

Altium ***Designer***

Module 20: Interfacing to 3D Mechanical CAD

Module 20: Interfacing to 3D Mechanical CAD

20.1 Interfacing to 3D Mechanical CAD	20-1
20.1.1 Viewing a Board in 3D	20-1
20.1.2 Exporting a 3D description of the board to an MCAD package	20-2
20.1.3 Checking external mechanical objects against the board.....	20-3
20.1.4 IDF import/export.....	20-4

Software, documentation and related materials:

Copyright © 2009 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, Board Insight, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, P-CAD, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.

Module Seq = 20

20.1 Interfacing to 3D Mechanical CAD

Altium Designer has strong support for interfacing to mechanical CAD design tools, including 3D visualization within the PCB editor, and 2D and 3D export. The completed PCB can be exported as a 3D STEP format file, or a 2D description can be exported as IDF.

You can also import 3D STEP objects, such as components and housings, into the PCB library and PCB editors, and also import 2D IDF and DXF data.

20.1.1 Viewing a Board in 3D

- To view the board in 3D, press the **3** shortcut key.

Note: Viewing the board in 3D requires a graphics card that supports DirectX 9 or better, and Shader Model 3 or better. If you are unsure if your graphics card meets these specifications, click the button in the **PCB Editor – Display** page of the *Preferences* dialog.

- By default the board will display as a blank, or unloaded board. Press the **L** shortcut to display the *View Configurations* dialog, where you can configure the color scheme, and also enable the display of 3D bodies. Note that components can only be displayed if the footprint includes 3D body objects, or an imported STEP format component description.
- Press the **2** shortcut to change back to the traditional 2D display mode.

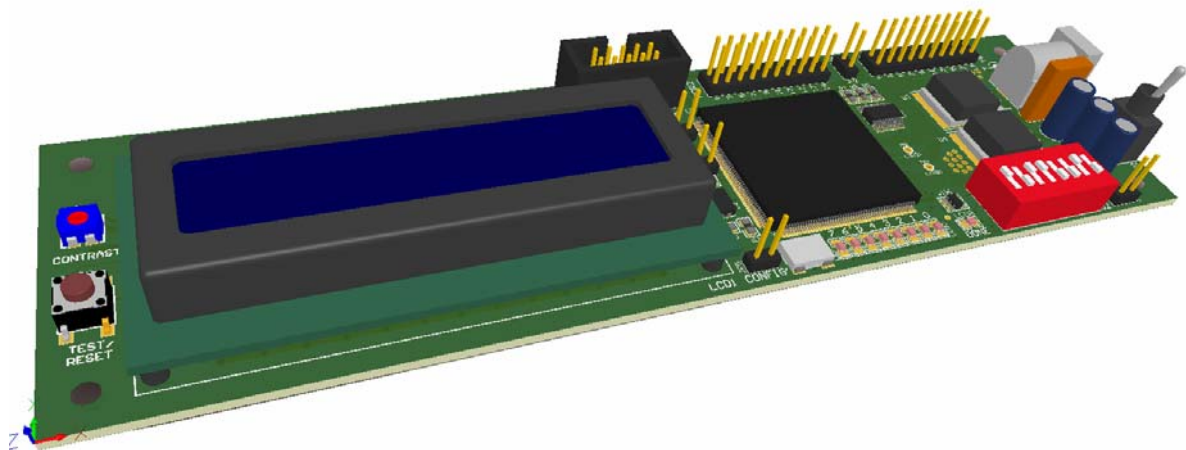


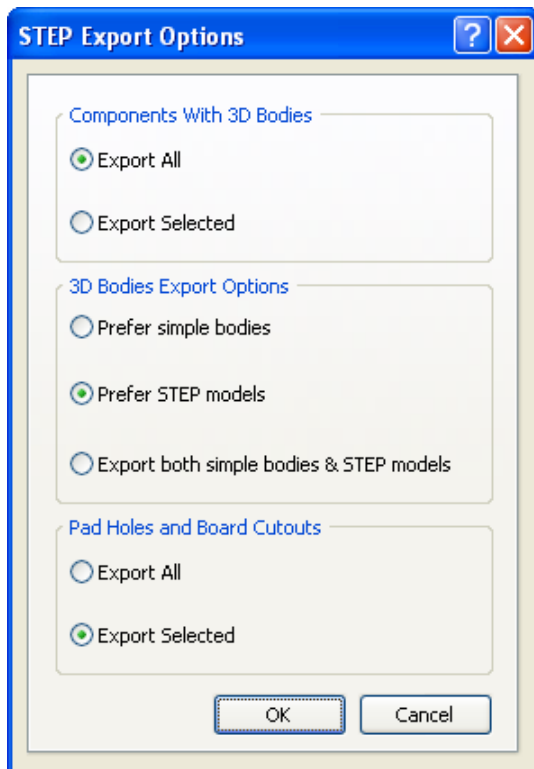
Figure 1. 3D view of Altium's Spiritlevel design.

