Altium Designer

Advanced Training with Altium 365

Clearance Checking in 3D

Software, documentation and related materials:

Table of Contents

Clearance Checking in 3D

1 Purpose

In this module, the supplied design has 3D bodies attached to most of the components. You will be working with the Xilinx Spartan-IIE FPGA (U1) and the 16-character two-line LCD display (LCD1) to investigate the interaction of the component 3D bodies in clearance checking, and experiment with the Component Clearance rule constraints.

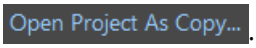
Note: If a 3D Mouse or Space Navigator is available, feel free to use the 3D Mouse in the Altium Designer 3D environment.

2 Shortcuts

Shortcuts used when working with Clearance Checking in 3D

F1	Help – Shortcut Key List
2	2D Mode
3	3D Mode
L	View Configuration Panel
R	Component Placement - Change Pushing Mode
In 3D Mode:	
8	Isometric View
0	Top View
SHIFT + Right Mouse Button	Navigation Ball / Orb to rotate the 3D View

3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `Clearance Checking in 3D`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button  .
5. Navigate to the predefined Training Project `Clearance Checking in 3D (Top\Projects\Altium Designer Advanced Training Course\...)`.
6. Select **Open Project as Copy...**  .
7. In the new dialog *Create Project Copy:*
   a) Add your name to the project name: `Clearance Checking in 3D – [Your Name]`.
   b) Add a description: `Altium Advanced Training – [Your name]`.
   c) Open the *Advanced* section.
   d) Select the **Ellipsis Button**  from the *Folder* configuration to open the *Choose Folder* dialog.
      i) Select the folder with your name: `Project\For Attendees\[Your name]`.
      ii) Select **OK**.
   e) Change the **Local Storage** path if needed.
   f) Select **OK** to create the copy.
8. Wait until Altium Designer creates the copy of the project and opened the Project for you in the *Projects* panel, this may take up to 1 minute.

Hint: For details how to copy the predefined training project, see module *03 Getting started - Opening a Project*.

4 3D View Control Menu

9. When in the 3D view, as well as using some of the options described in this module, there is a dedicated 3D menu, as shown in Figure 1 below. Also, notice on the right of the menus, there are keyboard shortcuts that can be applied using the numerical keys on the right-hand side of a full keyboard.

