

## Table des matières

|  |           |
|--|-----------|
| <b>1 - Schematic files format:</b>               | <b>2</b>  |
| 1.1 - Units                                      | 3         |
| 1.2 - Header                                     | 3         |
| 1.3 - Description of a component                 | 3         |
| 1.4 - Description of a NoConnect symbol          | 4         |
| 1.5 - Description of a hierarchical sheet symbol | 4         |
| 1.6 - Description of a text note                 | 5         |
| 1.7 - Description of a Global Label              | 5         |
| 1.8 - Description of a Hierarchical label        | 5         |
| 1.9 - Description of a label                     | 5         |
| 1.10 - Description of a junction                 | 5         |
| 1.11 - Description of a wire segment (Wire)      | 6         |
| 1.12 - Description of a Bus segment              | 6         |
| 1.13 - Description of a dotted line segment      | 6         |
| 1.14 - Description of a bus entry                | 6         |
| <b>2 - Schematic Libraries Files Format:</b>     | <b>7</b>  |
| 2.1 - Units                                      | 7         |
| 2.2 - Heading                                    | 7         |
| 2.3 - Description of a component                 | 7         |
| 2.3.1 - Description of Alias                     | 8         |
| 2.3.2 - Description of the fields                | 8         |
| 2.3.3 - Description of graphic elements          | 8         |
| 2.3.3.1 - Polygon                                | 8         |
| 2.3.3.2 - Rectangle                              | 9         |
| 2.3.3.3 - Circle                                 | 9         |
| 2.3.3.4 - Arc of circle                          | 9         |
| 2.3.3.5 - Text field                             | 9         |
| 2.3.4 - Description of pins                      | 10        |
| <b>3 - Board File Format</b>                     | <b>12</b> |
| 3.1 - General Informations:                      | 12        |
| 3.2 - Layer numbering:                           | 13        |
| 3.3 - First line of description:                 | 13        |
| 3.4 - \$GENERAL                                  | 13        |
| 3.5 - \$SHEETDESCR                               | 14        |
| 3.6 - \$SETUP block:                             | 14        |
| 3.7 - \$EQUIPOT                                  | 15        |
| 3.8 - \$MODULE                                   | 16        |
| 3.8.1 - General description:                     | 16        |
| 3.8.2 - Field Description:                       | 17        |
| 3.8.3 - Drawings:                                | 17        |
| 3.8.4 - Pad Descriptions:                        | 18        |
| 3.8.5 - \$SHAPE3D                                | 18        |
| 3.8.6 - \$PAD                                    | 18        |
| 3.9 - Graphic items:                             | 19        |
| 3.9.1 - \$DRAWSEGMENT                            | 19        |
| 3.9.1.1 - Line:                                  | 19        |
| 3.9.1.2 - Circle:                                | 19        |
| 3.9.2 - Arc:                                     | 20        |
| 3.9.3 - \$TEXTPCB                                | 20        |
| 3.9.4 - \$MIRE                                   | 20        |
| 3.9.5 - \$COTATION                               | 20        |
| 3.10 - Track, vias and Zone section:             | 21        |
| 3.10.1 - \$TRACK                                 | 21        |
| 3.10.2 - \$ZONE                                  | 22        |
| 3.10.3 - \$CZONE_OUTLINE                         | 22        |
| 3.11 - \$EndBOARD                                | 23        |

## Files Format:

1 - Schematic files format:

- 1.1 - Units
- 1.2 - Header
- 1.3 - Description of a component
- 1.4 - Description of a NoConnect symbol
- 1.5 - Description of a hierarchical sheet symbol
- 1.6 - Description of a text note
- 1.7 - Description of a Global Label
- 1.8 - Description of a Hierarchical label
- 1.9 - Description of a label
- 1.10 - Description of a junction
- 1.11 - Description of a wire segment (Wire)
- 1.12 - Description of a Bus segment
- 1.13 - Description of a dotted line segment
- 1.14 - Description of a bus entry

2 - Schematic Libraries Files Format:

- 2.1 - Units
- 2.2 - Heading
- 2.3 - Description of a component
  - 2.3.1 - Description of Alias
  - 2.3.2 - Description of the fields
  - 2.3.3 - Description of graphic elements
    - 2.3.3.1 - Polygon :
    - 2.3.3.2 - Rectangle
    - 2.3.3.3 - Circle
    - 2.3.3.4 - Arc of circle
    - 2.3.3.5 - Text field
  - 2.3.4 - Description of the pins

