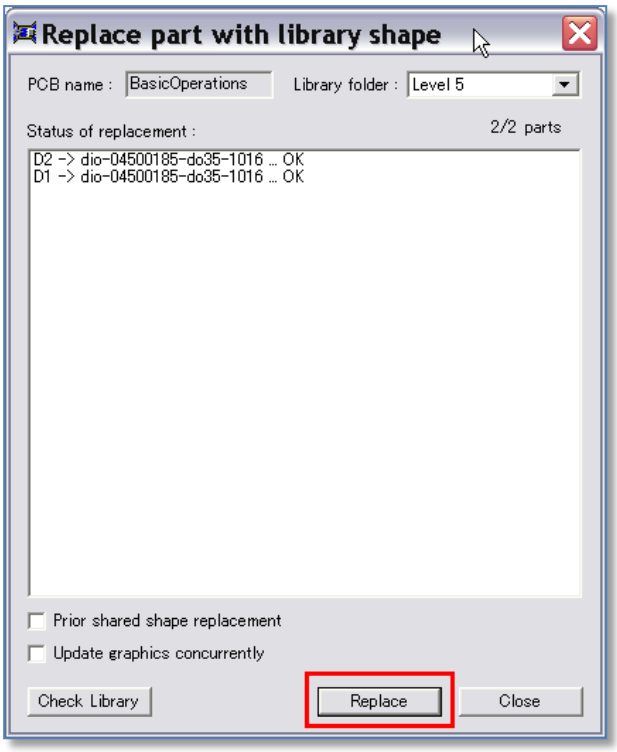
14. Adjust the graphics display so that you can see the two diodes (D1, D2) in the design.



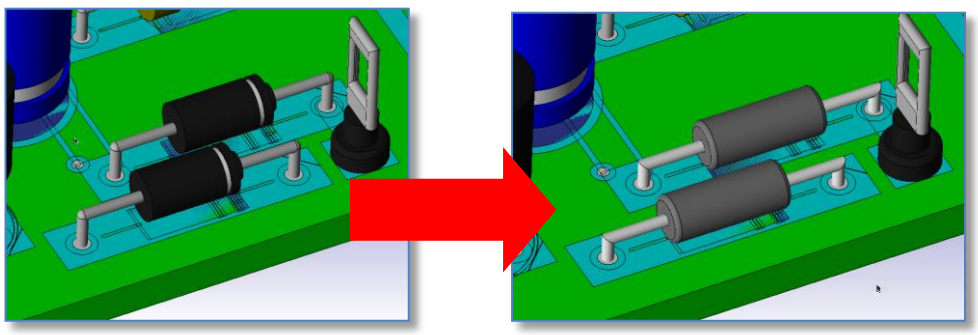
15. Select both diodes and from the RMB menu select the 'Replace with library…' option.



16. The *Replace part with library shape* dialog will be displayed. Set the Library folder to 'Level 5' and select the Replace button and then Close the dialog



The original models have now been replaced with the new ones we have created.



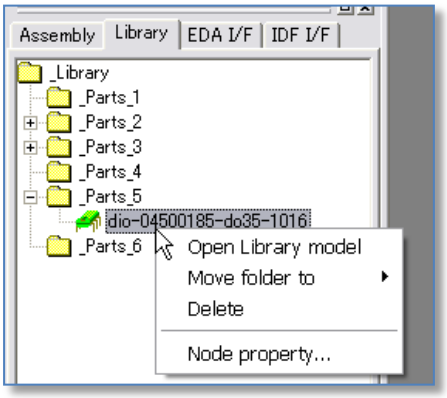
You should now be able to create suitable 3D models to match the components used in your own designs and use them to replace the simple, extruded models generated by BML.

End of Task

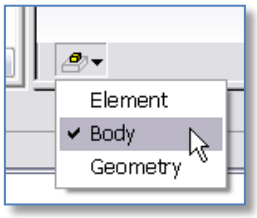
Task 15 - Setting colours in 3D models

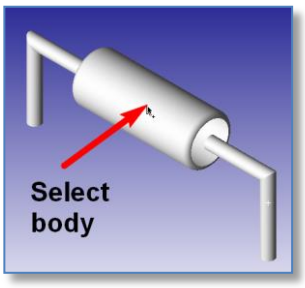
After creating 3D models using the Create Part Model wizard, it is possible to change the colours which are used in the models. The following steps demonstrate how to achieve this.

