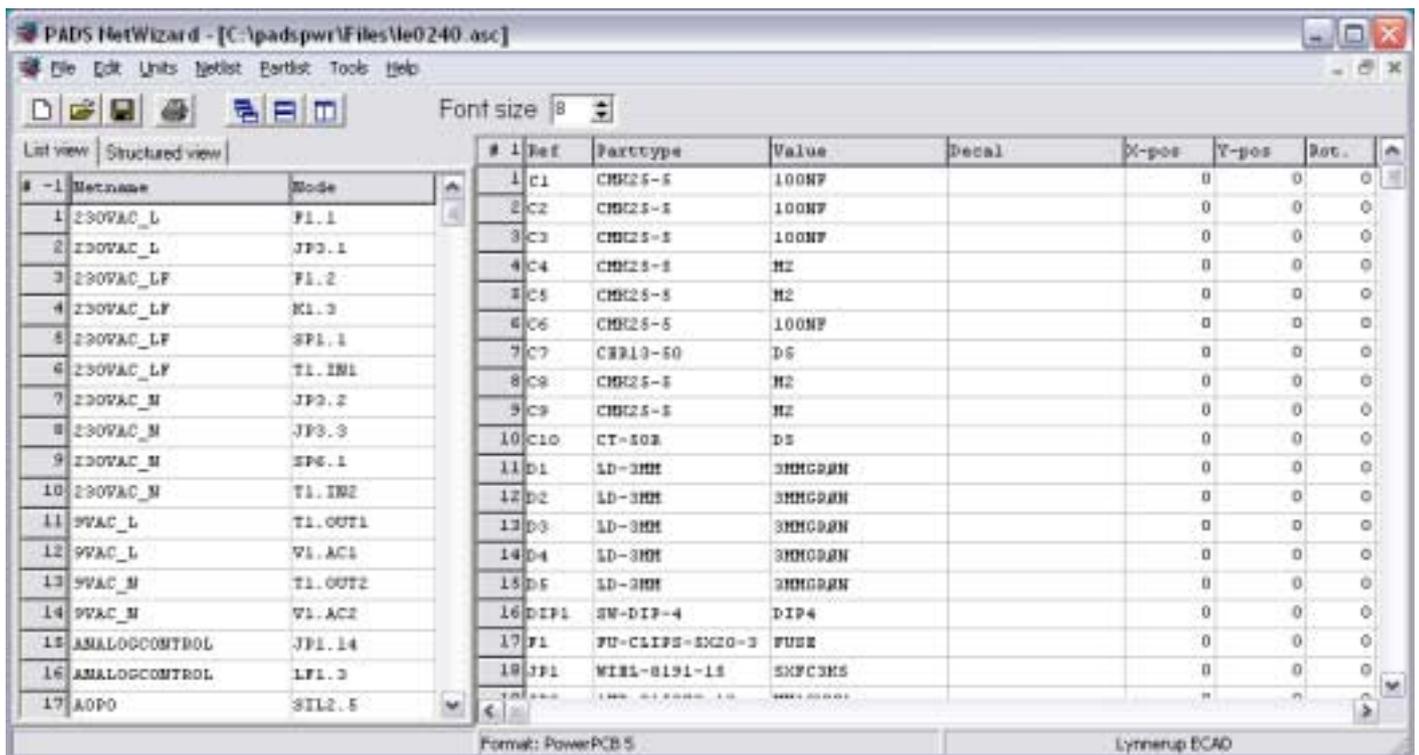


PADS Netlist Wizard.

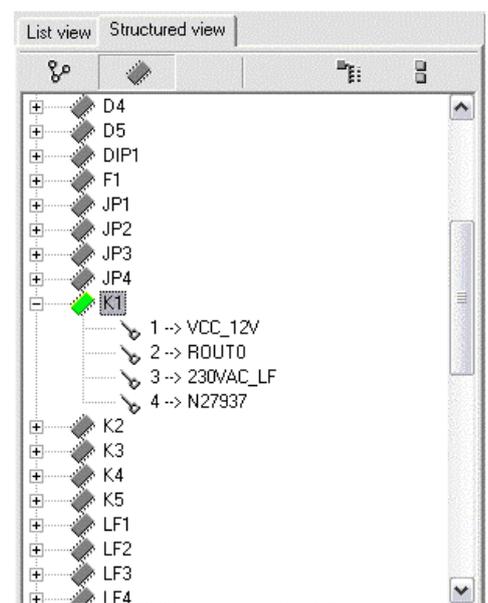
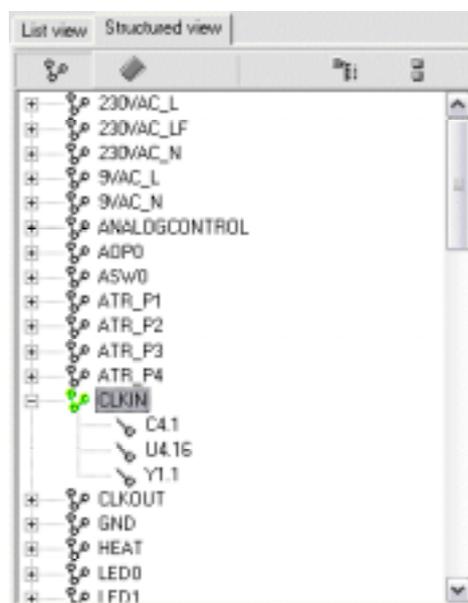
PADS Netlist Wizard is a full Windows multidocument application, that opens PADS ASC-files and let you **view**, **change** and **check** the part- and netlist. The userinterface is customizable – make columns that shows your custom attributes.



Format: PowerPCBS Lynnerup ECAD

#	Ref	Parttype	Value	Decal	X-pos	Y-pos	Rot.
1	C1	CH25-S	100NF		0	0	0
2	C2	CH25-S	100NF		0	0	0
3	C3	CH25-S	100NF		0	0	0
4	C4	CH25-S	N2		0	0	0
5	C5	CH25-S	N2		0	0	0
6	C6	CH25-S	100NF		0	0	0
7	C7	CH10-S0	D5		0	0	0
8	C9	CH25-S	N2		0	0	0
9	C9	CH25-S	N2		0	0	0
10	C10	CT-S0R	D5		0	0	0
11	D1	LD-3HH	3HHGDJN		0	0	0
12	D2	LD-3HH	3HHGDJN		0	0	0
13	D3	LD-3HH	3HHGDJN		0	0	0
14	D4	LD-3HH	3HHGDJN		0	0	0
15	D5	LD-3HH	3HHGDJN		0	0	0
16	DIP1	SW-DIP-4	DIP4		0	0	0
17	F1	FU-CLIPS-SG20-3	FUSE		0	0	0
18	JP1	WIRE-0191-15	SKPC3HS		0	0	0

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.