

1 - Board File Format

Board Files Format:

1 - Board File Format.....	1
1.1 - General Informations:.....	1
1.2 - Layer numbering:.....	2
1.3 - First line of description:.....	3
1.4 - \$GENERAL.....	3
1.5 - \$SHEETDESCR.....	3
1.6 - \$SETUP block:.....	3
1.7 - \$EQUIPOT.....	5
1.8 - \$MODULE.....	5
1.8.1 - General description:.....	6
1.8.2 - Field Description:.....	6
1.8.3 - Drawings:.....	7
1.8.4 - Pad Descriptions:.....	7
1.8.5 - \$SHAPE3D.....	7
1.8.6 - \$PAD.....	8
1.9 - Graphic items:.....	8
1.9.1 - \$DRAWSEGMENT.....	9
1.9.1.1 - Line:.....	9
1.9.1.2 - Circle:.....	9
1.9.2 - Arc:.....	9
1.9.3 - \$TEXTPCB.....	9
1.9.4 - \$MIRE.....	10
1.9.5 - \$COTATION.....	10
1.10 - Track, vias and Zone section:.....	10
1.10.1 - \$TRACK.....	10
1.10.2 - \$ZONE.....	11
1.10.3 - \$CZONE_OUTLINE.....	11
1.11 - \$EndBOARD.....	13

1.1 - General Informations:

- Board file (*.brd files) are in ASCII format.
- Dimensions are in 1/10000 inch, except for the page size (in 1/1000 inch).

First line is something as:

PCBNEW-BOARD Version 0 date 5/1/2005-14:45:23

All the following descriptions are like this:

\$DESCRIPTION

some data

...

\$endDESCRIPTION

Example:

\$GENERAL

Ly 1FFF8001

Links 66

NoConn 0

Di 24940 20675 73708 40323

Ndraw 16
 Ntrack 267
 Nzone 1929
 Nmodule 29
 Nnets 26
 \$EndGENERAL

\$SHEETDESCR
 Sheet A4 11700 8267
 Title ""
 Date "23 feb 2004"
 Rev ""
 Comp ""
 Comment1 ""
 Comment2 ""
 Comment3 ""
 Comment4 ""
 \$EndSHEETDESCR

1.2 - Layer numbering:

Tracks and other items (texts, drawings ...) use one layer.

Pads and vias use several layers.

There are 16 copper layers and 13 technical layers.

The *layer* parameter used in descriptions has the value:

value	layer name	
0	Copper layer	"Copper" layers
1 to 14	Inner layers	
15	Component layer	
16	Copper side adhesive layer	Technical layers
17	Component side adhesive layer	
18	Copper side Solder paste layer	
19	Component Solder paste layer	
20	Copper side Silk screen layer	
21	Component Silk screen layer	
22	Copper side Solder mask layer	
23	Component Solder mask layer	
24	Draw layer (Used for general drawings)	
25	Comment layer (Other layer used for general drawings)	
26	ECO1 layer (Other layer used for general drawings)	
27	ECO2 layer (Other layer used for general drawings)	
28	Edge layer. Items on Edge layer are seen on all layers	
29	Not yet used	
30	Not yet used	
31	Not yet used	

Mask layer:

Sometimes, a *mask layer* parameter is used.

It is a 32 bits mask used to indicate a layer group usage (0 up to 32 layers).

Board File format.**PCBNEW****update: feb. 07 2009**

A *mask layer* parameter is given in *hexadecimal form*.

Bit 0 is the copper layer, bit 1 is the inner 1 layer, and so on...(Bit 27 is the Edge layer).

Mask layer is the ORed mask of the used layers

1.3 - First line of description:**Format:**

PCBNEW-BOARD Version <version number> date <date>-<time>

Date and time are useful only for information (not used by pcbnew).

1.4 - \$GENERAL

This data is useful only when loading file.

It is used by Pcbnew for displaying activity when loading data.

