Altair PollEx for Altium 2021.2

Tutorials

Updated: 2/10/2022

altair.com

Contents

Altair PollEx for Altium Tutorial

Altair PollEx for Altium is available for Altium users with limited functionalities. The Altair PollEx for Altium supports limited functions of PollEx PCB (a PCB design viewer), DFM (Design for Manufacturing), DFE (Design for Electric), SI (Signal Integrity), and Thermal analysis.

This tutorial will focus on downloading the tools needed and running a quick DFM check on a test board to confirm functionality.

For additional questions, please contact our PollEx for Altium support team at pfasupport@altair.com.

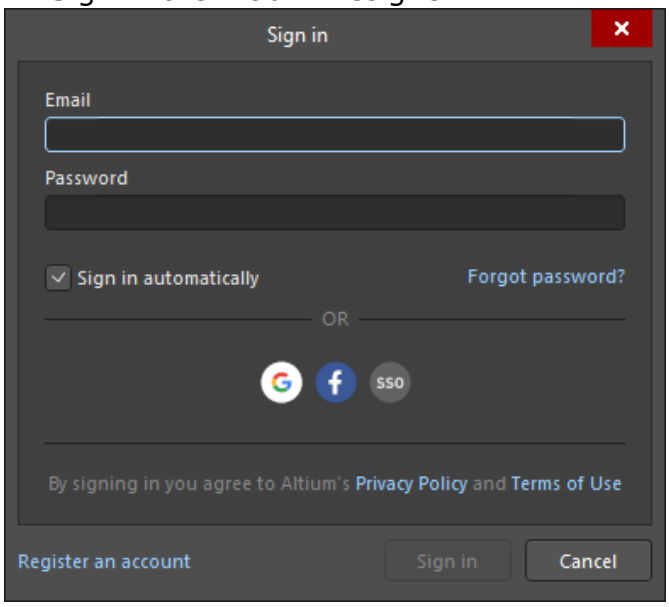
1. Altair PollEx for Altium Workflow

The Altair PollEx for Altium workflow has two significant parts: The PollEx for Altium standalone software and the Altium Extension. The Altium extension will automate the process of exporting the .PcbDoc file into a PDBB project file that PollEx can read.

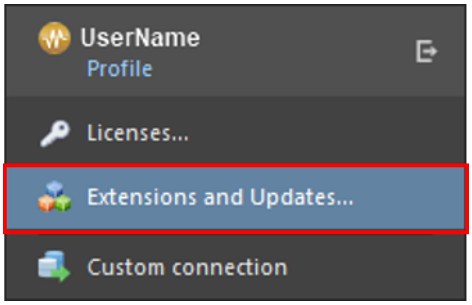
The common workflow is to use the Altair PollEx for Altium extension to export the current state of the PcbDoc in an ASCII format. PollEx is then used to run analysis and verification on the board. If changes to the board need to be made, they can be made within Altium Designer, and then exported again to PollEx to run another analysis or verification.