1. Open the model of the diode we created earlier in the tutorial into the Library Model editor.

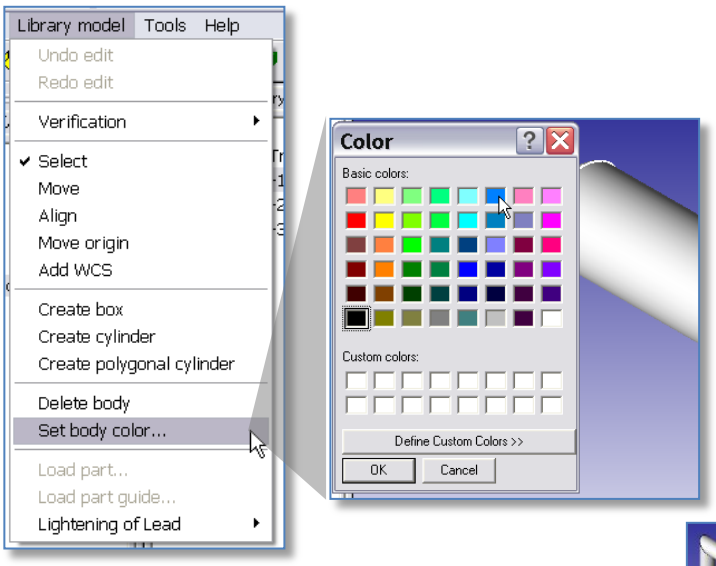


2. Make sure that the ***Level for shape*** option (at the bottom of the graphics window) is set to '*Body'* and then click on the part of the model for which you would like to set the colour.

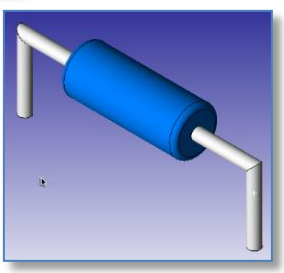




3. From the ***Library model*** menu, select the ***Set body color…*** option. This will open a colour picker dialog which will allow you to select the desired colour for the body.



4. Repeat the process as required to change the colour(s) of other parts of the model, before saving it (***File / Save / Library model***) back to the library.



This completes the interactive exercises which comprise the BoardModeler Lite tutorial.

Appendix

Preparing CADSTAR designs for use with BoardModeler Lite

Please note: *The remaining sections in this document are not intended to be part of the tutorial (i.e. you do not need to follow any further steps or complete tasks). The information is presented as a reference to assist in your future use of BML, when setting up your own designs for transfer from CADSTAR to 3D.*

For a CADSTAR PCB design to be used successfully with BML, to obtain a meaningful representation in 3D, we need to ensure that suitable information is available within the CADSTAR data to define the following:

- The height of the components mounted on the board
- The outline shape used by BML to create the simple representation of the components in 3D. This shape is 'extruded' to the specified height
- The correct thickness and material for the electrical layers in the design
- Construction layers (e.g. laminate and pre-preg) to define the correct layer-stack for the design and a material and thickness for these layers

In addition to the above, in order to pass a CADSTAR PCB design to BML, there **must** be a board outline defined in the data – otherwise the transfer will fail. This board outline can easily be replaced later within BML if required.

Specifying the Component Height

The component height may be passed to BML in a number of ways, as follows:

- As an attribute defined in the parts library with the default name of 'height'
- As an attribute defined in the parts library with a user-defined name (e.g. 'Height/Inches')
- As a component property specified in each footprint symbol

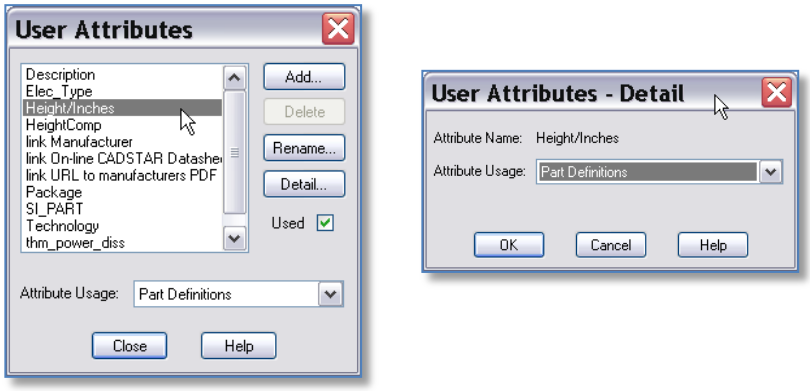
If a component has height information specified as both an attribute on the part and as a property in the footprint, BML will use the specified attribute in preference to the setting in the component footprint. The height in the component footprint will only be used if there is no height attribute present.

Please also note that as attribute names in CADSTAR are not case-sensitive, attribute names such as 'height', 'Height' and 'HEIGHT' would all be considered to be the same attribute. You may still use capitalisation of any letters to assist with readability if required.

Specifying Height as a Part Attribute

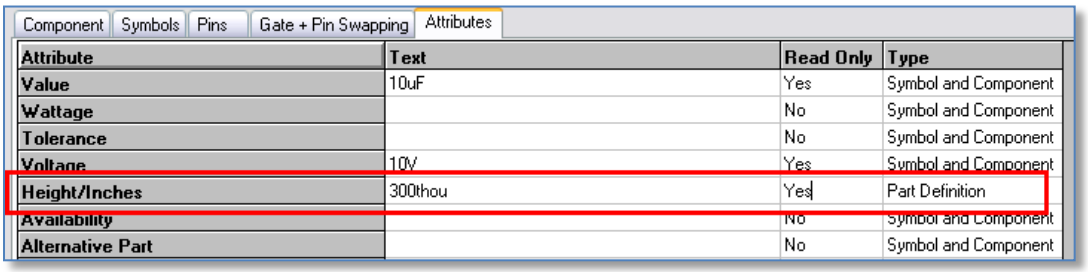
To specify the height as an attribute requires that the parts library is edited. First add an attribute definition to the library, using either the default name ('height') or any other name if preferred. In the example below we have used an attribute called 'Height/Inches'.