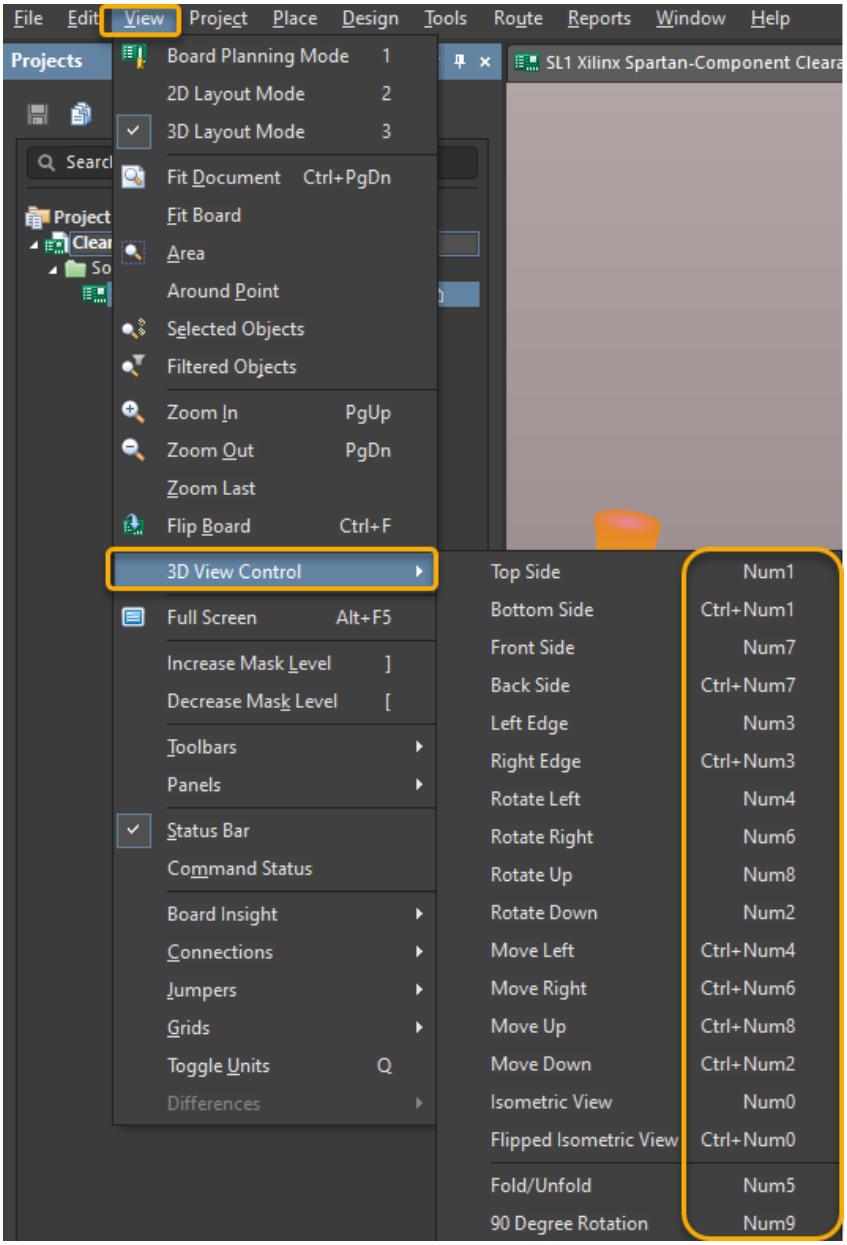


Figure 1. 3D View Control Menu

5 Positioning Components

5.1 Viewing the Component Bodies in 3D

10. Open the PCB `SL1 Xilinx Spartan-Component Clearance.PcbDoc`.
11. If not already in the 3D view, switch to the 3D using the menu **View » 3D Layout Mode** or use the shortcut key **3** on the alpha keyboard.
12. Press the **8** key at the top of the keyboard to rotate the board to an isometric view.
13. Press and hold the **Shift** key. A golden Navigation ball will appear allowing you to move the board in 3D, Figure 2.
14. Keeping the **Shift** key held down, press and hold the **Right Mouse Button**. With both buttons held down, move the mouse to rotate the PCB.

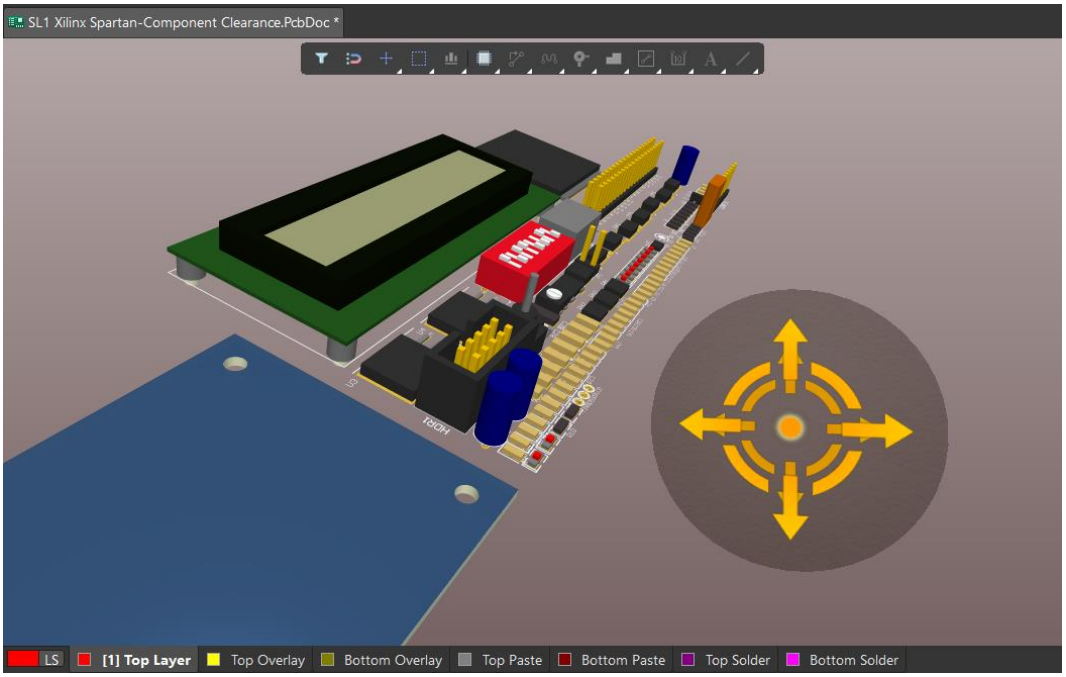


Figure 2. Navigation Orb used to navigate in 3 dimensions

15. Orient the PCB so you can see the clearance between the bottom of the LCD display (LCD1) and the top of the FPGA (U1).