2. Install Altium Extension

2.1. Sign in the Altium Designer



2.2. From the Current User Information, click **Extensions & Updates**.



2.3. Click **Purchased**.

2.4. Install **AltairPollEx** extension from the Software Extensions.

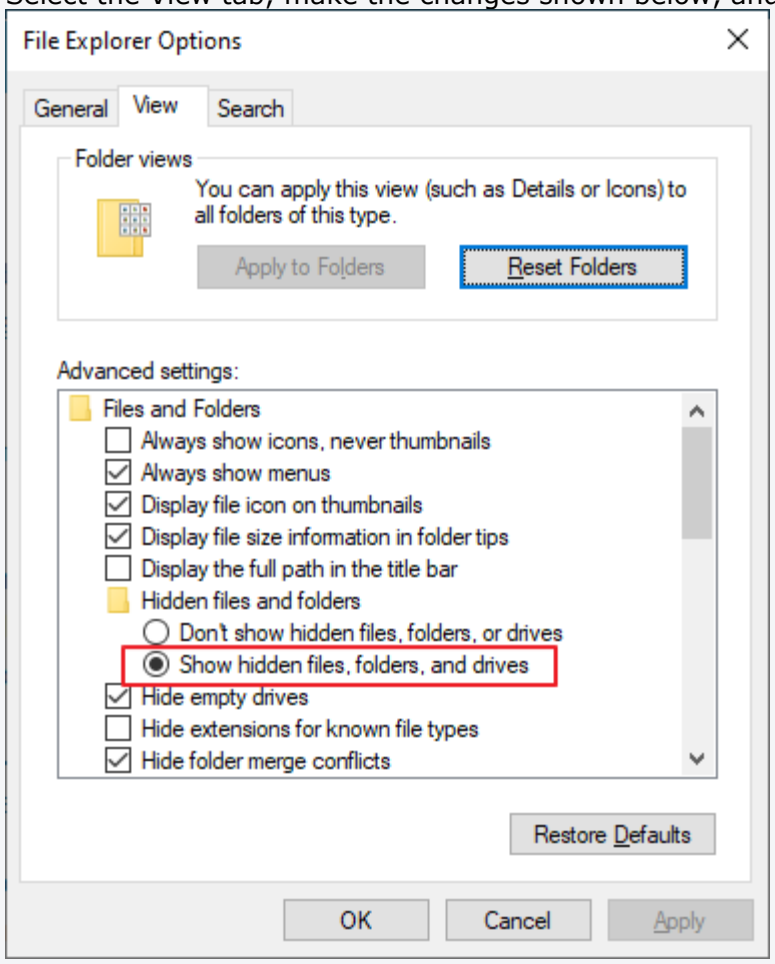
2.5. Restart the Altium Designer to complete the install process.

3. Altair PollEx Environment

For this tutorial, you will use a sample file located in `C:\ProgramData\altair\PollExForAltium\2021.2\Examples` directory. This will help aid in the process of learning the Altair PollEx tool.

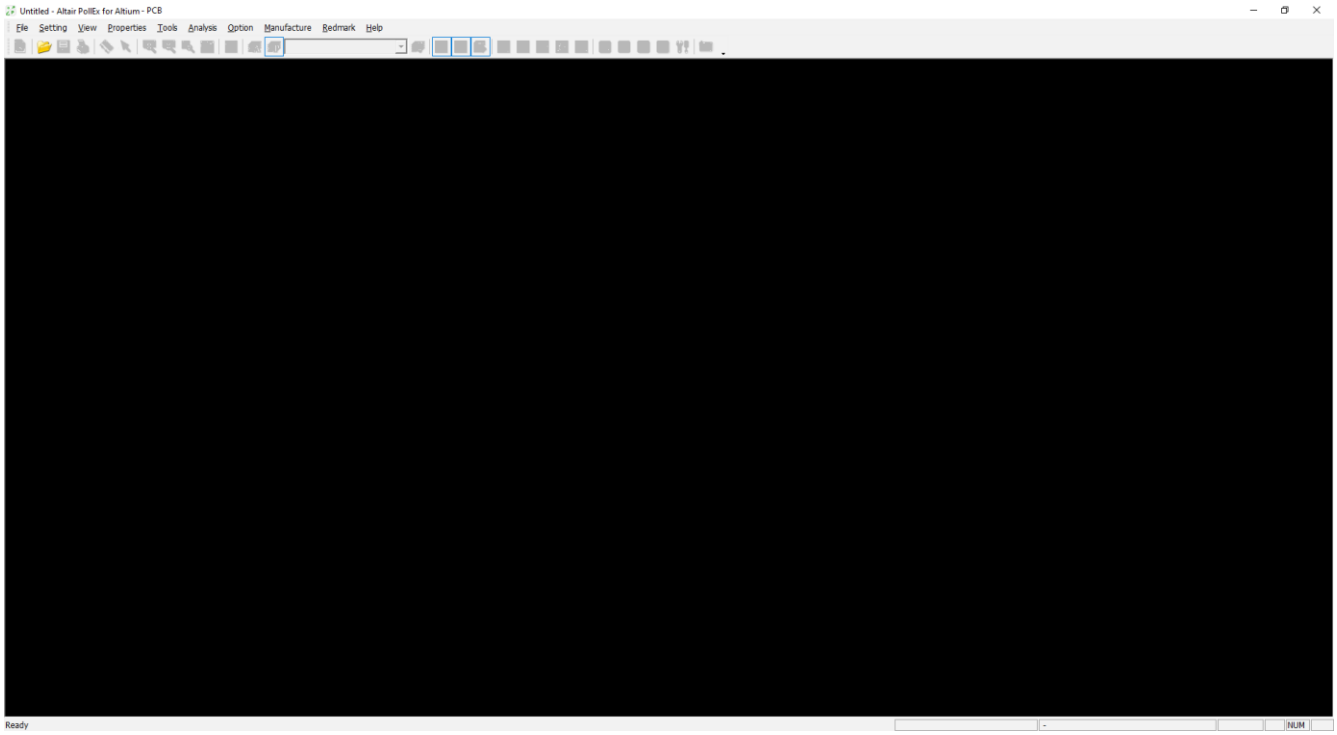
In addition to the **AltairPollEx** for Altium Software Extension, you will use the Altair PollEx for Altium software. You need to setup the environment the first time to use the Altair PollEx PCB.