Set the ***Attribute Usage*** value to 'Part Definitions'.



Then for each part required, set the value of the attribute to the correct height for the part. You may also decide to make the attribute 'read only' – but this is not necessary for BML to use the information correctly.

The height may be specified together with a suitable units specified (e.g. 300thou in the example below), or if no units are specified, the value will be interpreted in whatever the current design units are.

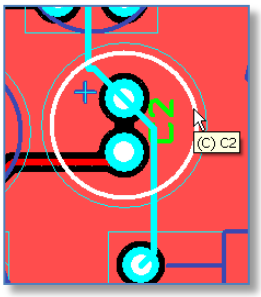


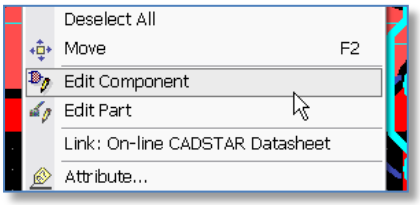
If you have used the default attribute name 'height', then this will be used automatically by BML and no changes to the settings in the ***CADSTAR Import*** dialog are required (simply leave the ***Height attribute:*** field empty).

However, if you have decided to use a different attribute name (e.g. 'Height/Inches' etc.) then in the ***CADSTAR Import*** dialog you must specify the name of the attribute which you would like BML to use.

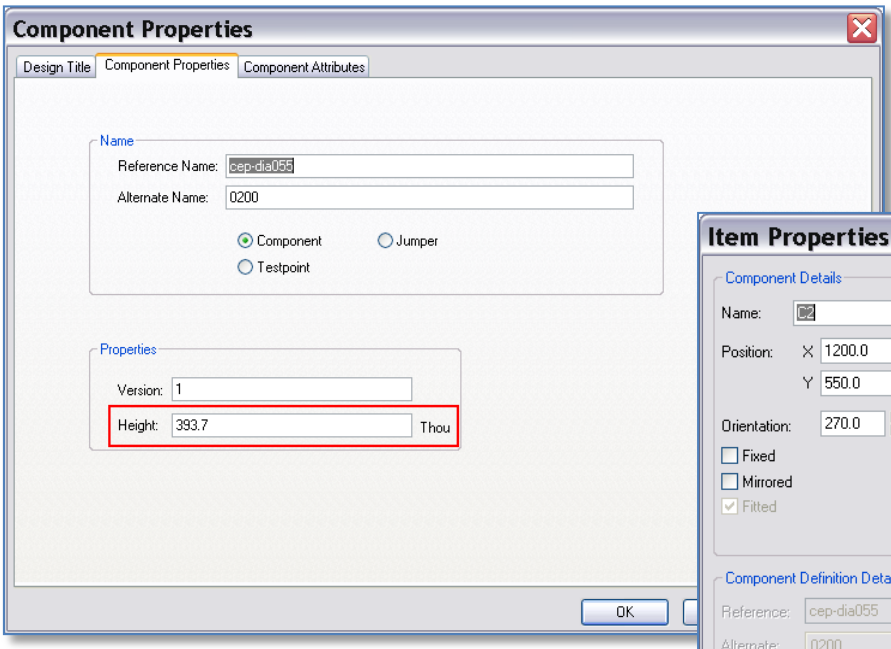
Specifying Height in the Component Footprint

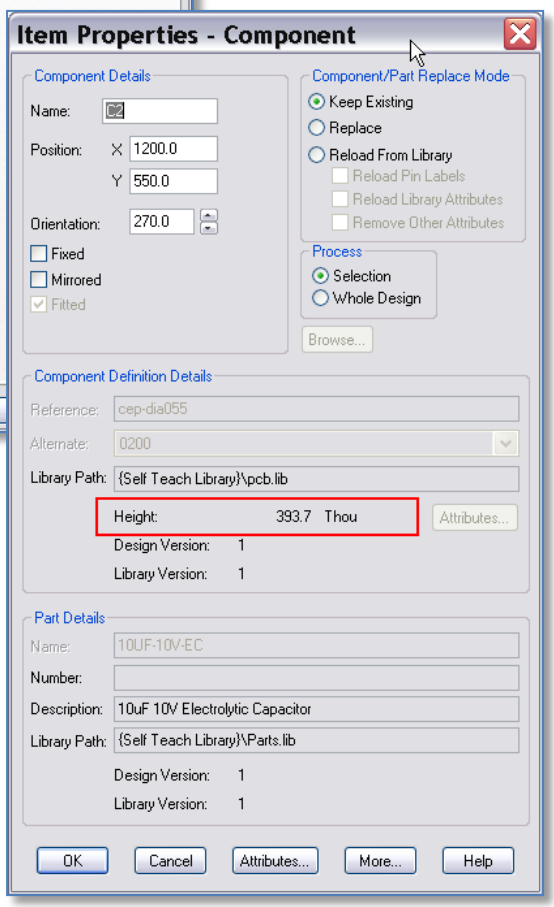
The alternative approach is to specify the height as a 'component property' in the footprint symbol. To set this up, select the required component in the design and open it for editing (select ***Edit Component*** from the right mouse button menu or from the main ***Libraries*** menu).