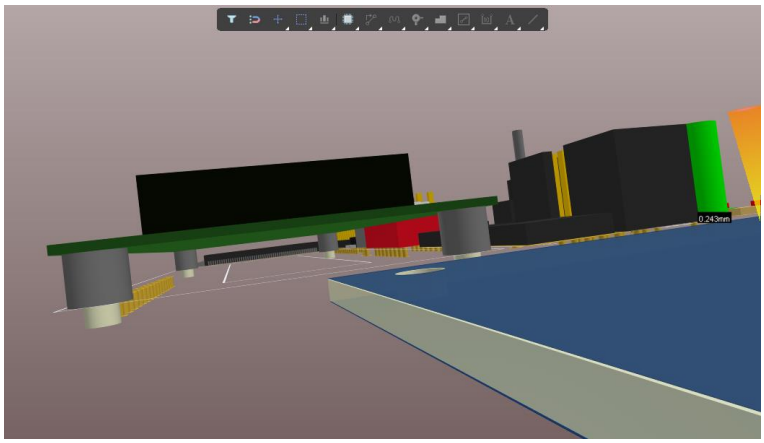


Figure 3. Clearance between LCD and FPGA

16. Press the **0** (zero) key on the alpha keyboard to rotate the PCB to a facing surface.
17. Switch the view back to 2D mode: **View » 2D Layout Mode** or use the shortcut key **2** on the alpha keyboard.
18. In the **Preferences - PCB Editor –** General (Figure 4), follow the steps below:
    a) Verify the **Snap To Center** option is enabled, and the **Smart Component Snap** is disabled. This configuration will snap the mouse cursor to a component reference point. The **Smart Component Snap** will snap the cursor to the nearest hotspot object of the component when left clicking and holding.
    b) Ensure the **Online DRC** is enabled, the option is located just above the Object Snap Options.