\$GENERAL	Start description
Ly 1FFF8001	Obsolete (used for old pcbnew compatibility)
Links 66	Total number of connections
NoConn 0	Remaining connections
Di 24940 20675 73708 40323	Bounding box coordinates: X_start Y_start X_end Y_end
Ndraw 16	Number of draw items like eged segments, texts...
Ntrack 267	Number of track segments
Nzone 1929	Number of zone segments
Nmodule 29	Number of modulss
Nnets 26	Number of nets
\$EndGENERAL	End description

1.5 - \$SHEETDESCR

This the page size and texts.

\$SHEETDESCR	Start description
Sheet A4 11700 8267	<Page size> X_size Y_size in mils (1/1000 inch)
Title ""	Title text
Date "23 feb 2004"	Date text
Rev ""	Revision text
Comp ""	Company name text
Comment1 ""	Comment text, line 1
Comment2 ""	Comment text, line 2
Comment3 ""	Comment text, line 3
Comment4 ""	Comment text, line 4
\$EndSHEETDESCR	End description

1.6 - \$SETUP block:

This data bock is used for design settings

This is useful only for board edition.

Example:

\$SETUP

Board File format.**PCBNEW****update: feb. 07 2009**

InternalUnit 0.000100 INCH
 Layers 2
 Layer[0] Cuivre signal
 Layer[15] Composant signal
 TrackWidth 250
 TrackWidthHistory 25
 TrackWidthHistory 170
 TrackWidthHistory 250
 TrackClearence 110
 ZoneClearence 150
 DrawSegmWidth 150
 EdgeSegmWidth 50
 ViaSize 600
 ViaDrill 250
 ViaSizeHistory 600
 MicroViaSize 200
 MicroViaDrill 80
 MicroViasAllowed 0
 TextPcbWidth 170
 TextPcbSize 600 800
 EdgeModWidth 150
 TextModSize 600 600
 TextModWidth 120
 PadSize 1500 2500
 PadDrill 1200
 AuxiliaryAxisOrg 29500 55500
 \$EndSETUP

\$SETUP	Start block "SETUP"
InternalUnit 0.000100 INCH	Internal unit for Pcbnew, all coordinates are in this unit
Layers 2	Number of layers (2 = double sided board) must be 1 to 16
Layer[0] Cuivre signal	layer name and type name = name given to the layer by the user (here: "cuivre") type = signal (not current used in Pcbnew)
Layer[15] Composant signal	
TrackWidth 250	Current track width
TrackWidthHistory 170	Last used track widths
TrackWidthHistory 250	
TrackWidthHistory 400	
TrackClearence 100	Isolation for DRC (Design rules check)
ZoneClearence 200	Isolation used in zone filling
DrawSegmWidth 120	Current segment width for drawings on technical layers
EdgeSegmWidth 120	Current segment width for drawings on "edge layer"
ViaSize 700	Current via size
ViaDrill 250	Via drill for this board
ViaSizeHistory 450	Last used via sizes
ViaSizeHistory 650	
ViaSizeHistory 700	
TextPcbWidth 120	Current text width for texts on copper or technical layers. This is not for text on footprints
TextPcbSize 600 600	Current text X Y size
EdgeModWidth 120	Current Segment width for footprint edition

Board File format.**PCBNEW****update: feb. 07 2009**

TextModSize 120 600	Current text XY size for texts for footprint edition
TextModWidth 120	Current text width for texts for footprint edition
PadSize 700 700	Current X Y pad size (footprint edition)
PadDrill 320	Current pad drill
AuxiliaryAxisOrg 0 0	Auxiliary axis position (Auxiliary axis is the reference coordinate (0 0 coordinate) for EXCELLON drilling files)
\$EndSETUP	End block "SETUP"

1.7 - \$EQUIPOT

\$EQUIPOT describes a net name.

\$EQUIPOT	Start block
Na 2 "N-000026"	Na <internal net number> « net name »
St ~	
\$EndEQUIPOT	End block

Note1:

Internal net number is an arbitrary number.
It is computed by Pcbnew when compiling netlist.

Note2:

Net 0 is not a real net.
Net 0 is the net number used internally by Pcbnew for all the no connected pads.