Once the component has been opened for edit, the height can be set on the ***Component Properties*** dialog (***File / Component Properties...***). Enter the value in the units specified. You will need to re-load the footprint symbol into the design for any changes to be reflected in the design.



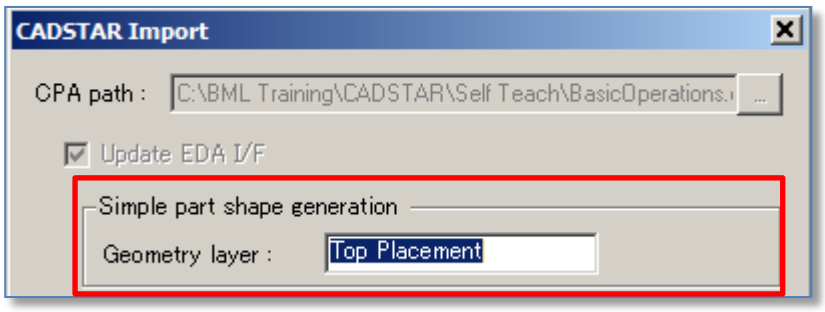


Although this value cannot be changed subsequently in the design (i.e. it can only be modified by editing the footprint symbol again), it can be viewed on the ***Item Properties*** dialog for the component. →

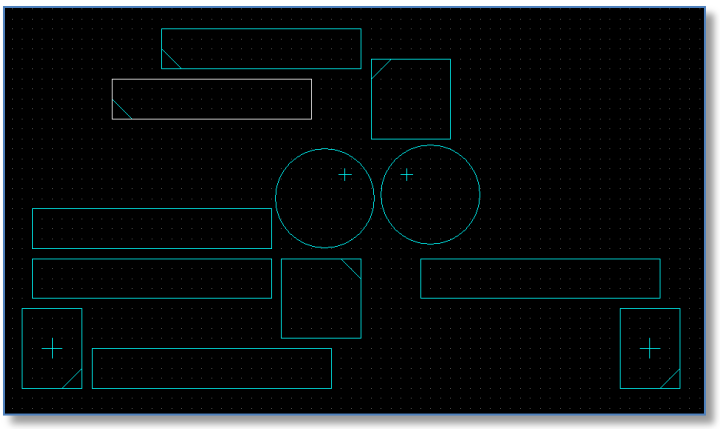
Specifying the Component Outline Shape

To create the default, simple 3D view of components in the design, BML will use a single shape to create a 2D outline, to which it will then add the specified component height (as outlined above), to create the 3D representation.

Users may specify the CADSTAR layer name that the Simple 3D shapes can be read from. I.e. Top Placement, top Silk, Top Assembly, etc., on the CADSTAR Import dialog as the design is passed forward from CADSTAR to BML.



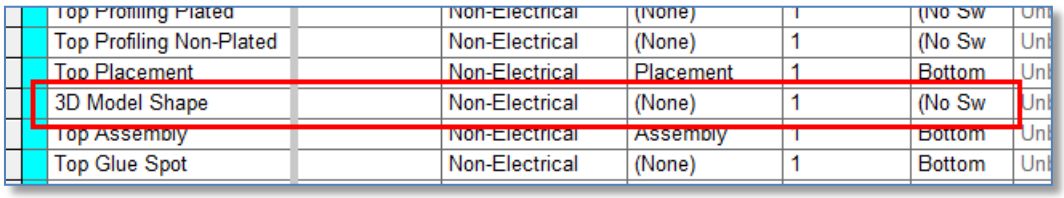
The image below shows the tutorial design in CADSTAR with only the Top Placement layer visible (since there are no components placed on the bottom of the board).



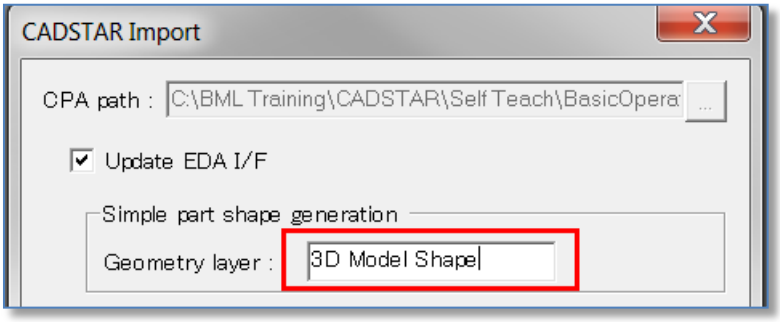
It is possible to use a different layer (other than the placement layer) to define the outline which BML will use to generate the simple 3D representation for the components. This can be achieved under the following conditions:

- There must be a closed shape on the chosen layer to represent the component outline.
- The layer chosen must be a non-electrical layer.