3 - Board File Format

- 3.1 - General Informations:
- 3.2 - Layer numbering:
- 3.3 - First line of description:
- 3.4 - \$GENERAL
- 3.5 - \$SHEETDESCR
- 3.6 - \$SETUP block:
- 3.7 - \$EQUIPOT
- 3.8 - \$MODULE
  - 3.8.1 - General description:
  - 3.8.2 - Field Description:
  - 3.8.3 - Drawings:
  - 3.8.4 - Pad Descriptions:
  - 3.8.5 - \$SHAPE3D
  - 3.8.6 - \$PAD
- 3.9 - Graphic items:
  - 3.9.1 - \$DRAWSEGMENT
    - 3.9.1.1 - Line:
    - 3.9.1.2 - Circle:
  - 3.9.2 - Arc:
  - 3.9.3 - \$TEXTPCB
  - 3.9.4 - \$MIRE
  - 3.9.5 - \$COTATION
- 3.10 - Track, vias and Zone section:
  - 3.10.1 - \$TRACK
  - 3.10.2 - \$ZONE
  - 3.10.3 - \$CZONE\_OUTLINE
- 3.11 - \$EndBOARD

**1 - Schematic files format:****Schematic Files Format:**

|  |   |
|--|---|
| 1 - Schematic files format:                      | 1 |
| 1.1 - Units                                      | 1 |
| 1.2 - Header                                     | 1 |
| 1.3 - Description of a component                 | 1 |
| 1.4 - Description of a NoConnect symbol          | 2 |
| 1.5 - Description of a hierarchical sheet symbol | 2 |
| 1.6 - Description of a text note                 | 3 |
| 1.7 - Description of a Global Label              | 3 |
| 1.8 - Description of a Hierarchical label        | 3 |
| 1.9 - Description of a label                     | 3 |
| 1.10 - Description of a junction                 | 4 |
| 1.11 - Description of a wire segment (Wire)      | 4 |
| 1.12 - Description of a Bus segment              | 4 |
| 1.13 - Description of a dotted line segment      | 4 |
| 1.14 - Description of a bus entry                | 4 |

## 1.1 - Units

Sizes and coordinates are given in mils (1/1000 inch)

## 1.2 - Header

Format :

**EESchema Schematic File Version 1**

**LIBS:** *libraries list* (not used, for information only).

**EELAYER** *nn mm* (*nn mm* not used, reserved)

**EELAYER END**

**\$Descr** Sheet size *dimx dimy* (sheet size = A4..A0 ou A..E)

*Title block description* (Texts of the title block)

**\$EndDescr**

```
EESchema Schematic Spins Version 1
LIBS:brooktre, cypress, ttl, power, linear, memory, xilinx, idiot, aaci, INTEL, special, device, dsp
EELAYER 20 0
EELAYER END
$Descr A3 16535 11700
Sheet 1 4
""
Date "28 DEC 1996"
Rev ""
Comp ""
Comment1 ""
Comment2 ""
Comment3 ""
Comment4 ""
$EndDescr
```

## 1.3 - Description of a component

Format:

**\$Comp**

**L** *name reference*

**U** *N mm time\_stamp*

**P** *posx posy*

*List of fields:*

**F** *field\_number "text" orientation posX posY size Flags* (see below) *hjustify vjustify/italic/bold "name"*

**1** *posx posy* (redundant: not used)

**A B C B** ( orientation matrix with A, B, C, D = - 1, 0 or 1)

**\$EndComp**

Description of the fields:

**F** *n "text" orientation posx posy dimension flags hjustify vjustify/italic/bold "name"*

with *n* = field number (reference field = 0, value field = 1, *N* = 0..11 or more)

orientation = H (horizontal) or V (vertical).

- n = field number :
  - reference = 0.
  - value = 1.
  - Pcb FootPrint = 2.
  - User doc link = 3. At present time: not used (reserved)
- n = 4..11 = fields 1 to 8 (since January 2009 more than 8 field allowed, so n can be > 11).
- text (delimited by double quotes)
- orientation = H (horizontal) or V (vertical).
- position X and Y
- dimension (default = 50)
- Flags: visibility = 0 (visible) or 1 (invisible)
- hjustify vjustify = L R C B or T
  - L = left
  - R = Right
  - C = centre
  - B = bottom
  - T = Top
- Style: Italic = I or N ( since January 2009)
- Style Bold = B or N ( since January 2009)
- Name of the field (delimited by double quotes) (only if it is not the default name)

Note: vjustify, Italic and Bold are in the same 3 chars word.