Example:

```

$EQUIPOT;
Na 0 ""
St ~
$EndEQUIPOT$EQUIPOT
Na 1 "DONE"
St ~
$EndEQUIPOT
$EQUIPOT
Na 2 "N-000026"
St ~
$EndEQUIPOT
$EQUIPOT
Na 3 "TD0/PROG"
St ~
$EndEQUIPOT

```

1.8 - \$MODULE

Description =start by:

\$MODULE <module name>

And ends with

\$EndMODULE <module name>

Module description has four sections:

1. General description (fixed size)
2. Field description (variable size)
3. Drawing description (variable size)
4. Pad description. (variable size)

5. 3D shape informations.

Note:

All coordinates are relative to the module position.

Its means the coordinates of segments, pads, texts ... are given for a module in position 0, rotation 0.

If a module is rotated or mirrored, real coordinates must be computed according to the real position and rotation.

1.8.1 - General description:

\$MODULE bornier6	\$MODULE <module lib name>
Po 62000 30500 2700 15 3EC0C28A 3EBF830C ~~	Po Xpos Ypos Orientation(0.1deg) Layer TimeStamp Attribut1Attribut2 Attribut1 = ~or 'F' for autoplacement (F = Fixed, ~= moveable) Attribut2 = ~or 'P' for autoplacement (P = autoplaced)
Li bornier6	Li <module lib name>
Cd Bornier d'alimentation 4 pins	Cd comment description (displayed when browsing libraries)
Kw DEV	Kw Keyword1 Keyword2 ... (for footprint selection by keywords)
Sc 3EBF830C	Sc TimeStampOp
Op 0 0 0	Op <rotation cost 90 deg> <rotation cost 180 deg> for auto place. rotation cost = 0 (no rotation allowed) to 10 (null cost)

Note:

Usually, components are on layer 15 (*component layer*) or 0 (*copper layer*).

If the component is on layer 0, it is "mirrored". The "mirror axis is the X axis

1.8.2 - Field Description:

There are 2 to 12 fields

Field 0 = component reference (U1, R5 ...) (required)

Field 1 = component value (10K, 74LS02 ...) (required)

Other fields (optional) are comments.

Format:

T<field number> <Xpos> <Ypos> <Xsize> <Ysize> <rotation> <penWidth> N <visible> <layer> "text"

Field	Units	Meaning
field number	enumeration	0=>reference, 1=>value, etc.
Xpos	tenths of mils (.0001 inches)	The horizontal offset relative to the module's overall position
Ypos	tenths of mils (.0001 inches)	The vertical offset relative to the module's overall position
Xsize	tenths of mils (.0001 inches)	The horizontal size of the character 'M'
Ysize	tenths of mils (.0001 inches)	The vertical size of the character 'M'
rotation	tenths of degrees	Angular rotation from horizontal, counterclockwise
penWidth	tenths of mils (.0001 inches)	Width of the pen used to draw characters
N	none	flag for the parser?

Board File format.**PCBNEW****update: feb. 07 2009**

visible	boolean	I=> invisible, V=> visible
layer	enumeration	see layer numbers above

Examples:

T0 500 -3000 1030 629 2700 120 N V 21 "P1"	T0 => reference
T1 0 3000 1201 825 2700 120 N V 21 "CONN_6"	T1 => value

1.8.3 - Drawings:

Tells how to draw module shape.

Drawings are segment, circle, arc.

DS -6000 -1500 -6000 1500 120 21	DS is a D raw S egment DS Xstart Ystart Xend Yend Width Layer
DS 6000 1500 6000 -1500 120 21	An other Draw Segment

Other Drawings are:

DC ox oy fx fy w	DC is a D raw C ircle DC Xcentre Ycentre Xpoint Ypoint Width Layer
DA x0 y0 x1 y1 angle width layer	DA is a D raw A rc X0,y0 = Start point x1,y1 = end point

1.8.4 - Pad Descriptions:

All the pads of this footprint are listed here (Many \$PAD/\$EndPAD sections here)..

See \$PAD description.

