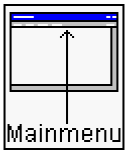
Design manager PCB elegance

The design manager of PCB elegance is the central tool to start the schematic editor, geometry editor, and the layout editor..

Getting started

An example of a new project/design can be found in the manual.

Make new design



Sub menu **File** menu item **New design**

In the next dialogbox the parameters of the new design can be edited.

In the **Design directory**, the new directory has to be filled in. In the **Design name** editbox the name of the design should be edited. Usually it is not necessary to change the next two editboxes **Schematic symbol library** directory and **Geometrie library directory**. In the **Top sheet name** editbox the first sheetname should be filled in. When all parameters are filled a new design will be created. The following directories/files in the **Design directory** are created.

File	<Design name>.dsn	
File	sch.ini	Settings schematic editor

The file sch.ini will be copied from the executables directory. (If exists)

Directory	backup	
Directory	pcb	layout file (*.pcb)
Directory	pcb\backup	Backup layout files (*.pcb)
Directory	pcb\gerber	Gerber/drill output files (*.ger)
Directory	pcb\hpgl	HPGL output file (*.hgl)
Directory	pcb\shapes	Local shape files (*.shp)
File	pcb\shapes\geom.ini	Settings geometry editor

The file geom.ini will be copied from the executables directory. (If exists)

Directory	pcb\shapes\backup	Backup local shape files (*.shp)
File	pcb\pcb.ini	Settings layout editor

The pcb.ini file will be copied from the executables directory. (If exists)

Directory	sch	Schematic files (*.sch)
Directory	sch\backup	Backup schematic files
File	sch\<top sheet name>.sch	Top sheet file
Directory	sym	Local (sheet)symbol files
Directory	sym\backup	Backup sym files

Directory structure design

Root directory design

<Design name>.dsn	Design settings
sch.ini	Settings schematic editor
component.txt	Bill Of Materials output file
<Design name>.bom	Bill Of Materials output file

Subdirectory **Backup**

Previous Bill Of Materials output files.

Subdirectory **pcb** (layout subdirectory)

Subdirectory backup

- Backup previous version layout file
- Backup layout file before today (<Design name>.1)
- Backup previous netlist file

<Design name>.pcb	Layout file
pcb.ini	Settings layout editor
<Design name>.net	Netlist file
<Design name>.neu	Neutral file (PCB testing)
gatepin.swp	Gate/pin swap file
gatepin.ban	Gate/pin swap back annotation file

pos_mils.txt	Component position file (mils)
pos_inch.txt	Component position file (inch)
pos_mm.txt	Component position file (mm)

Subdirectory **pcb\shapes** (local geometries subdirectory)

Subdirectory **backup**

Backup local geometry files

geom.ini	Settings geometry editor

Subdirectory **pcb\gerber** (gerber output subdirectory)

<Design name>.drl	Drill output file
drills.txt	Drill tool file
drill.rck	Drill tool file (binary)
gerber.txt	Aperture file gerber output files
layers.txt	Layer info file
Top.ger	Gerber output file top (Component side)
Bottom.ger	Gerber output file bottom (Solder side)
Inner1.ger	Gerber output file inner layer 1
Inner2.ger	Gerber output file inner layer 2
SolderMaskTop.ger	Gerber output file solder mask top
SolderMaskBottom.ger	Gerber output file solder mask bottom
PasteMaskTop.ger	Gerber output file paste mask top
PasteMaskBottom.ger	Gerber output file paste mask bottom
SilkScreenTop.ger	Gerber output file silkscreen top
SilkScreenBottom.ger	Gerber output file silkscreen bottom
BoardOutline.ger	Gerber output file board outline
Info.ger	Gerber output file info layer

Info2.ger	Gerber output file info layer2

Subdirectory	**pcb\hpgl**	(penplot output subdirectory)

Top.hgl	Penplot output file top (Component side)
Bottom.hgl	Penplot output file bottom (Solder side)
Inner1.hgl	Penplot output file inner layer 1
Inner2.hgl	Penplot output file inner layer 2
SolderMaskTop.hgl	Penplot output file solder mask top
SolderMaskBottom.hgl	Penplot output file solder mask bottom
PasteMaskTop.hgl	Penplot output file paste mask top
PasteMaskBottom.hgl	Penplot output file paste mask bottom
SilkScreenTop.hgl	Penplot output file silkscreen top
SilkScreenBottom.hgl	Penplot output file silkscreen bottom
BoardOutline.hgl	Penplot output file board outline
Info.hgl	Penplot output file info layer
Info2.hgl	Penplot output file info layer2