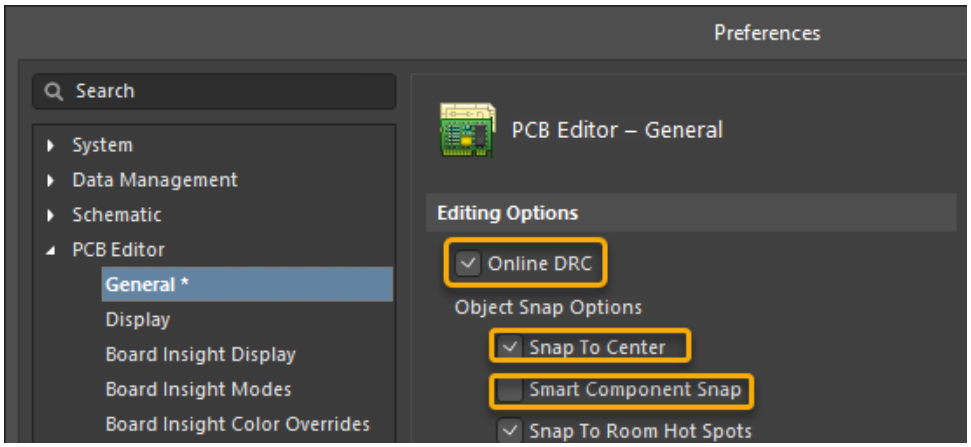


Figure 4. PCB Preferences

    c) Select **OK** to close the *Preferences.*
19. Following the steps below, make Component Reference points visible:
    a) Press **L** to open the *View Configuration* panel. If opening for the first time, it may need docking.
    b) In the *Layers & Colors* Tab, scroll down to the *System Colors* section.
    c) Ensure the Visibility icon  next to the **Component Reference Point** is enabled.
20. To position the LCD display component, adhere to the instructions below:
    a) Select **Move » Component** (**M»C**) or select **Edit » Move » Component** from the main menu, and hold on the component `LCD1`. While holding the component, drag it on your cursor.
    b) With the component attached on your cursor, type **J-L** on your keyboard (shortcut for Jump To Location).
    c) Release the left mouse button when the *Jump To Location* dialog opens and enter the values X = `53mm` Y = `43mm`.
    d) Once you have entered the X and Y values, press the **Enter key twice**. The first enter press will move the component to the location, the second enter will anchor the component at this location. It's important that you don't move the mouse between Enter presses.
    e) Pres **ESC** or do a right-click to stop the Move-Component command.

21. Next, you will position the FPGA `U1` under the body of the LCD display at location X = `85mm` and Y = `21mm`, using the *Properties* panel, as descried below:
    a) Ensure the **Double Click Runs Interactive Properties** option is enabled in the *General* section of the PCB Editor *Preferences*.
    b) Select `U1` using left mouse click, if the *Properties* panel is already open. Double left mouse click if it's not, to make it pop up (it may need to be docked after it opens).
    c) Enter the new coordinates in the *Location* section of the *General* tab and press **Enter**.
22. Since there is enough vertical clearance, no DRC violations are generated, as shown back in 3D mode in Figure 5 (switching to 3D mode not required yet).

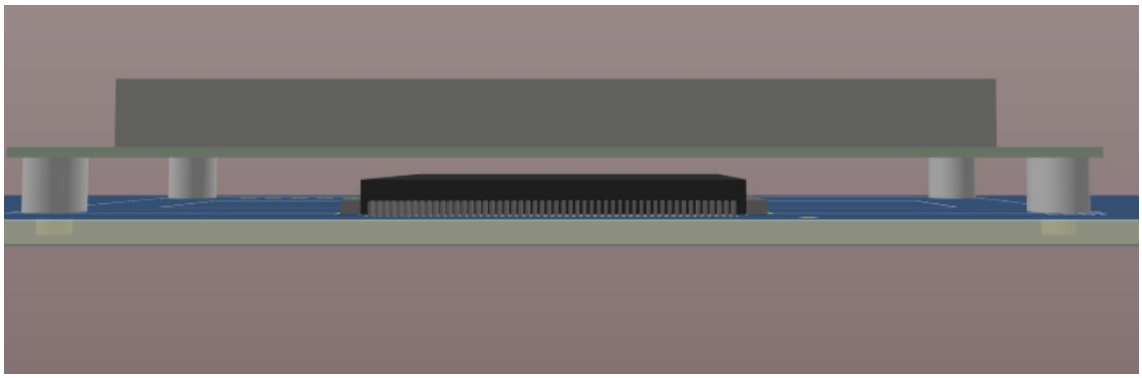


Figure 5. Sufficient vertical clearance

23. Move the FPGA to a new location using the **Move By... Selection** command:
    a) Select `U1`.
    b) Select **Edit » Move » Move Selection by X, Y...** and enter *X Offset:* `-2mm` and *Y Offset:* `5.5mm`.
    c) Select **OK**. This should cause the pads of the LCD display and the FPGA to violate and will highlight in the online DRC violation color, Figure 6. This is an Electrical Clearance rule violation and not a Component Clearance rule violation.