1.8.5 - \$SHAPE3D

3D shape informations:

The real shape description is a vrml file, build by **Wings3d**.

This shape can be scaled, moved and rotated.

This is because a single 3D shape can be used for many footprints (for instance, we use the shape resistor.wrl for several resistor footprints, by tuning the X, Y, Z scale of the 3D shape according to the different size of resistor footprints).

Some smd footprints are using this feature.

For the same reasons, the 3D shape can be moved (by the move factor) and/or rotated.

Real shape unit is 0.1 inch (1 unit vrml = 0.1 inch = 2.54 millimeter).

An other reason exists: when a footprint is very big (a big connector) or very small (a small SMD resistor) we must create a 3D shape small or bigger than real size, in order to use easily the 3D modeler.

\$SHAPE3D	Start description
Na "device/bornier_6.wrl"	File N ame (default path is kicad/modules/packages3d/)
Sc 1.000000 1.000000 1.000000	X Y Z s cale f actor
Of 0.000000 0.000000 0.000000	X Y Z o ffset (move vector, in 3D units (0.1 inch))
Ro 0.000000 0.000000 0.000000	X Y Z r otation (in degree)
\$EndSHAPE3D	End description

The 3D shape coordinates are relative to the footprint coordinates.

The 3D shape must be scale, moved and rotated according to the parameters Sc Of and Ro,

and after moved and rotated according to the footprint coordinates and rotation.

If the footprint is « inverted » (that is, located on copper side) the 3D shape must be « inverted » too.

Note:

A footprint may have several 3D shapes (for instance an integrated circuit and his socket).

1.8.6 - \$PAD

Pads have different shapes and attributes.

Pad shapes are:

Circle.

Oblong(or oval).

Rectangular (Square is like a rectangle).

Trapeze.

Pad attributes are:

- Normal (Has usually a hole)
- Smd (used for Surface Mounted Devices). Has no hole.
- Connector (used for connectors like a PC Board Bus connector)
- Mechanical. (Like a hole for mechanical use)

And shape can be draw with an offset related to the drilling hole.

The hole shale is round or oblong

\$PAD	Start description
Sh "2" C 1500 1500 0 0 2700	Shape: <pad name> shape Xsize Ysize Xdelta Ydelta Orientation
Dr 600 0 0 or (oblong hole) Dr 600 0 0 O 600 650	Drill <Pad drill> Xoffset Yoffset (round hole) or (oblong hole) Drill <Pad drill.x> Xoffset Yoffset <Hole shape> <Pad drill.x> <Pad drill.y>
At STD N 00E0FFFF	Attributs: <Pad type> N <layer mask>
Ne 8 "GND"	Net reference of the pad: <netnumber> <net name>
Po -3000 0	X_pos Y_pos (relative to the module position)
\$EndPAD	End description

Note:

<Pad type> is the Pad Attribute. It is one of: "STD" "SMD" "CONN" "HOLE" "MECA".

Shape is one of:

- C (circle)
- R (Rectangular).
- O (Oblong)
- T (Trapèze)

Hole shape = O (O for **O**blong)

Example:

```
$PAD
Sh "3" C 1500 1500 0 0 2700
Dr 600 0 0
At STD N 00E0FFFF
Ne 10 "TD0_1"
Po -1000 0
$EndPAD
```

1.9 - Graphic items:

There are drawing items like segments, circles, texts, targets and cotations.

1.9.1 - \$DRAWSEGMENT

Draw segments are :

- segments (strait line)
- circles
- arcs

1.9.1.1 - Line:

\$DRAWSEGMENT	Start description
Po 0 67500 39000 65500 39000 120	Position shape Xstart Ystart Xend Yend width
De 28 0 900 0 0	Description layer type angle timestamp status
\$EndDRAWSEGMENT	End description

Note:

- shape = 0
- Angle is used only for arc segments (unused for line, left for compatibility).