Note: To view the "ProgramData" folder, you will need to go to the `Windows Control Panel`, select `Appearance and Personalization`, and click `Show hidden files and folders`. Select the View tab, make the changes shown below, and click OK.

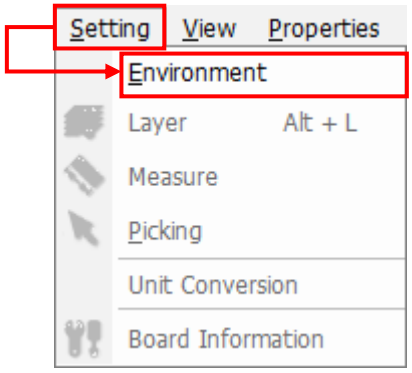


4. Altium Layer Setup in Altair PollEx PCB

4.1. From the start menu, click **Altair 2021.2 > PollEx PCB 2021.2**.



4.2. From the menu bar of the Altair PollEx PCB, click **Setting > Environment**.

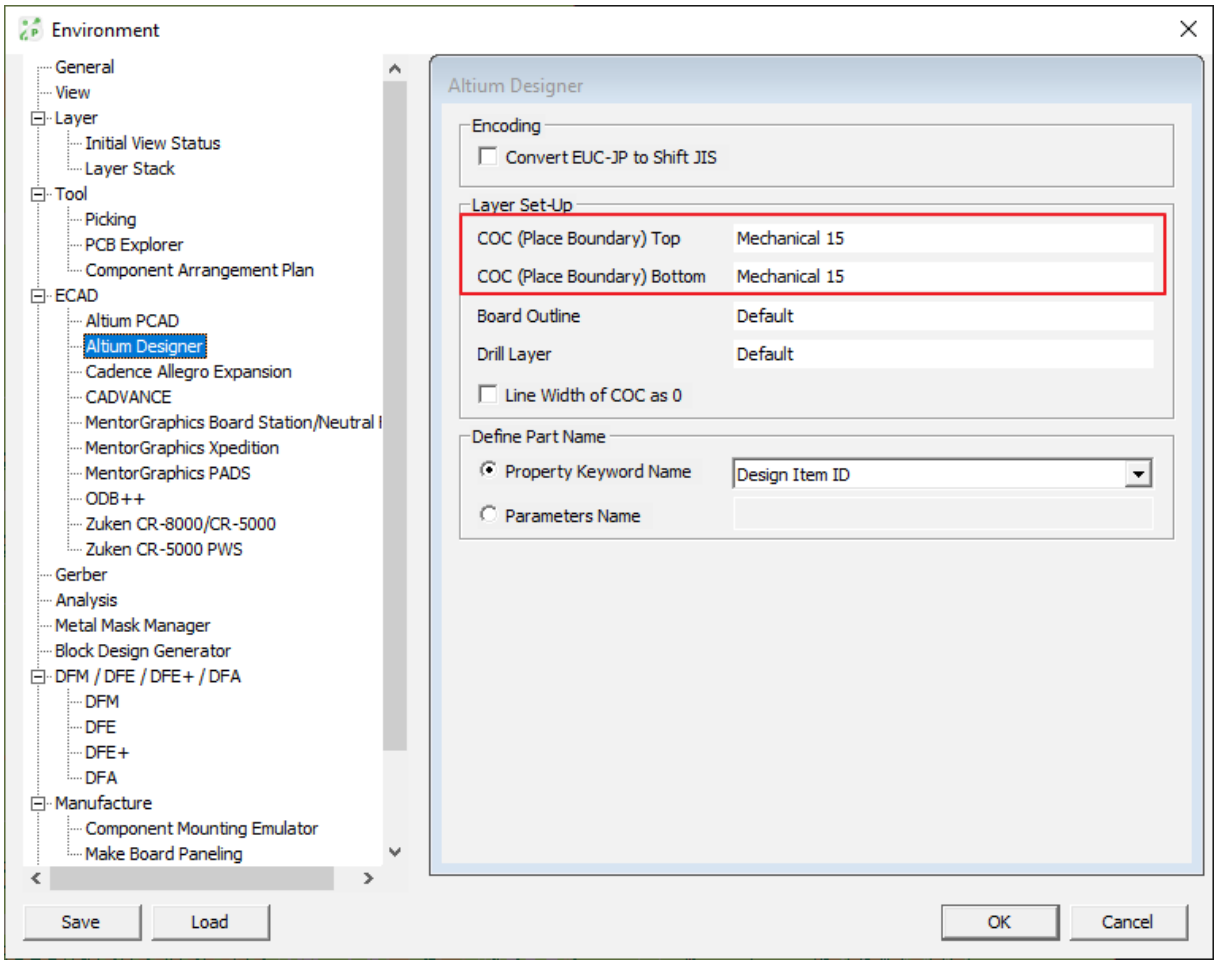


4.3. From the tree menu, click **ECAD > Altium Designer**.

4.4. Under the Layer Set-up menu, enter the following:

- For COC (Place Boundary) Top, enter `Mechanical 15`.
- For COC (Place Boundary) Bottom, enter `Mechanical 15`.

> **Note:** The COC (Component Overlapped Check) means a component boundary.



4.5. Click **OK**.

4.6. Close Altair PollEx PCB.

Note: The above environment setting in the Altair PollEx PCB is essential to import correct data into Altair PollEx. The layer names will vary depending on the Altium Designer Design Data.

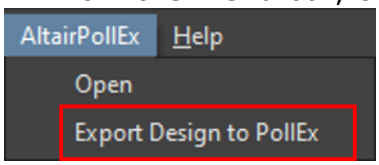
5. Open Altium Designer PcbDoc Design File

5.1. Open the file, `AltairPollExForAltium_A01.PcbDoc`,in Altium Designer. This file is located in `C:\ProgramData\altair\PollExForAltium\2021.2\Examples` directory.