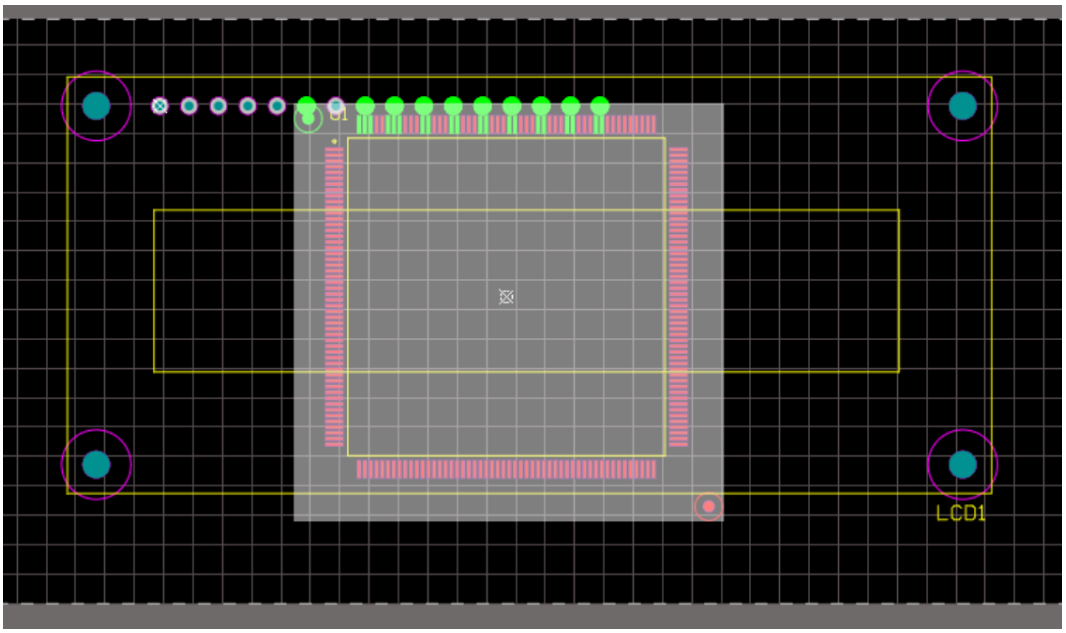


Figure 6. Pads violating

5.2 Component Body Clearance

24. Switch to the 3D view by pressing the **3** key.
25. Open the *PCB* panel *(if not already open)* from the **Panels** button or with **View » Panels » PCB**.
    a) Set the scope of the *PCB* panel (using the drop-down arrow at the top) to **3D Models**.
    b) In the *Component Classes* section, select the **<All Components>** class.
    c) Scroll down through the components in the center area of the *PCB* panel to the LCD display, `LCD1`, and select it as shown in Figure 7.

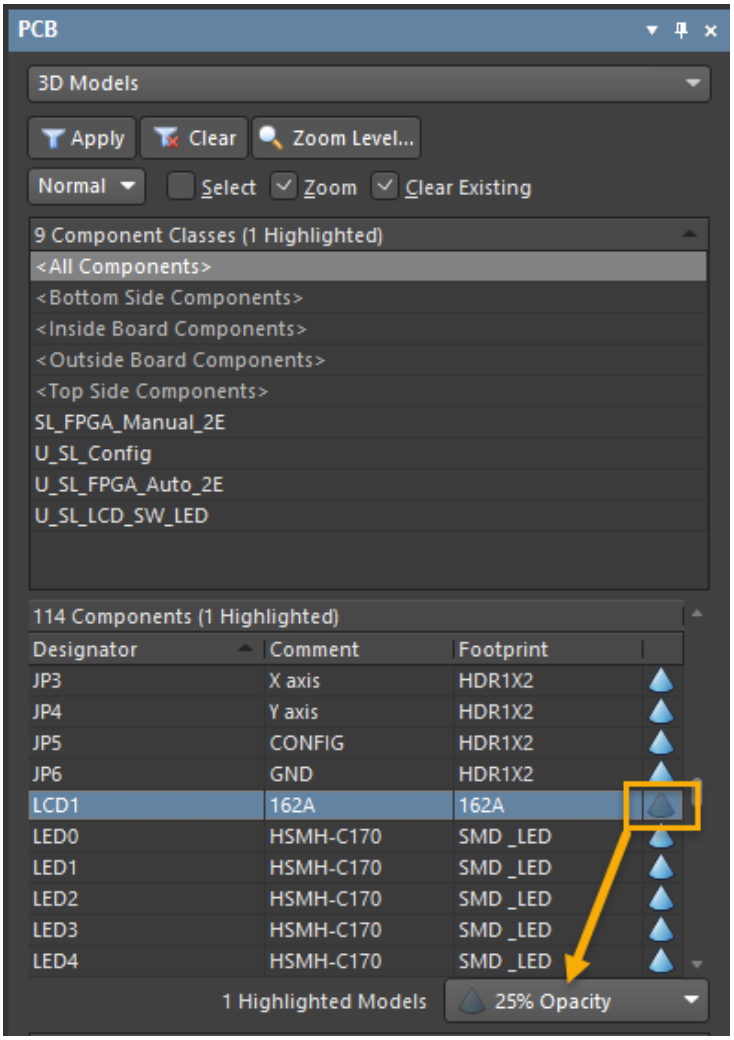


Figure 7. Locate LCD1 using the PCB panel

    d) Click on the blue cone shaped ICON to the right of `LCD1` to toggle the opacity of the LCD 3D body between Solid 100%, Hide, 75%, 50%, and 25%. Set the opacity of the LCD display to 25% so that you are able to see through it and view the FPGA underneath.