- To rotate the 3D view of the board, hold the Shift key to display the spherical rotation cursor, then click and hold the right mouse button and drag the board around to a new position.
- Altium Designer also supports the Space Navigator, a 3 dimensional navigation device developed specifically for 3D CAD work. Visit www.3dconnexion.com for more information.

20.1.2 Exporting a 3D description of the board to an MCAD package

The best way to pass a 3 dimensional description of your board to your MCAD software is to export it from Altium Designer as a STEP file.

- Although STEP (**S**tandard for the **E**xchange of **P**roduct model data – ISO10303) is a standard data interchange format, not all CAD packages produce the same STEP format files, so you may need to experiment with the export options to see what gives the best result.
- To export the board to a STEP format file, select **File » Save As** from the menus, and select **Export STEP** from the **Save as Type** list. The *STEP Export Options* dialog will appear, where you can configure what is exported, and how components are exported. Exporting holes will increase the file size, exclude these if necessary.



Note: For information on creating 3D component definitions from imported STEP models or Altium Designer 3d bodies, ready for use in your board, refer to the training module *Linking Models, Parameters, Library Package and Updates*.

Figure 2. The STEP Export options dialog

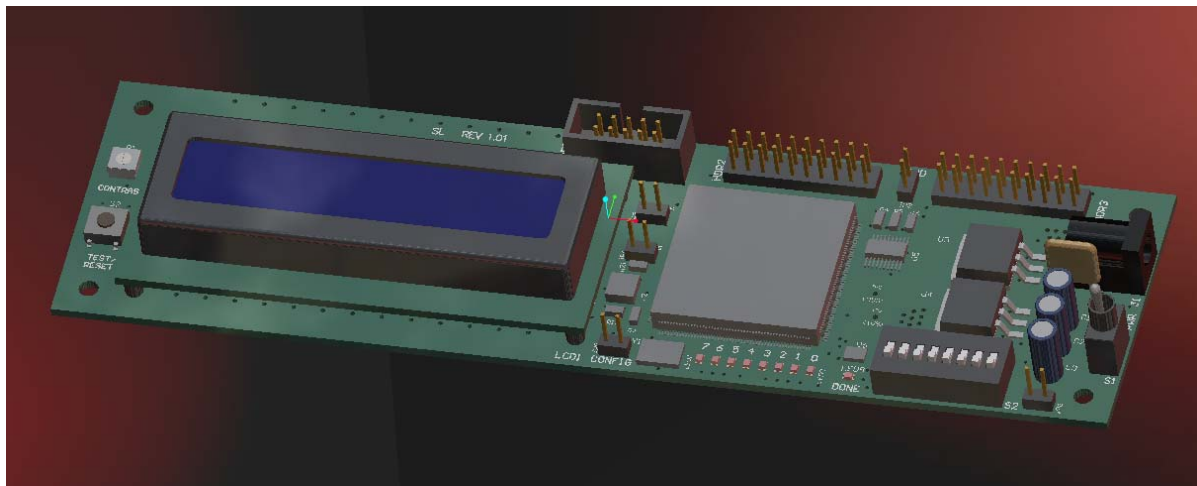


Figure 3. The STEP file imported into Pro Engineer Wildfire

20.1.3 Checking external mechanical objects against the board

A common development requirement for a compact, complex electronic product is to be able to test the fit of the board inside its housing, before committing to manufacturing the board or the tooling for the housing.

- Altium Designer supports importing of any 3D model, such as a case or a bracket, directly into the PCB editor. To do this, switch to 3D view mode, then select **Place » 3D Model** from the menus.
- Orient the model using the Spacebar and arrow keys on the numeric keypad, or use the **PCB Inspector** panel to test different rotation settings.
- When you click and hold to move the model into position, you will be holding it by the nearest vertex in the model. You can also add your own snap points, either via the *3D body* dialog (double click on the imported model to open it), or via the **Tools » 3D Body Placement » Add Snap Points From Vertices** command.
- Set the **PCB Panel** to **3D Models** mode, select the model, and use the **Highlighted Models** drop down to change the transparency level

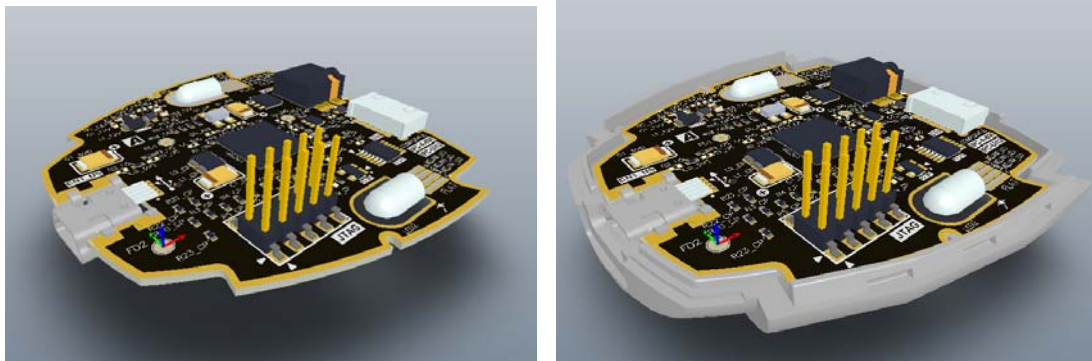


Figure 4. Board only in 3D view mode, then shown inside bottom half of enclosure.



