How To Export To A *.csv 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 select
3. Select File->Assembly Files -> Generate pick and place files
4. In the folder which all the Altium files are stored in, it will create a CSV.
5. Upload the newly created CSV file to a thumbdrive.
6. Plug the thumb drive into the machine's laptop, drag and drop the file to the desktop.
7. Open a new File Finder window, navigate to the folder titled "Files" (located on the left hand side).
8. Drag and drop the CSV to the folder.
9. 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
10. Open the application in the folder titled "ConvertTool-V2.4.3.
    a. A new window will open.
11. In the new window, select open.
12. Navigate to where you stored the CSV (will be in the folder called Files).
13. Select the CSV file.
    a. You will return to the original window.
14. In the bottom right hand corner of the window, click "Convert"
15. Select Save, and a new window will open. This is where you will save it as a DPV. Use the same name as the CSV.
    a. It will ask you if you want to replace the csv, select yes.
16. Exit the program. When it asks you if you want to exit (even though you are editing), select yes.
17. Close the file finder window.