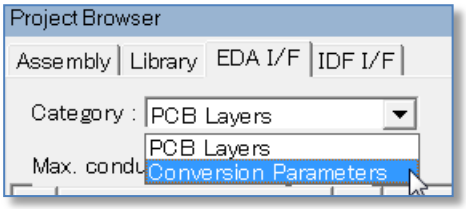
As an example, let's assume that we have added a new non-electrical layer called '3D Model Shape' which will contain the component outlines which we want BML to use.

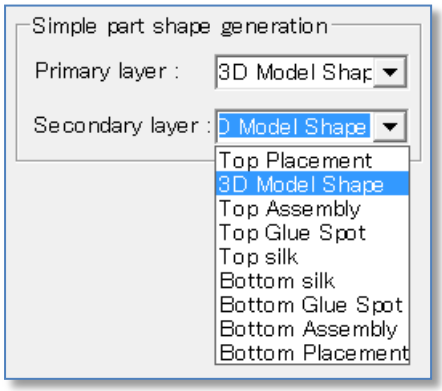


To configure BML to use this new layer, we need to specify the correct layer name on the CADSTAR Import dialog when the design is passed forward to BML.



Note that once the design has been loaded into BML, it is possible to see which layer has been specified under ***Conversion Parameters*** on the ***EDA I/F*** tab of the Project Browser.





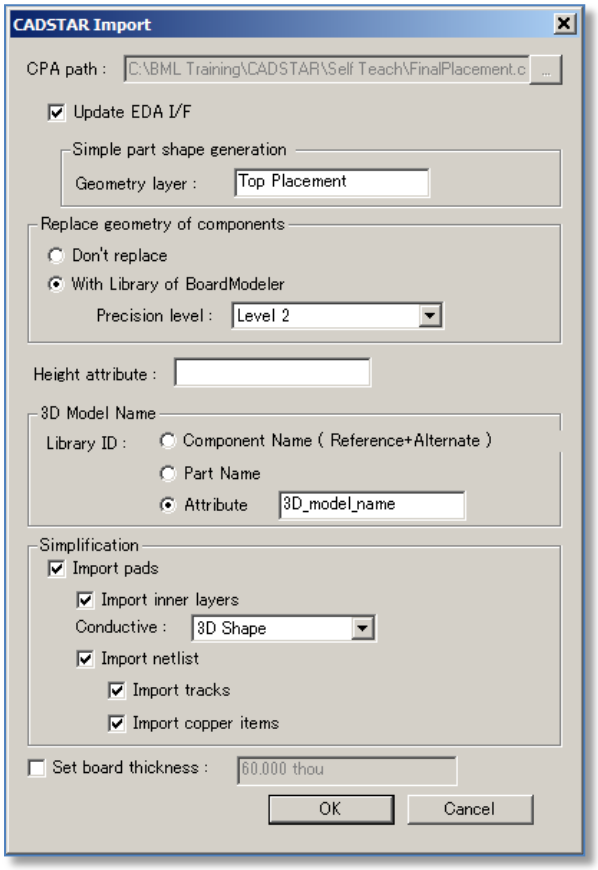
Specifying a 3D model using an attribute in the CADSTAR Parts Library

It is possible to specify the preferred 3D model name to be associated to a CADSTAR Part by using an attribute.

Create a new attribute in your CADSTAR Parts Library I.E. **3D_Model_name**.

Enter the 3D shape file name that exists in the preferred Precision level. I.e. Level 2 of the ***BM/_Library*** folder structure.

Enter the Attribute name in the empty field in the 3D Model Name section of the import dialog.

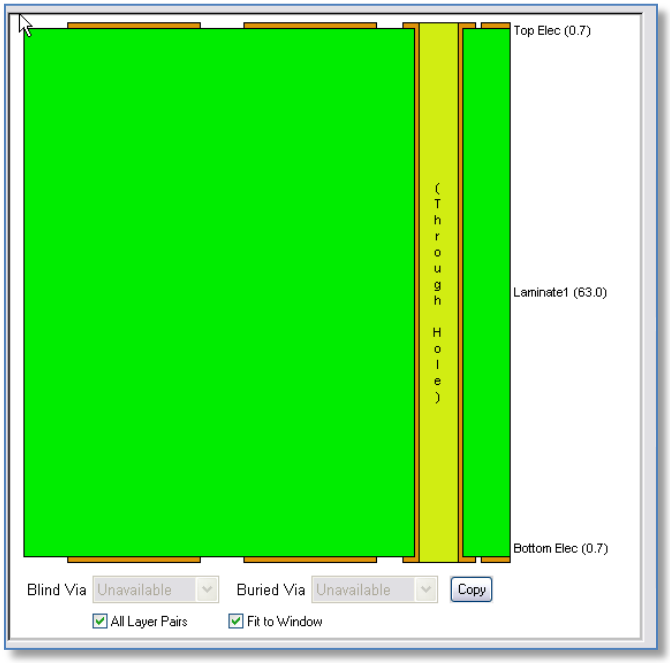


Adding Construction Layers to the Design Layer-stack

The layer-stack for the design used in this tutorial is only a simple, double-sided board. Even so, for it to be represented correctly in 3D within BML it is necessary to define the core laminate layer and assign it the correct thickness. The same applies to the top and bottom electrical (conductor layers). The image below shows data from the CADSTAR Layers dialog showing the layers and their thicknesses.

