

## Procedure to convert V6 files to V5

It's important to keep in mind that this is a work-around. There is no native nice way to export a V5 file from V6 of EAGLE. With that said this process will get you 90-95% of the way there. The reason being that there are new features in V6 (ie text-justification) which did not exist in V5. When V5 runs into them it will flag an error, just keep clicking OK until the the script has finished running. The procedure is more verbose than it needs to be, but this is to make sure new users can follow it.

Best Regards,  
Jorge Garcia  
Cadsoft Support

1. Go to [www.cadsoftusa.com](http://www.cadsoftusa.com) → Downloads → User Language Programs(ULPs). Go ahead and download [export-board.ulp](#) and [export-schematic.ulp](#). Where you save them to is not critical as long as you know where they are. I will assume that the user decides to leave the files in the downloads folder.
2. In EAGLE V6 please open the file you wish to convert. EAGLE will open both the schematic and the board of interest.
3. Go to the schematic editor and type the word RUN in the command line followed by pressing the Enter key. This will bring up a small dialog with a bunch of files in it.
4. Browse to the downloads folder where you saved the export-schematic.ulp to. Click on the ULP and then click Open. If everything worked then it will appear as if nothing has happened.
5. The ULP that was run previously created a script in the same folder that contains the design files you're trying to export. It is through this script that we will import the schematic into V5. Now close EAGLE V6 and open EAGLE V5.
6. From the EAGLE control panel go File → New → Schematic. This will open a blank schematic.
7. Type the word SCRIPT and press the Enter key, this will bring up a file open dialog. Browse to the same directory that contains the V6 files you're trying to export.
8. You should find the temp.scr file. Click it once and then click Open. The schematic will now start to import, OK any errors that pop up. That's it.
9. Save your schematic(under a different name compared to the original) and close V5.
10. In EAGLE V6 please open the file you wish to convert. EAGLE will open both the schematic and the board of interest.
11. Go to the layout editor and type the word RUN in the command line followed by pressing the Enter key.
12. Again, browse to the downloads folder where you saved the export-board.ulp to. Click on the ULP and then click Open. If everything worked then it will appear as if nothing has happened.
13. Now close EAGLE V6 and open EAGLE V5. From the EAGLE control panel go File → New → Board. This will open a blank board.
14. Type the word SCRIPT and press the Enter key, this will bring up a file open dialog. Browse to the same directory that contains the V6 files you're trying to export.
15. You should find the temp.scr file. Click it once and then click Open. The board will now start to import, OK any errors that pop up. That's it.
16. Now save the board with same name and in the same location as the schematic file of Step 9. This will link the two files together.
17. Now click on the Switch to Schematic Icon(top toolbar 5th icon from the left). The schematic will open and hopefully both files are consistent if they're not you'll have to correct that.