5.2. Click the opened PcbDoc design sample in Altium Designer and you can see **AltairPollEx** extension toolbar added to the menu bar.

6. Convert PcbDoc Design File for PollEx

6.1. From the menu bar, click **AltairPollEx > Export Design to PollEx**.



6.2. Altair PollEx for Altium converts the Altium Designer PCB design (`*.PcbDoc`) into Altair PollEx database format (`*.pdbb`).

> **Note:**
> - Altair PollEx for Altium creates a project folder with the PCB design file name in the same directory of the Altium Designer PCB design file.
> - In the created folder, **Part** folder and **PDBB** file are created.
> - The Part folder is used for the Altair PollEx Solver products like SI and Thermal.

6.3. The converted PCB design (`*.pdbb`) opens in **Altair PollEx PCB**.

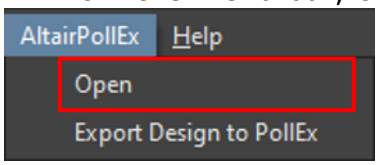
> **Note:** If a project folder exists when you click the **Export Design to PollEx** menu, you can only save the Altair PollEx file (`*.pdbb`).

6.4. Here you can either close the **Altair PollEx PCB** to go to step 8 or start evaluating PollEx using step 9.

7. Open PollEx PDBB File from Altium Designer

If a PDBB file is already created, you can easily open PollEx to view the PDBB file.

