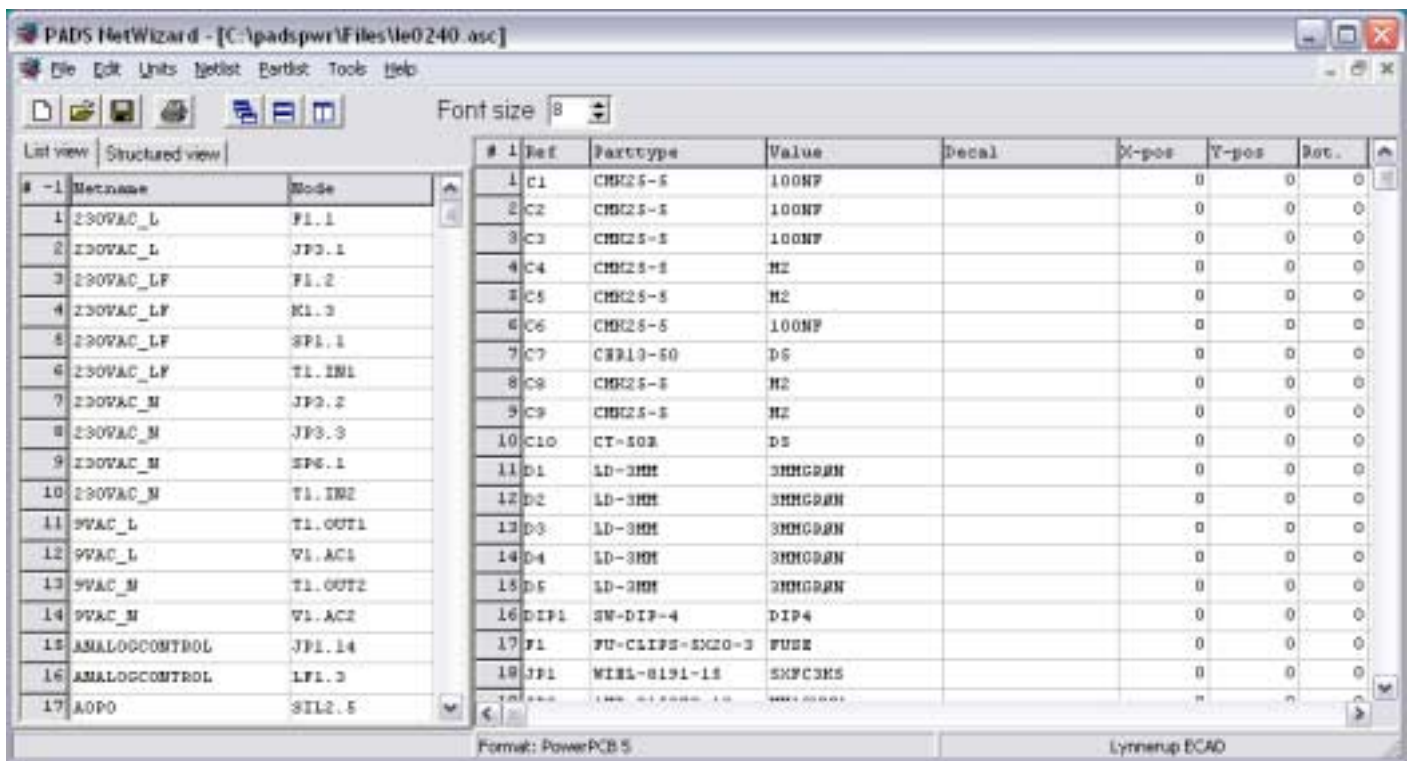
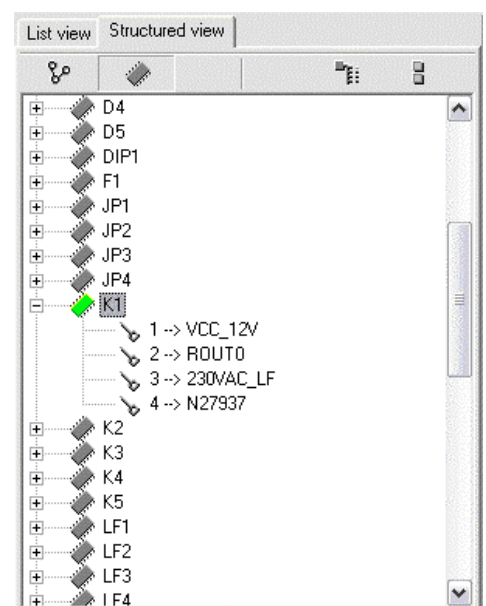
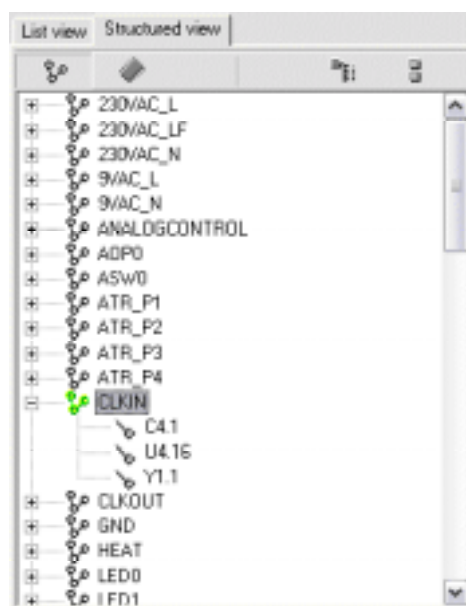