Subdirectory	**sch**	(schematic subdirectory)

Subdirectory **backup**

Backup previous version schematic files

*.sch files	Schematics
*.wir files	Link files to the layout editor
*.wtx files	Link files to the layout editor (Wires/busses)

Subdirectory	**sym**	(local symbols subdirectory)

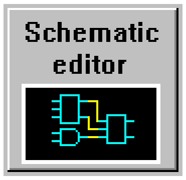
Subdirectory **backup**

Backup previous version local symbols files

*.sym Local (sheet)symbol files

Make new design

Open sheet

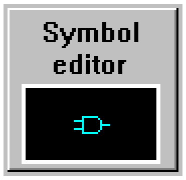


Press **Schematic editor** button

Press this button with the **left mouse button**, and the **Schematic editor** will be executed with the topsheet

Press this button with the **right mouse button**, and the **Schematic editor** can be loaded with a sheet, selected from the pulldown menu.

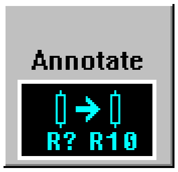
Edit symbol



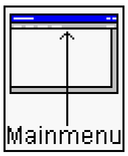
Press **Symbol editor** button

Press this button with the **left mouse button**, and the **Symbol editor** will be started.

Annotation



Press **Annotation** button



Sub menu **Edit** menu item **Annotation**

Annotation means numbering the component references who are named like (R?,C?,U?) automatically. After annotation the quotation character of the component references have been replaced by a number. The numbering of component references is done per sheet. The Top sheet will start with the number 100. For example resistors will start with R100 etc. The next (sub)sheet will start with the number 200. If the top sheet contains for example more then 100 resistor, the next (sub)sheet will start with the number 300. If possible do not place more than 100 component references of each family (resistors, capacitors) on each sheet.

Component references who do not have this quotation character will not change.

If there has been a small modification to a sheet, for example one resistor added, annotation will proceed as follows. This new resistor with the component reference R? will be renamed. The number that will replace the quotation character will be the highest not used number on this sheet. This means that resistors (numbers) that have been deleted in an earlier stage, will not be used again.

In the next dialogbox four annotation methods can be selected

<u>Restart annotation (Standard numbering)</u>

All the component references will be renumbered starting with one.

<u>Restart annotation (Numbering per sheet)</u>

All the component references will be renumbered, starting on a hundred per sheet. The resistors on the first sheet start with 100, and resistors on the seconds will start with 200, etc.

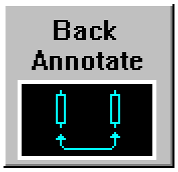
Appending annotation (Standard numbering)

In a existing design this annotation form will be used. Only the component references who are not numbered will get a new number. A new number means a number that has not be used before. Usually this will be a number greater than the highest number used.

Appending annotation (Numbering per sheet)

In a existing design this annotation form will be used. Only the component references who are not numbered will get a new number. A new number means a number that has not be used before. Usually this will be a number greater than the highest number used. On every sheet this new number will be the last number used plus one.

Back annotation



Press **Back annotation** button

Back annotation means the changes made with the gate and pin swaps will be reflected into the schematics. When the **Back annotation** function is executed, the file **pcb\gatepin.ban** will be red, and the necessary schematics will be modified.

For example: The two pins of a 7400 ttl device have been swapped by using the layout editor. After **Back annotation** the wires connected to those two 7400 pins in the schematics will be switched.

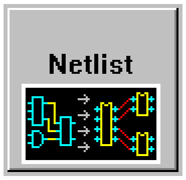
»

Gate/pin swap

»

Edit gate/pin swap in symbols

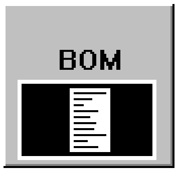
Create netlist



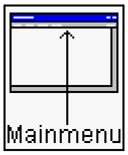
Press **Netlist** button

When this button is pressed the netlist will be calculated. This netlist consists of components, and the actual netlist. This netlist will be placed in the designs **pcb** subdirectory. Also the gate/pin swap info file **pcb\gatepin.swp** will be generated.