Figure 5. Note how connector is currently interfering with enclosure (DRC violations disabled).

20.1.4 IDF import/export

Altium Designer supports both import and export of IDF v2.0 and v3.0.

- Two files are generated when exporting to IDF, *.BRD and *.PRO. The BRD file is the board file and the PRO file is the library information.
- The exported BRD file contains the board outline and component position information.
- The PRO file details the names of the components and some basic component shape information. If component bodies are used this information is a lot more usable. The ideal solution is to have a 3D representation of the component in your MCAD package, which is placed from the library information.
- To export the current board, select **File » Save As** and set the **Save As Type** option to **Export SDRG-IDF Brd Files**.

Note: Different CAD packages use different file extensions to Altium Designer, for example Pro Engineer uses:
*.emp - IDF library information (also called profiles files)
*.emn - IDF board files (also called neutral files)

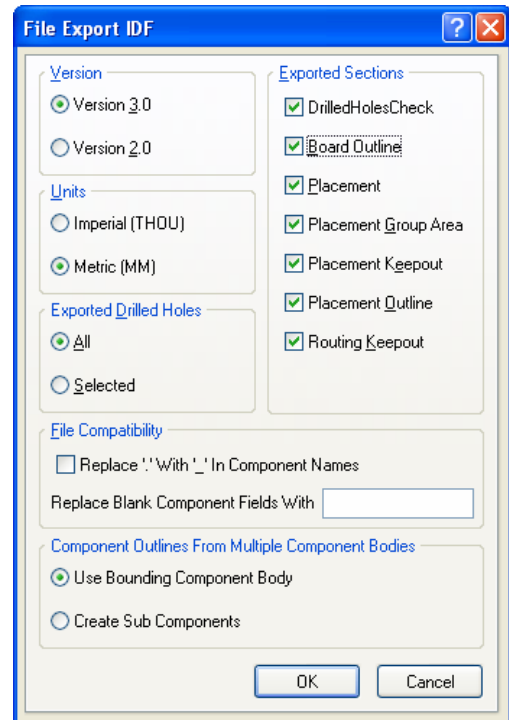


Figure 6. Export to IDF dialog

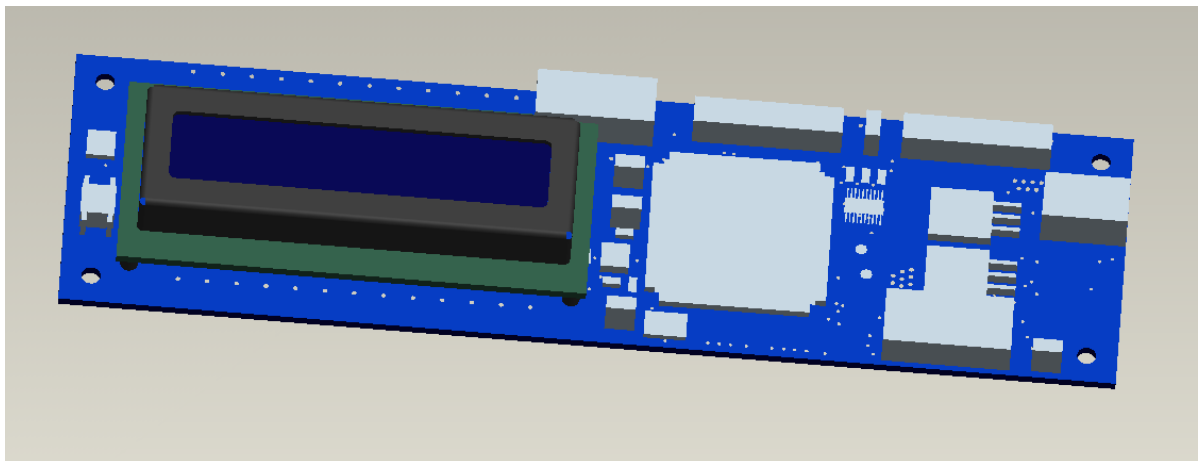


Figure 7. Imported IDF with a 3D component for the LCD screen drawn from a Pro Engineer library. The rest are built from Altium Designer component bodies.