1.9.1.2 - Circle:

\$DRAWSEGMENT	Start description
Po 1 67500 39000 65500 39000 120	Position shape Xcentre Ycentre Xend Yend width
De 28 0 900 0 0	Description layer type angle timestamp status
\$EndDRAWSEGMENT	End description

Note:

- shape = 1
- Angle is used only for arc segments (unused for circle, left for compatibility).
- End is a point of this circle. (If Xend or Yend is 0, the other coordinate is the radius)

1.9.2 - Arc:

\$DRAWSEGMENT	Start description
Po 2 67500 39000 65500 39000 120	Position shape Xstart Ystart Xend Yend width
De 28 0 900 0 0	Description layer type angle timestamp status
\$EndDRAWSEGMENT	End description

Note:

- shape = 2
- *start* and *end* are the 2 points of the arc. *angle* is the arc angle (in 0.1 degree). Center coordinates are computed by pcbnew from *start*, *end* and *angle*.

Currently, only 90 degrees arcs are supported.(thereby, angle = 900)

Example:

```
$DRAWSEGMENT
Po 0 67500 34000 67500 39000 120
De 28 0 900 0
$EndDRAWSEGMENT
```

1.9.3 - \$TEXTPCB

Example: **TDI**

\$TEXTPCB	Start description
Te "TDI"	Text "string"
Po 57250 35750 600 600 150 0	Position Xstart Ystart Xsize Ysize Width rotation
De 15 1 B98C Normal	Description layer normal timestamp style normal = 0 : text is mirrored. normal = 1 : text is normal. style = Normal or Italic
\$EndTEXTPCB	End description

Example:

\$TEXTPCB



Te "TCK"

Po 57250 33500 600 600 150 0

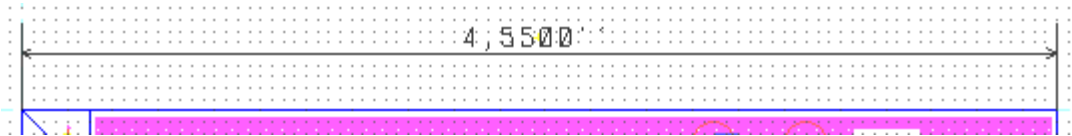
De 15 1 B98C Normal

\$EndTEXTPCB

1.9.4 - \$MIRE

	shape 1
	shape 0

\$MIREPCB	Start description
Po 0 28 28000 51000 5000 150 00000000	Position shape Xpos Ypos size width timestamp
\$EndMIREPCB	End description

1.9.5 - \$COTATION

\$COTATION	Start description
Ge 0 24 0	General shape layer timestamp currently, shape = 0.
Te "4,5500"	Text "string" string is the cotation value in inches or millimeters
Po 50250 5791 600 800 170 0 1	Position (for text) Xpos Ypos Xsize Ysize width orient normal
Sb 0 27500 6501 73000 6501 150	Coordinates of segments (axis, arrows...)
Sd 0 73000 9000 73000 5081 150	
Sg 0 27500 9000 27500 5081 150	
S1 0 73000 6501 72557 6731 150	
S2 0 73000 6501 72557 6271 150	
S3 0 27500 6501 27943 6731 150	
S4 0 27500 6501 27943 6271 150	
\$EndCOTATION	End description

1.10 - Track, vias and Zone section:**1.10.1 - \$TRACK**

Track section describes tracks and vias on copper layers.

Each track (or via) has a two line description:

For a track segment:

Position shape Xstart Ystart Xend Yend width

Description layer 0 netcode timestamp status

Shape parameter is set to 0 (reserved for future changes).

For a via:

Board File format.**PCBNEW****update: feb. 07 2009****Position** shape Xstart Ystart Xend Yend diameter**Description** layer 1 netcode timestamp status

For a via, layer parameter gives :

On the 4 less significant bits: the starting layer of the via

On the 4 next bits: the ending layer.

For instance, a via starting at copper kayer (layer 0) end ending at component layer (layer 15) has the layer parametre set to F0 hexadecimal or 240 decimal.

Shape parameter is the via type (*through* = 3, *blind* = 2, *buried* = 1)*Timestamp parameters are set to 0 (reserved for future changes).*

Status parameter can be set to 0 (Used internally for routing infos)..