Bill Of Materials



Press **BOM** button



Sub menu **Edit** menu item **Bill Of Materials**

In the next dialogbox three Bill Of Materials methods can be selected

List of components

Every component will be listed on a line.

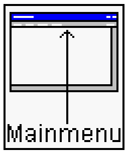
Bill of materials with component references listed

In this Bill Of Materials components are summed and listed.

Bill of materials without component references

In this Bill Of Materials components are summed and listed, but the component references are not listed.

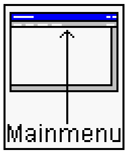
Check



Sub menu **Edit** menu item **Check schematics**

Checks the schematics of the design for errors.

Conversion ORCAD schematic/libraries



Sub menu **File** menu item

Convert ORCAD schematic

Convert ORCAD library

With these two functions ORCAD schematics and libraries can be converted to PCB elegance. ORCAD schematics and libraries being used in the windows version are not supported.

Conversion ORCAD schematic:

- After selection of the menu item the program continues with a dialogbox for the ORCAD schematic filename. Always select the top sheet, if there is a hierarchical structure for the schematics. The other sheets will be converted automatically.

  In the next dialogbox some parameters can be entered. If the project directory does not exist it will be created. The schematic(s) and symbols used will be copied inside this project directory.

  A provision has been made to use the geometries/shapes names attached to the components. ORCAD uses partfields for specifying such geomwtries/shapes attributes. The partfield nummer can be selected in the dialogbox.

  In most cases the geometries/shapes of ORCAD and PCB elegance do not match. In the "Geometry conversion file" field a filename can be placed, which will be used to translate the geometries.

  This file consists of a number of lines. Each line contains two strings. The first string is the geometry name used in the ORCAD file, and the second string is the PCB elegance geometry.

  Empty lines or lines starting with a ';' will be ignored.

  Now the conversion starts with reading the ORCAD configuration file **sdt.cfg**. This configuration file (Which libraries to be used) should be in the same directory as the schematic file. If the sdt.cfg file does not exist you should create this file, otherwise no symbol can be found.

Example ORCAD sdt.cfg

Conversion ORCAD library:

- After selection of the menu item the program continues with a dialogbox for the ORCAD library filename. In the next dialogbox the directory of the converted library can be entered.

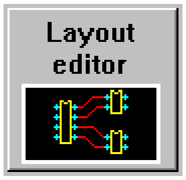
Orcad sdt.cfg example

```
{ OrCAD/SDT IV  Configuration File }
PDRV    = 'D:\ORCADESP\DRV\'
PSCH    = ''
PLIB    = 'D:\ORCADESP\SDT\LIBRARY\'
DD      = 'VGA640.DRV'
PRD     = ''
PLD     = 'HP.DRV'
LIB     = 'TTL.LIB'
LIB     = 'CMOS.LIB'
LIB     = 'DEVICE.LIB'
```

If necessary such a sdt.cfg file can be created. The lines containing the strings PLIB and LIB are important, and the first line for identification.

The PLIB string will contain the library directory, and the LIB strings the individual libraries in the library directory.

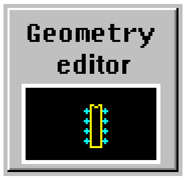
Layout editor



Press **Layout editor** button

When this button is pressed the layout editor will be started with the layout of the current design.

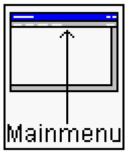
Geometry editor



Press **Geometry editor** button

When this button is pressed the geometry editor will be started.

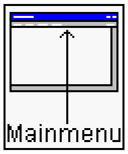
Library manager symbols



Sub menu **Library manager symbols**

Start the library manager for the schematic symbols. The symbol libraries are stored in the **lib** subdirectory.

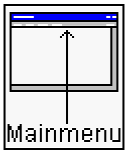
Library manager geometries



Sub menu **Library manager geometries**

Start the library manager for the geometries. The geometry libraries are stored in the **shplib** subdirectory.

Copy symbols/geometries locally



Sub menu **File** menu item **Copy symbols/geometries local**

All the symbols and geometries used in this project will be copied to the local directories in the project. This can be usefull to interchange a project with someone else, because the whole directory can be copied, inclusive all symbols and geometries.