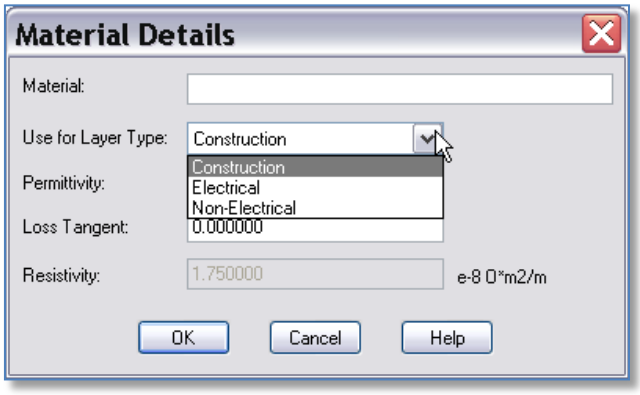
Name	Description	Type	Sub Type	Physical Layer	Swap Layer	Routing Bias	Thickness (Thou)	Material	Embedding
Top Elec		Electrical	(None)	1	Bottom El	X	0.7	Copper Foil	Above
Laminate1		Construction	(None)		(No Swa	Unbiased	63.0	FR4	None
Bottom Elec		Electrical	(None)	2	Top Elec	X	0.7	Copper Foil	Below

You should also set the correct 'embedding' for the electrical layers.\



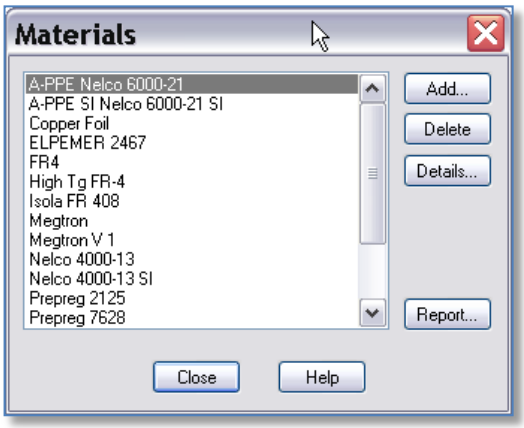
Materials for Electrical, Construction and Non-Electrical Layers

Although BML does not currently use the information regarding the actual materials used for each layer, this may change in the future and it is good practice to set-up the design data so that it is correct. Other tools in the CADSTAR suite (e.g. SI Verify) do require this information to be specified in order to function correctly.



For non-electrical layers (for example, solder mask layers), it should be noted that unless a material is assigned to the layer in CADSTAR, the layer thickness will not be output into the CPA file and therefore would not be available for use within BML.

Materials may be defined or modified by selecting the *Materials* button on the *Layers* dialog. If no suitable pre-defined material exists in the list, select the *Add* button to add a new one and enter the required values on the *Material Details* dialog. Any new material definitions added are saved with the CADSTAR design.



Materials need to be given a name and a type (e.g. Construction, Electrical or Non-Electrical depending on the type of layer they are to be associated with). Depending on the type of the material, one or more of the Permittivity, Loss Tangent and Resistivity values must be specified.

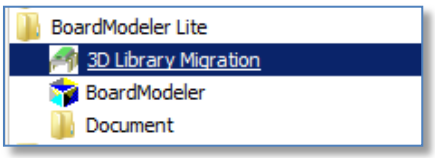
Migrating 3D Libraries from CADSTAR 3D to BoardModeler Lite

Users of the previous 3D EM Collaboration tools

- CADSTAR 3D,
- EM Checker and
- EM Designer

who have a fully developed 3D Library, can easily migrate the legacy folder structure to be BoardModeler (Lite) compliant.