7.1. From the menu bar, click **AltairPollEx > Open**.



7.2. Saved Altair PollEx design data is opened in Altair PollEx PCB.

> **Note:** If the PDBB file is not available, you should convert PcbDoc design file into Altair PollEx file (`*.pdbb`).

8. Run Altair PollEx DFM

8.1. From the menu bar, click **Option > DFM > DFM Input**.

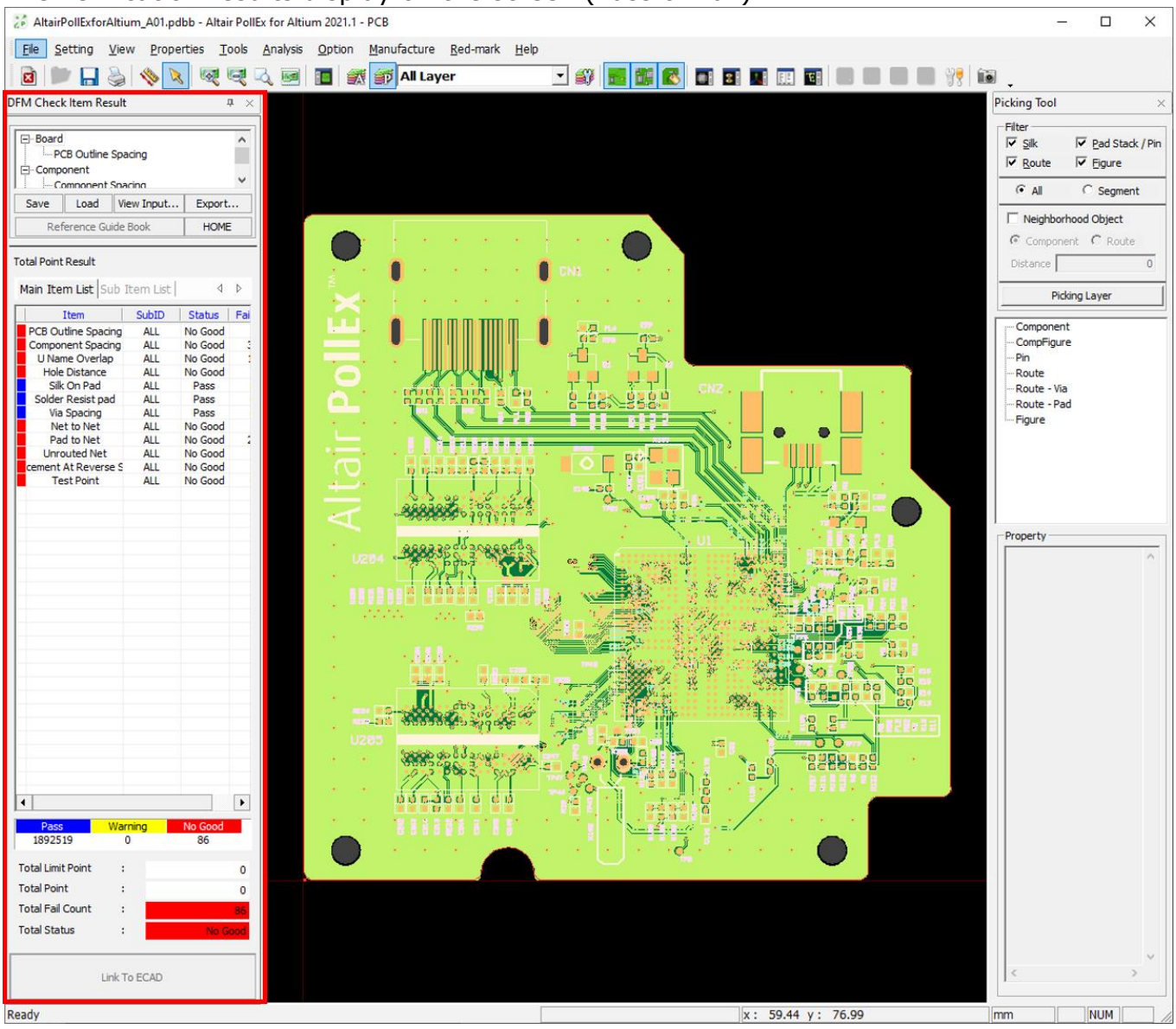
8.2. Click **Load** and select `DFM_Input.DFMI` file located at `C:\ProgramData\altair\PollExForAltium\2021.2\Examples\DFM`. This is a template file for DFM, and the same can be achieved by adjusting the settings manually for each of the DFM checks you wish to run.