Example:

```
Comp
L CONN_3 JP3
U 1 1 329879E1
P 1200 2000
F 0 "JP3" H 1250 2200 60 0000 C CNN
F 1 "CONN_3" V 1350 2000 50 0000 C CNN
F 4 "example" H 8000 4350 60 0000 C CIB "myfield"
  1 1200 2000
  1 0 0 - 1
$EndComp
```

## 1.4 - Description of a NoConnect symbol

Format: **NoConn** ~ *posx posy*

Example:

```
NoConn ~ 13400 5500
```

## 1.5 - Description of a hierarchical sheet symbol

Format:

**\$Sheet**

**S** *posx posy dimx dimy*

*List of Sheet Labels*

**\$EndSheet**

Format of Sheet Labels

Fn "text" forms side posx posy dimension

With:

n = sequence number (0..x).

n = 0: name of the corresponding schematic file.

n = 1: name of the sheet of hierarchy.

form = I (input) O (output)

side = R (right) or L (left).

Example:

```
$Sheet
S 1800 1600 1500 1500
```

```

F0 "PROGALIM.SCH" 60
F1 "PROGALIM.SCH" 60
F2 "CLK" O R 3300 1800 60
F3 "/RESET" O R 3300 2000 60
F4 "VPWR" O R 3300 2700 60
F5 "/HALT" O R 3300 2100 60
F6 "TRANSF1" I L 1800 1900 60
F7 "TRANSF2" I L 1800 2000 60
F8 "3.84MH" O R 3300 2200 60
$EndSheet

```

### 1.6 - Description of a text note

Format: **Text Notes** *posx posy orientation dimension ~*  
*Text*

Example:

```

Text Notes 2100 3250 1 60 ~
TOTO

```

### 1.7 - Description of a Global Label

Format: **Text GLabel** *posx posy orientation dimension shape*  
*Text*

Example:

```

Text GLabel 3100 2500 2 60 UnSpC
TITI
Text GLabel 3150 2700 1 60 3State
3STATES
Text GLabel 2750 2800 0 60 UnSpC
BIDI
Text GLabel 2750 2650 0 60 Output
GLABELOUT
Text GLabel 2750 2400 0 60 Input
RESET

```

### 1.8 - Description of a Hierarchical label

Format: **Text HLabel** *posx posy orientation dimension shape*  
*Text*

Example:

```

Text HLabel 3400 2000 0 60 Input
/RESET

```

### 1.9 - Description of a label

Format: **Text Label** *posx posy orientation dimension ~*  
*Text*

Example:

```

Text Label 3400 2000 0 60 ~
/RESET

```

### 1.10 - Description of a junction

Format: **Connection** *~ posx posy*

Example:

```
Connection ~ 13300 6500
```

### 1.11 - Description of a wire segment (Wire)

Format:

**Wire Wire Line**

*startx starty endx endy*

Example:

```
Wire Wire Line
3300 1800 3900 1800
```

### 1.12 - Description of a Bus segment

Format:

**Wire Bus Line**

*startx starty endx endy*

Example:

```
Wire Bus Line
3900 5300 4500 5300
```

### 1.13 - Description of a dotted line segment

Format:

**Wire Notes Line**

*startx starty endx endy*

Example:

```
Wire Notes Line
2850 3350 2850 3050
```

### 1.14 - Description of a bus entry

Format:

- For an entry wire/bus :

**Wire Wire Bus**

*startx starty endx endy*

- For an entry bus/bus :

**Wire Bus Bus**

*startx starty endx endy*

Example:

```
Entry Wire Bus
4100 2300 4200 2400
Entry Bus Bus
4400 2600 4500 2700
```

## 2 - Schematic Libraries Files Format:

### Schematic Libraries Files Format:

|   |   |
|---|---|
| 1 - Schematic Libraries Files Format:   | 1 |
| 1.1 - Units                             | 1 |
| 1.2 - Heading                           | 1 |
| 1.3 - Description of a component        | 1 |
| 1.3.1 - Description of Alias            | 2 |
| 1.3.2 - Description of the fields       | 2 |
| 1.3.3 - Description of graphic elements | 2 |
| 1.3.3.1 - Polygon                       | 3 |
| 1.3.3.2 - Rectangle                     | 3 |
| 1.3.3.3 - Circle                        | 3 |
| 1.3.3.4 - Arc of circle                 | 3 |
| 1.3.3.5 - Text field                    | 4 |
| 1.3.4 - Description of pins             | 4 |

#### 2.1 - Units

Sizes and coordinates are given in mils (1/1000 inch)