> Hint: Changing the Highlighted Models drop-down manually, allows setting the opacity for the selected components.

26. Select `U1` and move it to the left until it highlights in the DRC violation color. This is the Component Clearance rule violation.
    If `U1` is jumping outside the Board, or the `LCD1` is pushed, change the Component Pushing mode by pressing **R** on the keyboard while holding the left mouse button.
27. Move `U1` back to the right until the violation is removed.

5.3 Component Clearance Rules

28. Open the *PCB Rules and Constraints Editor* by accessing the command **Design » Rules**.
29. In the pane on the left side of the dialog browse to the *Placement - Component Clearance* branch and select the rule `ComponentClearance_LCD_U1`, Figure 8.
30. Note the scope of the two queries **Component - LCD1** and **Component - U1**. This scopes the rule to a Component Clearance specifically between `LCD1` and `U1`.

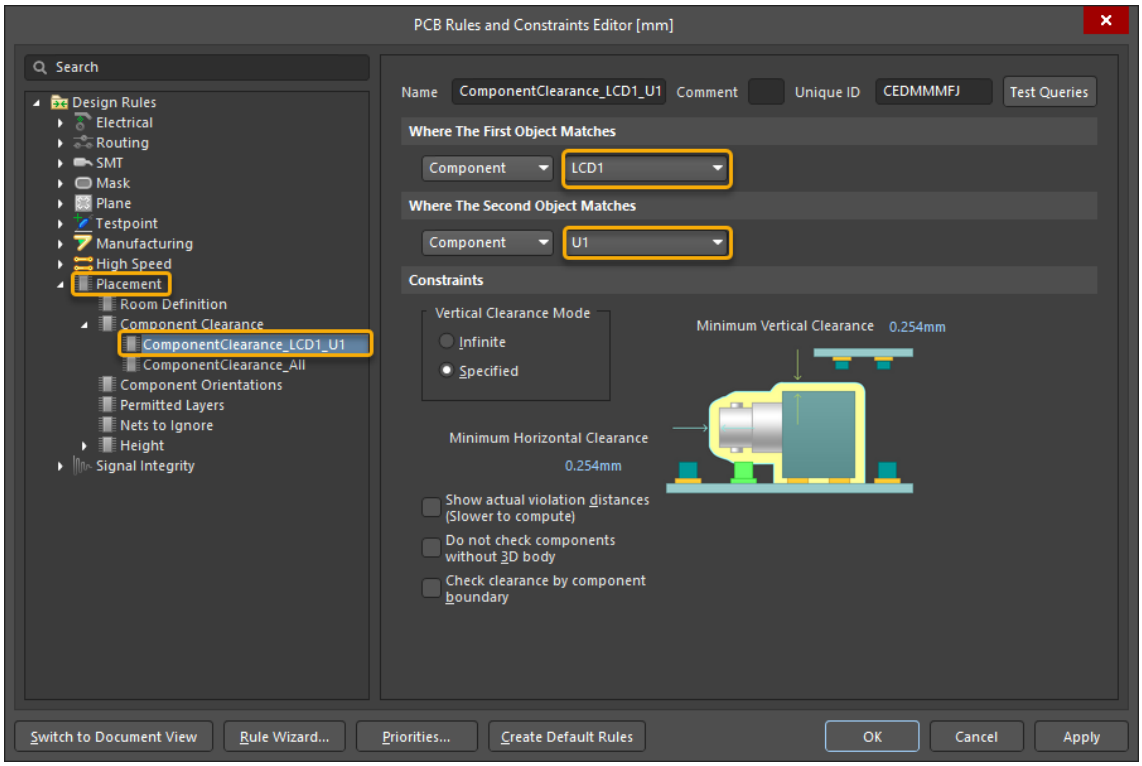


Figure 8. Component Clearance Design Rule for LCD1 and U1

31. Change the **Minimum Vertical Clearance Constraint** to `10mm` and select **OK**.
32. In the PCB, the LCD display and the FPGA will be highlighted in the DRC violation color, as illustrated in Figure 9.