8.3. Click **Start Checking** to run the verification.

PollEx DFM checks whether the design violates the rules.

8.4. Check results.

The verification results display on the screen (Pass or Fail).



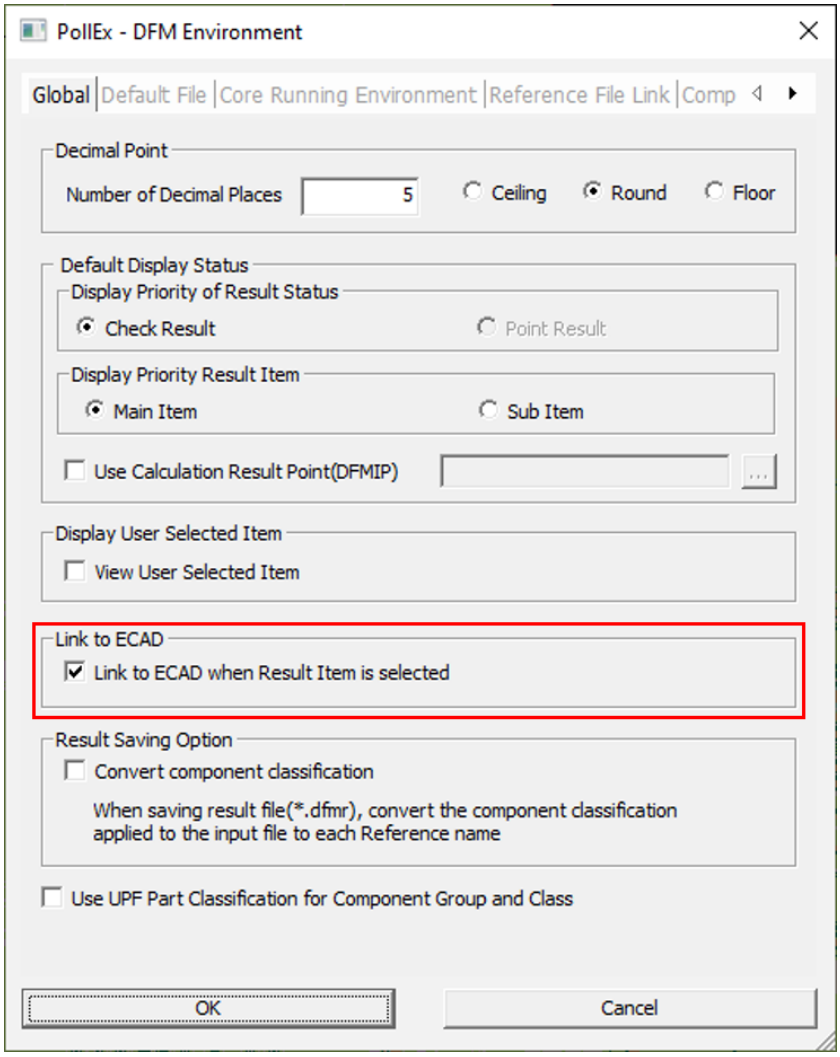
From here, you can use the DFM user guide to learn more about the settings needed to create your own rule checks. This can be found in the tutorial highlighted in the note above.

9. Link to ECAD (Cross-Probing)

When reviewing the results of the DFM checks, it might be necessary to cross probe the Altium PcbDoc files. The following steps will show you how to do that.