## PADS Netlist Wizard.

*PADS Netlist Wizard* is a full Windows multidocument application, that opens PADS ASC-files and let you **view**, **change** and **check** the part- and netlist. The userinterface is customizable – make columns that shows your custom attributes.



You can even have the netlist shown in a tree structure, ordered by nets or by parts: When ordered by parts, a parts connections are expanded in the netlist window, when the part is selected in the partlist.



You know the problems about customers parts- and netlists:

1. They use different partnames than you do. They even use the same name for parts that should have different footprints.
2. They use different pinnames/numbers than your parttypes.
3. Mountingholes, fiducials ect. Are missing in the part- and netslist.
4. Schematic-parts has less pins than the footprints.
5. Sometimes they only have a netlist. The partlist is missing.
6. They are not allways able to supply you with a PADS-format netlist.
7. The schematic-program generates a netlist that makes PowerPCB report errors.
8. Numerous schematicupdates forces you to do these tasks again and again to ensure as few layoutchanges as possible.

*Netlist Wizard* does this for you:

Ad. 1. Set parttype on all parts with the same Ref-letter, parttype, value , decal or other attribute you supply.

Ad 2. A batchmode pin-renamer for all parts of a parttype. Can even be set up to rename all the specified parttypes. (A definition can be made for each customer or job).

Ad 3. You can enter new parts in part- and netlist and keep them updated in future releases. See Ad. 8.

Ad 4. You can add pins to all parts of a parttype and select to connect to a specific net or the net of another pin on the part. And you can add parallel parts to a specific part or all parts of a parttype.

Ad 5. A partlist can be generated from the netlist.

AD 6. *Netlist Wizard* imports various netlistformats and will be extended when needed. The EDIF-interface lets you import designrules from ie. OrCAD and pass them allong to PowerPCB. You can save only PADS PowerPCB formats.

Ad 7. A built-in check-function that checks lots of things: Used any illegal letters? Are identifiers too long? Does netlist and partlist match up? Any two-pins parts with only one connection? Any reuse of ref-des or pin? Saves you from generating useless ECO-files, that you have to go all the way back to correct. Errors and warnings are colourmarked in the parts- and netlist.

Ad. 8. You can import the parttype definitions from earlier versions of the pcb. You can even have PADS Netlist Wizard save the changes in a database and apply these changes automatically to future revisions. The parttype-import function will also let you import parts not mentioned in the schematic: Mountingholes ect. and their netlistentries, so nothing gets lost inn the update.

**New function are added to PADS Netlist Wizard as customers want them.**