

## Formats

With our CAM Systems we are able to handle all current Data formats.

Beyond we hold CAD Tools ready to take over your data directly from your systems.

In details this are

Type	Name	from	
CAD	Eagle	<a href="http://www.cadsoft.de">www.cadsoft.de</a>	
	Target	<a href="http://www.ibfriedrich.com">www.ibfriedrich.com</a>	
	GC PrevuePlus	<a href="http://www.graphicode.com">www.graphicode.com</a>	
	Altium Designer	<a href="http://www.altium.com">www.altium.com</a>	
	Pulsonix	<a href="http://www.pulsonix.com">www.pulsonix.com</a>	
	Ultiboard	<a href="http://www.ni.com">www.ni.com</a>	
Viewsystems	GC Prevue		
PCB Data	Design	ODB++	Version 6.1
		Gerber 274	
		Gerber 274x	
	Mechanic	Excellon	
		Sieb & Meyer 2000/3000	

To avoid further inquiries and thus unpleasant disturbances for you complete and clear data are necessary.

In the following we arranged some information to the different data formats for you.

**Eagle**

The Eagle CAD Software has a modul to export PCB datas. There is a configuration for the export of the different design- and mechanical-layers.

If we don't get any further information with your order we use the following setting.

Eagle Nr	Eagle Name	Description	WEdirekt Name
1 17 18	Top Pads Vias	Top-Layer	VS
16 17 18	Bottom Pads Vias	Bottom-Layer	RS
29	tStop	Soldermask Top	LSMVS
30	bStop	Soldermask Bottom	LSMRS
21 25	tPlace tNames	Silkscreen Top	SEVS
22 26	bPlace bNames	Silkscreen Bottom	SERS
31	tCream	Pastemask Top	PASTE-VS
32	bCream	Pastemask Bottom	PASTE-RS
44	Drills	DK-Drills	BOHR1
45	Holes	NDK-Drills	BOHR2
2 17 18	Route2 Pads Vias	Innerlayer 2	L2A00
3 17 18	Route3 Pads Vias	Innerlayer 3	L3A00
4 17 18	Route4 Pads Vias	Innerlayer 4	L4A00
5 17 18	Route5 Pads Vias	Innerlayer 5	L5A00
20	Dimension	PCB-Contour	KONTUR

Eagle is using among other things octogonal pads which can lead to a false interpretation in other systems. To avoid those difficulties we use round pads instead.

**Target**

We planned the Layer allocation list in the target System as follows. Please send us with your data expenditure information if there were differences.

Target Nr	Target Name	Description	WEdirekt Name
16 15 14	Copper top Deletion top Area top	Top-Layer	VS
2 1 0	Copper bottom Deletion bottom Area bottom	Bottom-Layer	RS
18	Solder mask top	Solder Mask Top	LSMVS
4	Solder mask bottom	Solder Mask Bottom	LSMRS
21	Position top	Position Top	SEVS
7	Position bottom	Position Bottom	SERS
19	Solderpaste top	Solder Paste Top	PASTE-VS
5	Solderpaste bottom	Solder Paste Bottom	PASTE-RS
24	Drill hole	PTH/NPTH-Drills	BOHR1
13 12 11	Other Deletion inside Area inside	Innerlayer 2	L2A00
10 9 8	Other Deletion inside Area inside	Innerlayer 3	L3A00
23	Outline	LP-Contour	KONTUR

The copper layer in Target always consists of three layers.  
 The layer copper contains the tracks.  
 The Layer area contains ground areas.  
 The Layer delete contains surfaces around the tracks (to check distances).

## Altium Designer

We planned the Layer allocation list in Altium Designer System as follows. Please send us with your data expenditure information if there were differences.

Layer Name Extension	Description	WEdirekt Name
G1, G2, etc.	Mid-Layer 1, 2 , etc.	L2, L3, etc.
GBL	Bottom Layer	RS
GBO	Bottom Overlay	SERS
GBP	Bottom Paste-Mask	PASTE-RS
GBS	Bottom Solder Mask	LSMRS
GD1, GD2, etc.	Drill-Drawing	
GG1, GG2, etc	Drill Guide	
GKO	Keep Out Layer	
GM1, GM2, etc	Mechanical Layer 1, 2, etc.	
GP1, GP2, etc.	Internal Plane Layer 1, etc.	
GPB	Pad Master Bottom	
GPT	Pad Master Top	
GTL	Top Layer	VS
GTO	Top Overlay	SEVS
GTP	Top Paste Mask	PASTE-VS
GTS	Top Solder Mask	LSMVS
P01, P02, etc	Gerber Panels	
APR	Aperture-File	Aperturetable (RS274X)
APT	Aperture-File	Aperturetable (RS274D)

You can send us your projectfile (xxx.PrjPCB) or your pcb-file (xxx.PcbDoc).