\$TRACK	Start description
Po 0 36750 37000 36550 37000 250	Position shape Xstart Ystart Xend Yend width width = diameter for a via
De 15 0 1 0 400	Description layer type netcode timestamp status type = 0 for a track segment. type = 1 for a via
Po 0 39000 36750 38750 37000 250	An other track
De 15 0 1 0 0	
Po 3 53500 27000 53500 27000 650	This is a via (via "through") from layer 15 (component) to layer 0 (copper)
De 15 1 14 0 0	
\$EndTRACK	End description

1.10.2 - \$ZONE

Zone section is like track section. (There is no via in Zone section).

It is used to handle a zone filling, from a zone outline.

\$ZONE	Start description
Po 0 67100 33700 67100 38600 100	Same as track description
De 0 0 2 3EDDB09D 0	
\$EndZONE	End description

1.10.3 - \$CZONE_OUTLINE

Describes the main outlines of a zone and the outlines of filled areas (solid polygons) inside the zone main outlines.

Outlines of filled areas can be missing (if the zone is not currently filled)

Because a zone handles thermal reliefs, there are options to describe pads in zones options and thermal reliefs parameters.

Example:

\$CZONE_OUTLINE

ZInfo 47868246 1 "GND"

ZLayer 0

ZAux 4 E

ZClearance 150 T

ZMinThickness 190

ZOptions 0 32 F 200 200

ZCorner 74750 51750 0

ZCorner 74750 13250 0

ZCorner 29750 13250 0

ZCorner 29750 51750 1

```
....
$POLYSCORNERS
74655 51655 0 0
74655 13345 0 0
...
$endPOLYSCORNERS
$endCZONE_OUTLINE
```

\$CZONE_OUTLINE	Start description
ZInfo 478E3FC8 1 "/aux_sheet/INPUT"	<Time stamp> <internal netcode> "net name"
ZLayer 0	Layer (0 = copper, 15 = component, 1 ..14 = inner layers)
ZAux 4 E	<corners count> <zone hatching option> zone hatching option = N (none), E (edge hatching) or F (full hatching)
ZClearance 200 T	<Zone clearance> <pads option = I, T or X> I = pads in zone T = Thermal reliefs X = pads not in zone.
ZMinThickness 190	<Zone min thickness (for copper zone)>
ZOptions 0 32 F 200 200	<fill mode> <arc approx> <antipad thickness> <thermal stubs width> fill mode = 0 (use solid polygons) or 1 (use segments) arc approx = 16 or 32 (segments count to approximate a 360 arc)
ZCorner 49450 19150 0	First corner (external outline)
ZCorner 40600 19150 0	Next corner
ZCorner 40600 22850 0	
ZCorner 49450 22850 1	End corner (flag = 1)
\$POLYSCORNERS	Start of filled areas outlines
74655 51655 0 0	First corner (first filled area outline)
74655 13345 0 0	Next corner
\$endPOLYSCORNERS	
\$endCZONE_OUTLINE	End description

Other example:

\$CZONE_OUTLINE	Start description of an other outline
ZInfo 47B3E800 3 "VCC"	
ZLayer 1	
ZAux 8 F	
ZClearance 200 T	
ZMinThickness 190	Zone min thickness (for copper zone)
ZOptions 0 32 F 200 200	
ZCorner 49704 23032 0	First corner (external outline)
ZCorner 49704 18940 0	

Board File format.**PCBNEW****update: feb. 07 2009**

ZCorner 46140 19024 0	
ZCorner 46148 20000 0	
ZCorner 45250 20000 0	
ZCorner 44750 21250 0	
ZCorner 43750 22250 0	
ZCorner 46176 23068 1	End corner (flag = 1)
ZCorner 48450 19900 0	First corner (this is a hole)
ZCorner 48450 20800 0	
ZCorner 47350 20800 0	
ZCorner 47250 19900 1	End corner (flag = 1)
\$endCZONE_OUTLINE	End description

1.11 - \$EndBOARD

\$EndBOARD terminates the whole board description.
Must be the last line.