To begin the legacy 3D models migration select ***Start→ Programs →BoardModeler Lite → 3D Library Migration***



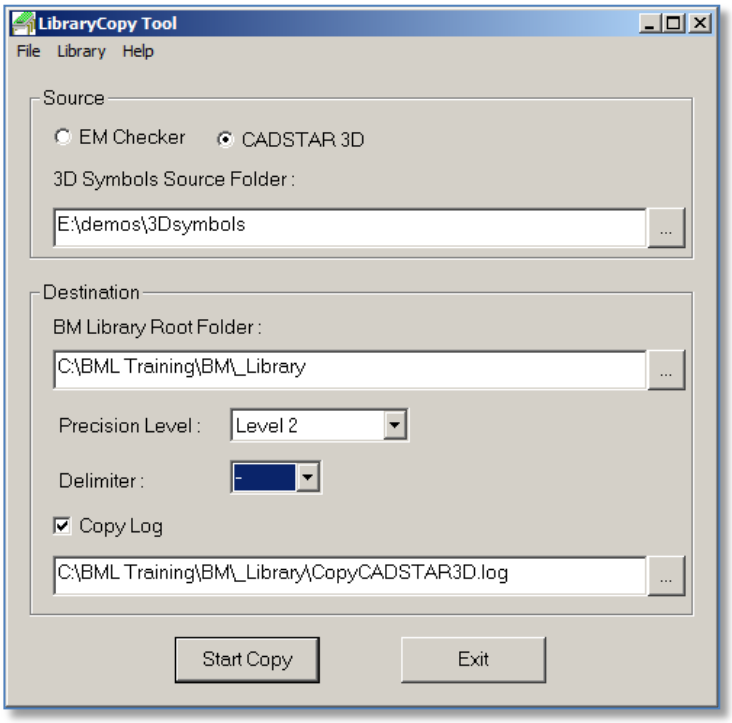
The adjacent dialog will appear.

Select the Source setting depending on the previous Zuken 3D applications, library folder.

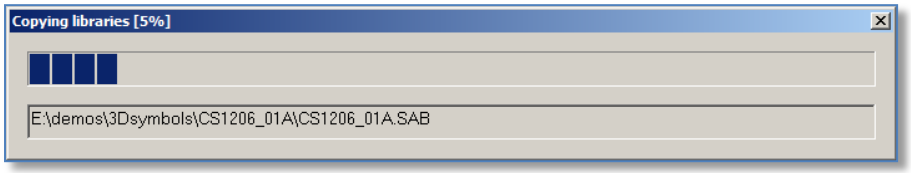
Specify the upper level folder containing the 3D symbols

A default library folder location will be derived from the current BML settings in Tools/Options.
Click the Browse button to redirect the 3D Symbols

Choose the precision level folder of where the legacy symbols will be copied.



Click [**Start Copy**].. A progress indicator will appear as shown below.



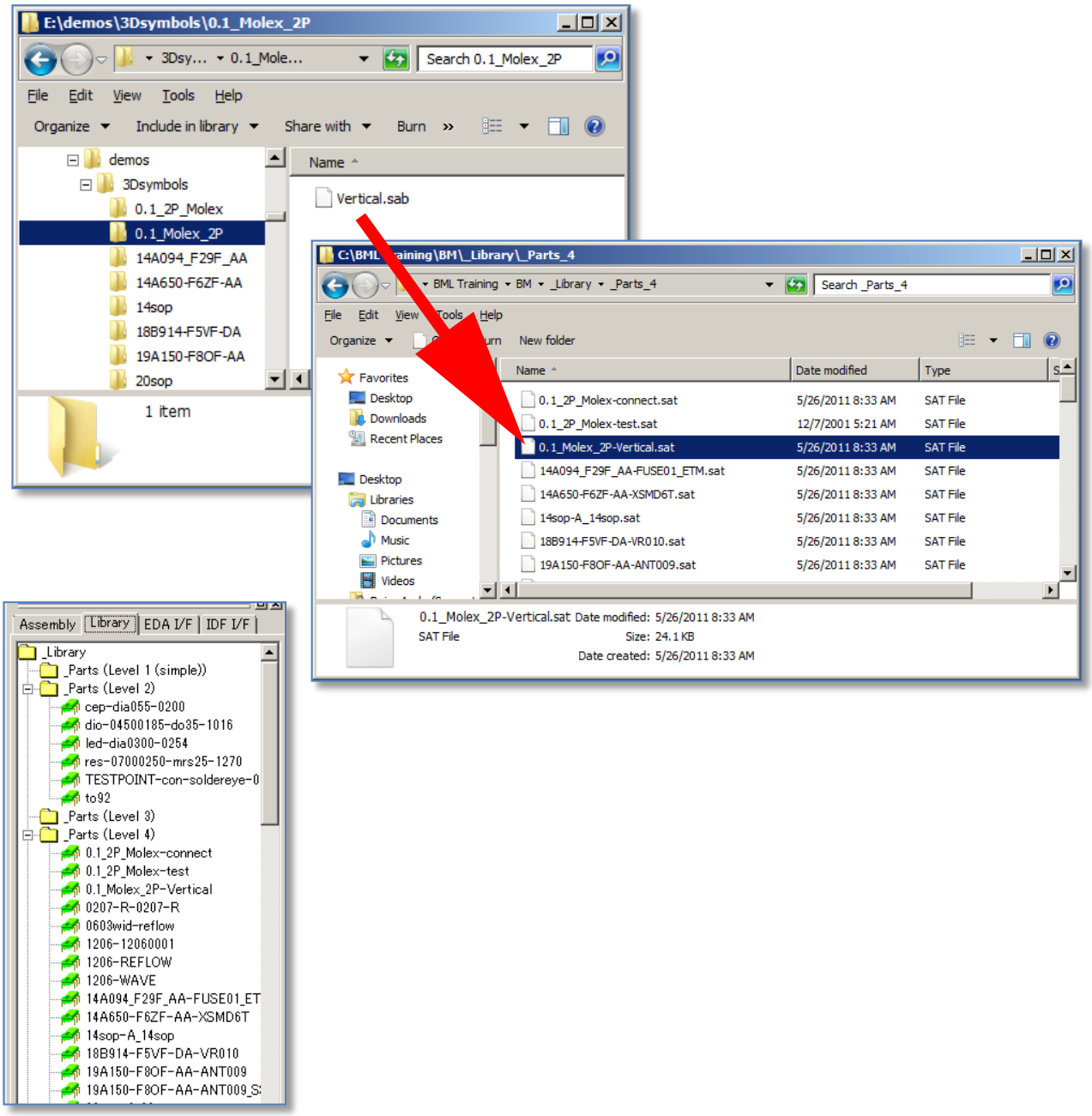
Upon completion the legacy folder and symbol name structure will be migrated into the BM(L) folder structure as follows;