9.1. From the menu bar, click **Option > DFM > DFM Environment**.

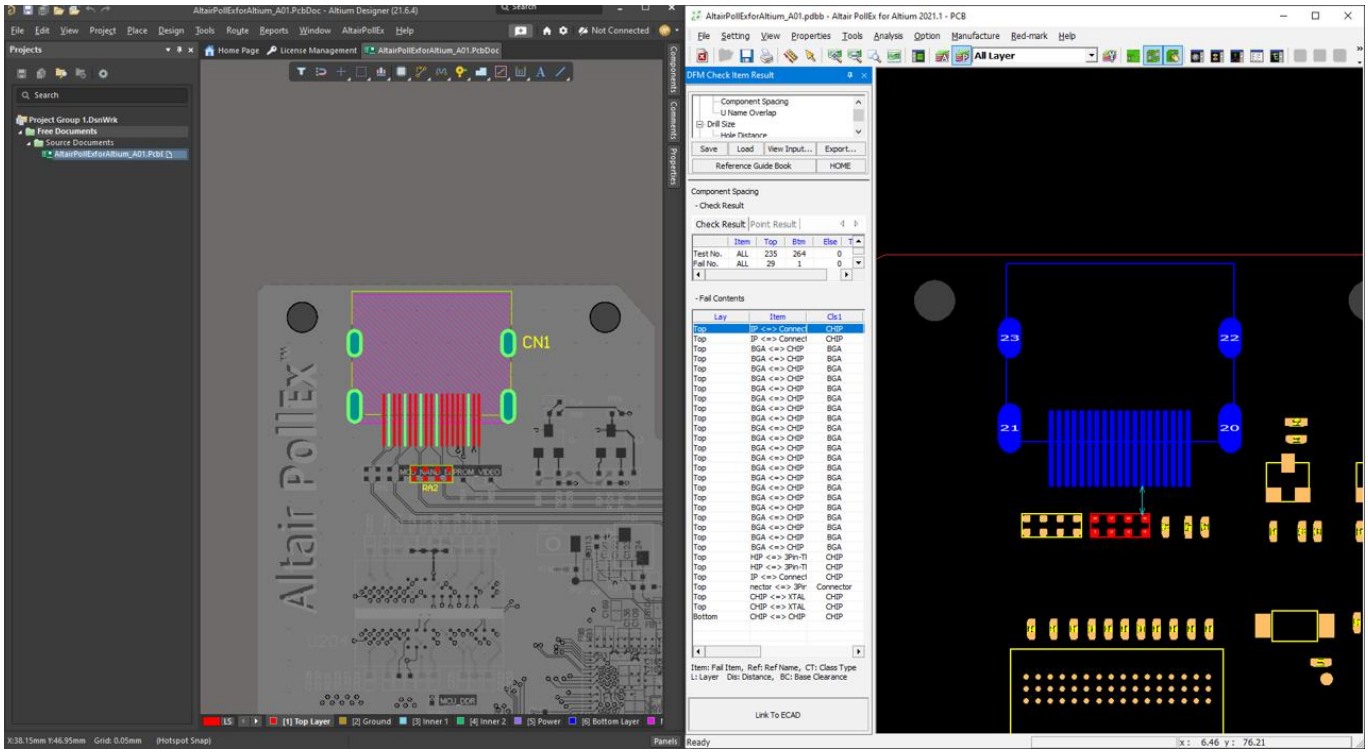
9.2. Enable the checkbox of the **Link to ECAD when Result Item is selected**.



Note: If the above checkbox is enabled, you do not need to click the **Link to ECAD** button in the Altair PollEx DFx (DFM and DFE).

9.3. Click **Ok** to close the DFM Environment dialog.

9.4. From the DFM Check Item Result, click **Component Spacing** and click the failure contents from the lists.



You can use the cross probe to check all of the failures in both PollEx PCB and in Altium. This is a useful tool for Design review meetings with people who are unfamiliar with the components or nets on the board.

10. Going Forward

Now that you have learned the basics of Altair PollEx for Altium, you can further your skills by working on the following tools:

- DFM
- DFE
- Signal Integrity
- Thermal Analysis

For more information, please visit the Altair Community.

Reminder: This is the limited version of PollEx for Altium. The full version includes more tools including a complete Verification checker, more Signal integrity analysis tools, and Power integrity analysis tools. If you are looking for the full PollEx experience, please visit https://www.altair.com/pollex-for-altium/.

For additional questions, please contact our PollEx for Altium support team at pfasupport@altair.com.