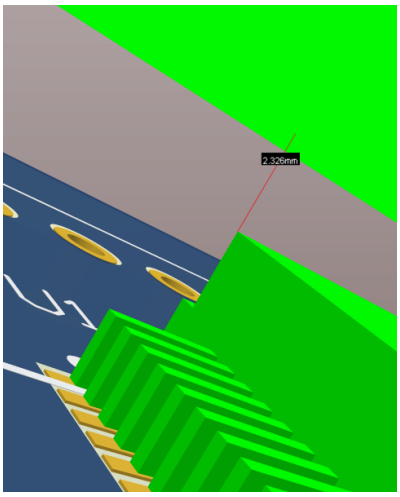


Figure 9. DRC Violation indicated by LCD and FPGA highlighted in green

33. Zoom in on the black collision indicator. This may require rotating the PCB to view the clearance between the two components. Note how the collision is indicated in the Z axis with a fine vertical line.
34. This next step will modify the clearance rule to remove the vertical violation and introduce a new horizontal violation. Open the *PCB Rules and Constraints Editor* and modify the Component Clearance rule for `LCD1` to `U1`:
    a) Set the **Minimum Vertical Clearance** back to `0.254mm`.
    b) Set the **Minimum Horizontal Clearance** to `50mm`.
35. Note how the collision is now indicated in the X and Y Axis with a fine horizontal line, as shown in Figure 10.

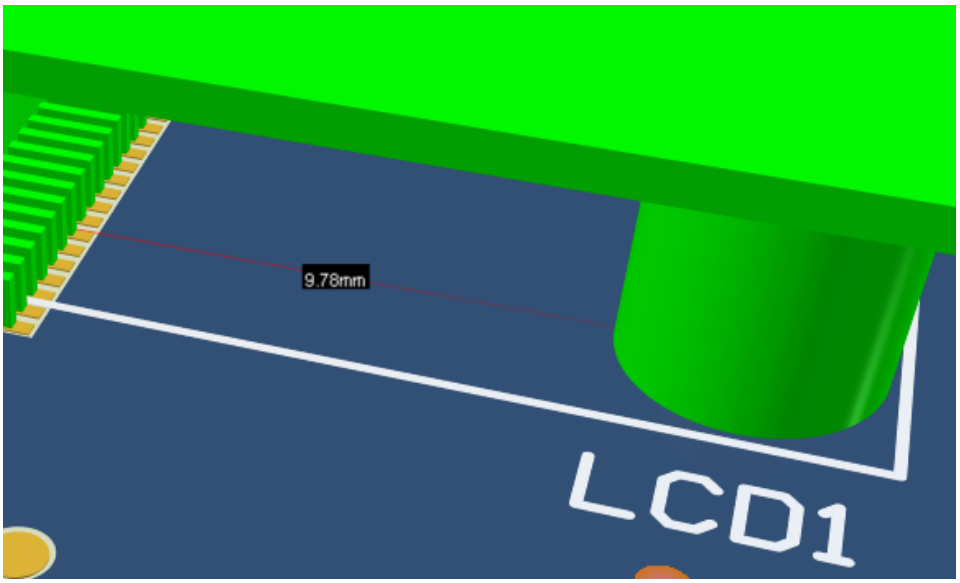
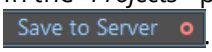


Figure 10. Collision between FPGA and LCD is indicated by a fine horizontal line

6 Conclusion

Component clearance is achieved independently from the Electrical Clearance rules. 3D component bodies are used to determine the clearance between components and the Component Clearance rule constraints adjust the minimum gap between 3D bodies of the components.

36. Save all documents using **File » Save All**.
37. Save the modifications to the server:
    a) In the *Projects* panel, next to the Project name you find the command **Save to Server** 
    b) Select **Save to Server**.
    c) In the dialog *Save [Project Name]:*
        i) Add the comment `Clearance Checking in 3D - [Add Your Name] - Finished`.
        ii) Select **OK**.
38. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Clearance Checking in 3D

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**