**ODB++**

ODB++ makes an optimized data exchange possible between Design and manufacturing.

It makes possible shorter turn-around times because all information for the printed circuit board for manufacturers are clearly defined.

It improves the quality by the avoidance of data exchange errors. By the clear definition no different interpretations are possible. Data formats with different dialects, like e.g. RS274X, can on different CAM systems cause different results. Thus substantial damage can develop.

With the expenditure of the layout data we get smaller data sets develop because filling of surfaces with vectors are avoided there.

ODB++ is a fully expandable ASCII data format with the following advantages :

- all datas are included in on file. Nothing will be lost
- There is a exact description of the graphical datas. No unnecessary filling of copper areas or pads with special forms which has to be exchange by the pcb manufacturer.
- as many as desired attributes for the elements of the printed circuit board can assign to describe these elements
- ODB++ contains a CAD Netlist description on the basis those that Printed circuit board manufacturer can examine the electrical connections during the production process for their agreement with the original Design
- ODB++ contains a position table in that the layers names, the kinds of the layer the polarity as well as the sequence of the situations in the printed circuit board is defined
- A layer matrix that the parts list and the total structure of the plate defines
- ODB++ contains for the drill- and rout-layers the allocation which drill contact which layer
- Graphic notes can be attached according to the kind of a post it.

**Gerber 274**

The Gerber format was originally used for the control of photo plotters. There for between the exposure light and the photosensitive film an aperture is brought in which has different form and size depending of the requirements. Open and closing the shutter and moving the table the image is brought on the film. The photo plotters are today replaced by laser plotters.

The Gerber format is a variant of the conventional numerical control format. Of conventional Numerical control formats, like e.g. drilling data, it differs only by the selecting of d-codes. The data are arranged thereby in blocks what is a combination of the commands for the aperture selection, shutter mode (shutter open/close) and movement (X- Y-coordinates) The data are block-by-block processed.

For defining the apertures on the CAM-System an aperture list will be necessary which described the form and size of the apertures. Um die Blenden am CAM-System definieren zu können wird eine Blendenliste benötigt die die Form und Größe der Blenden beschreibt. This must be usually entered manually into the CAM-System. This represents a substantial expenditure for the printed circuit board manufacturer.

Example for an aperture file

D11	round	4
D12	round	8
D13	square	4
D14	square	8

Example for a Gerber file

```
G54D10*
X0000Y0000D02*
X0000Y1000D02*
X1000Y1000D02*
X1000Y0000D02*
X0000Y0000D02*
```

<u>Data Type</u>	<u>Description</u>
Designlayers	one File per Layer
Contour	one File for the Single-PCB / Subpanel Only contour lines with minimal measures (measures for the outline, drill to contour, placement of the Single-PCB in the Subpanel)
Drill Datas	one File for PTH/NPTH/sequentiel PTH Drill Diameter defined in the Fileheader Format should be Excellon or Sieb & Meyer
Aperture Description	please add the aperture table to the data record the table should define the d-codes with form and size
Compression	all datas specified above should be summarized in a compressed ZIP File. The name of the file should correspond to your designated LP number in your order

**Gerber 274x**

The extended Gerber Format (RS274X) an extension of the standard Gerber Format. Contrary to the standard Gerber format an aperture list is not necessary list, since the aperture information is contained already in the data file.

Example for an Extended Gerber-File

```
%ADDD11,C0.004%*
%ADDD12,C0.008%*
%ADDD13,S0.004%*
%ADDD14,S0.008%*
G54D10*
X0000Y0000D02*
X0000Y1000D02*
X1000Y1000D02*
X0000Y1000D02*
X0000Y0000D02*
```

<u>Data Type</u>	<u>Description</u>
Designlayers	one File per Layer
Contour	one File for the Single-PCB / Subpanel Only contour lines with minimal measures (measures for the outline, drill to contour, placement of the Single-PCB in the Subpanel)
Drill Datas	one File for PTH/NPTH/sequentiel PTH Drill Diameter defined in the Fileheader Format should be Excellon or Sieb & Meyer
Aperture Description	please add the aperture table to the data record the table should define the d-codes with form and size
Compression	all datas specified above should be summarized in a compressed ZIP File. The name of the file should correspond to your designated LP number in your order

## Layerdescription

Please use only clear names in your Datas. Beneath you will find the common used and the internally used by Würth Elektronik Nomenclature.