3D symbol folder names containing the actual symbol are migrated as

3d Folder name {delimiter} symbol name → _Parts_4
{Comp name - Alternate name}

0.1_Molex_2P - Vertical.sab → 0.1_Molex_2P-Vertical.sat

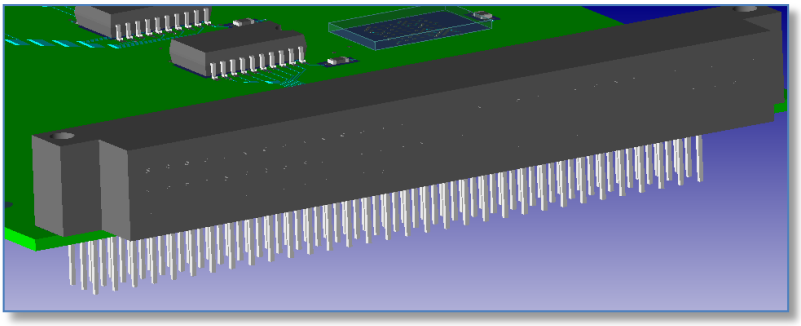
3D Symbol name
Comp name {No Alternate}
Soic16.sat → Soic16.sab



Trimming Through Hole Part leads

When importing 3D models from vendors or using the 3D Part model wizard, it may be necessary to alter through hole part lead lengths to replicate the final assembly length.

Do not worry about lead lengths during the creation stage. Long leaded part models can be trimmed to fit once they are in place.



Located in ***PCB→Utilities→*** select ***Trim leads of parts..***

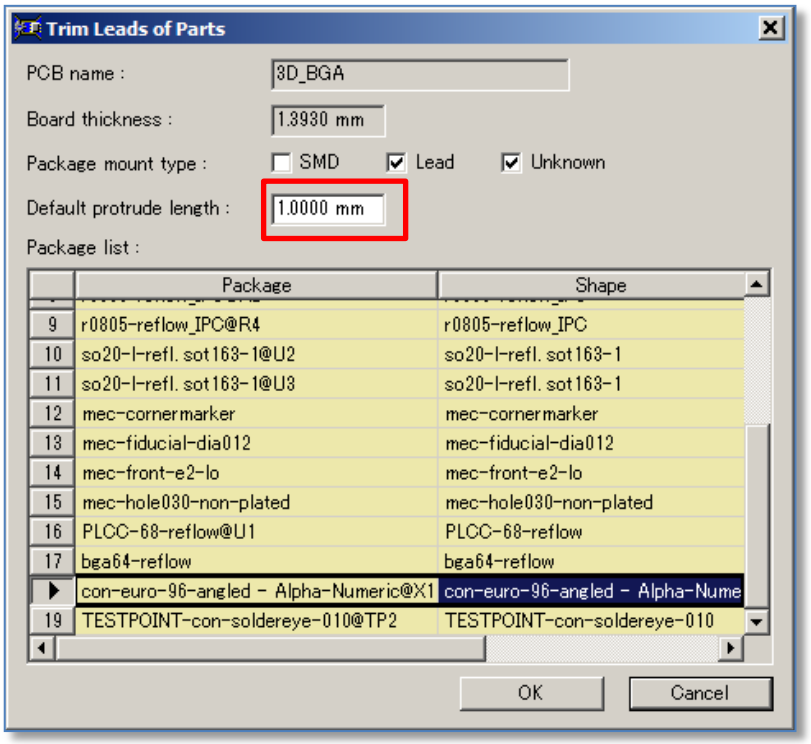
This tool is used as virtual lead trim saw that will adjust the lengths of all through hole leaded part models to meet real world assembly specifications.

Specify the required protrusion length as shown→

Select the parts to be trimmed

Select the [OK] button.

The result is a modified local instance of the 3D part model assigned to the parts in the PCB design.



Note: The actual library model is not altered. To return the part to its original state, simply use the "**Replace with Library**" function.