

How To Export To A *.dpx In Altium Design

1. Open Altium Design and navigate to your project. This project will contain a .PcbDoc which should show a final PCB.
2. Ensure the .PcbDoc is selected.
3. Select **File->Assembly Files -> Generate pick and place files.**
 - a. In the folder where all of the Altium files are stored, a CSV will be generated.
 - b. Move the file to the desktop for easier access for the future steps.
4. Open the CSV in Microsoft Excel.
5. When the file is first opened it will look like figure 1.
 - a. Edit the file to ensure it looks like figure 2.
 - b. Be sure to delete the alignment fiducials from the components, otherwise, the pick and place machine will try to place parts on the alignment fiducials.
 - c. Save the updated CSV.
6. Open a new File Finder Window and type "New Converter"
 - a. The first option will be a folder titled "New Converter 2.4.3", select this option
7. Open the application in the folder titled "ConvertTool-V2.4.3".
 - a. A new window will open.
8. In the new window, select open.
9. Navigate to where you stored the CSV.
10. Select the CSV file.
 - a. You will return to the original window.
11. In the various tabs, make the necessary edits needed.
12. When finished, click on "Convert".
 - a. This saves your changes, and converts the file into a .dpx.