#### 2.2 - Heading

format:

```
EESchema-LIBRARY Version 2.0 24/1/1997-18:9:6
description of the components
# End Library
```

#### 2.3 - Description of a component

The format is as follows :

```
DEF name reference unused text_offset draw_pinnumber draw_pinname unit_count units_locked
option_flag
ALIAS name1 name2...
fields list
DRAW
list graphic elements and pins
ENDDRAW
ENDDEF
```

Parameters for **DEF** :

- **name** = component name in library (74LS02 ...)
- **référence** = Reference ( U, R, IC ..., which become U3, U8, R1, R45, IC4...)
- **unused** = 0 (reserved)
- **text\_offset** = offset for pin name position
- **draw\_pinnumber** = Y (display pin number) ou N (do not display pin number).
- **draw\_pinname** = Y (display pin name) ou N (do not display pin name).
- **unit\_count** = Number of part ( or section) in a component package.
- **units\_locked** = L (units are not identical and cannot be swapped) or F (units are identical and therefore can be swapped) (Used only if unit\_count > 1)
- **option\_flag** = N (normal) or P (component type "power")

Example:

```
DEF BNC P 0 40 Y NR 1 L NR
F0 "P" 10.120 60 H V L C
F1 "BNC" 110 - 60 40 V V L C
DRAW
```

```
C 0 0 70 0 1 0
C 0 0 20 0 1 0
X Ext. 2 0 - 200 130 U 40 40 1 1 P
X In 1 - 150 0.130 R 40 40 1 1 P
ENDDRAW
ENDDEF
```

### 2.3.1 - Description of Alias

This line exists only if the component has alias names.

format:

**ALIAS** *name1 name2 name3...*

### 2.3.2 - Description of the fields

format:

**F** *n "text" posx posy dimension orientation visibility hjustify vjustify/italic/bold "name"*

with:

- n = field number :
  - reference = 0.
  - value = 1.
  - Pcb FootPrint = 2.
  - User doc link = 3. At present time: not used
- n = 4..11 = fields 1 to 8 (since January 2009 more than 8 field allowed, so n can be > 11).
- text (delimited by double quotes)
- position X and Y
- dimension (default = 50)
- orientation = H (horizontal) or V (vertical).
- Visibility = V (visible) or I (invisible)
- hjustify vjustify = L R C B or T
  - L= left
  - R = Right
  - C = centre
  - B = bottom
  - T = Top
- Style: Italic = I or N ( since January 2009)
- Style Bold = B or N ( since January 2009)
- Name of the field (delimited by double quotes) (only if it is not the default name)

Note: vjustify, Italic and Bold are in the same 3 chars word.

Example:

```
DEF DIODE D 0 40 Y NR 1 0 NR
F0 "D" 0.100 50 H V L CNN
F1 "DIODE" 0 -100 50 H V L CIB
F5 "2euros" 0 -200 50 H V L CIB "PRICE"
```

### 2.3.3 - Description of graphic elements

There are of 5 types:

- Polygon (succession of segments), filled or normal.
- Rectangle.
- Circle.
- Arc of circle.
- Text.

#### 2.3.3.1 - Polygon :

Format:

**P** *Nb parts convert ltrait x0 y0 x1 y1 xi yi cc*

With:

- Nb = a number of points.



- unit = 0 if common to the parts; if not, number of part (1. .n).
- convert = 0 if common to the 2 representations, if not 1 or 2.
- *ltrait* = line thickness.
- xi yi coordinates of end i.
- cc = N F or F ( F = filled polygon; f = . filled polygon, transparent background)

Example:

```
P 3 0 1 0 - 50 50 50 0 - 50 - 50 F
P 2 0 1 0 50 50 50 - 50 N
```

### 2.3.3.2 - Rectangle

Format:

**S** *startx starty endx endy unit convert ltrait cc*

With

- unit = 0 if common to the parts; if not, number of part (1. .n).
- convert = 0 if common to the representations; if not, 1 or 2.
- ltrait = thickness.
- cc = N F or F ( F = filled Rectangle,; f = . filled Rectangle, transparent background)

Example:

```
S 0 50.900.900 0 1 0 f
```

### 2.3.3.3 - Circle

Format:

**C** *posx posy radius unit convert ltrait cc*

With

- unit = 0 if common to the parts; if not, number of part (1. .n).
- convert = 0 so common to the representations, if not 1 or 2.
- ltrait = thickness.
- cc = N F or F ( F = filled Rectangle,; f = . filled Rectangle, transparent background)

Example:

```
C 0 0 70 0 1 0 F
C 0 0 20 0 1 0 N
```

### 2.3.3.4 - Arc of circle

Format:

**With** *posx posy radius start end part convert ltrait start\_pointX start\_pointY end\_pointX end\_pointY cc*

With:

- start = angle of the starting point (in 0,1 degrees).
- end = angle of the end point (in 0,1 degrees).
- unit = 0 so common to the parts; if not, number of part (1. .n).
- convert = 0 if common to the representations, if not 1 or 2.
- ltrait = thickness.
- start\_pointX start\_pointY = coord of the starting point (role similar to start)
- end\_pointX end\_pointY = coord of the point of arrival (role similar to end)
- cc = N F or F ( F = filled Rectangle,; f = . filled Rectangle, transparent background)

Example:

```
To 0.148 48 - 889 889 0 1 0 N
To 0 51 51 - 889 889 0 1 0 N
```

### 2.3.3.5 - Text field

Format:

**T** *orientation posx posy dimension unit convert Text*

With:

- orientation = horizontal orientation (=0) or vertical (=1).
- type = always 0.
- unit = 0 so common to the parts, if not number of part (1. .n).
- convert = 0 if common to the representations, if not 1 or 2.

Example:

```
T 0 - 320 - 10 100 0 0 1 VREF
```

### 2.3.4 - Description of pins

Format:

**X** *name number posx posy length orientation Snum Snom unit convert Etype [shape]*.

With:

- orientation = U (up) D (down) R (right) L (left).
- name = name (without space) of the pin. if ~: no name
- number = n pin number (4 characters maximum).
- length = pin length.
- Snum = pin number text size.
- Snom = pin name text size.
- unit = 0 if common to the parts; if not, number of part (1. .n).
- convert = 0 if common to the representations, if not 1 or 2.
- Etype = electric type (1 character)
- shape = if present: pin shape (clock, inversion...).

Example:

```
X TO 1 - 200 0.150 R 40 40 1 1 P
X K 2.200 0.150 L 40 40 1 1 P
X 0 1 0 0 0 R 40 40 1 1 W NC
X ~ 2 0 - 250 200 U 40 40 1 1 P
```

Etype list:

|                |          |
|----------------|----------|
| INPUT          | <b>I</b> |
| OUTPUT         | <b>O</b> |
| BIDI           | <b>B</b> |
| TRISTATE       | <b>T</b> |
| PASSIVE        | <b>P</b> |
| UNSPECIFIED    | <b>U</b> |
| POWER INPUT    | <b>W</b> |
| POWER OUTPUT   | <b>w</b> |
| OPEN COLLECTOR | <b>C</b> |
| OPEN EMITTER   | <b>E</b> |
| NOT CONNECTED  | <b>N</b> |

Shape list:

- If invisible pin, the shape identifier starts by **N**
- Next character is:

|                    |                |
|--------------------|----------------|
| Line               | None (default) |
| Inverted           | <b>I</b>       |
| Clock              | <b>C</b>       |
| Inverted clock     | <b>CI</b>      |
| Input low          | <b>L</b>       |
| Clock low          | <b>CL</b>      |
| Output low         | <b>V</b>       |
| Falling edge clock | <b>F</b>       |
| Non Logic          | <b>X</b>       |

Example:

A clock is coded **C** if visible, and **NC** if invisible.

### 3 - 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 |

### 3.1 - 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

```

### 3.2 - Layer numbering:

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

Pads and vias use several layers.

There are 16 copper layers and 12 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)            |                  |
| 26      | ECO2 layer (Other layer used for general drawings)            |                  |
| 27      | Edge layer. Items on Edge layer <b>are seen on all layers</b> |                  |

#### 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).

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

### 3.3 - 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).

### 3.4 - \$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:<br>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  |

### 3.5 - \$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                                 |

### 3.6 - \$SETUP block:

This data bock is used for design settings

This is useful only for board edition.

Example:

**\$SETUP**

*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<br>name = name given to the layer by the user (here: "cuivre")<br>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  |
| 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<br>(Auxiliary axis is the reference coordinate (0 0 coordinate) for EXCELLON drilling files)             |
| \$EndSETUP                 | End block "SETUP"  |

### 3.7 - \$EQUIPOT

\$EQUIPOT describes a net name.

|                 |  |
|-----------------|--|
| \$EQUIPOT       | Start block                                  |
| Na 2 "N-000026" | <b>Na</b> <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
```

**3.8 - \$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.

**3.8.1 - General description:**

|   |   |
|---|---|
| \$MODULE bornier6                           | <b>\$MODULE</b> <module lib name>   |
| Po 62000 30500 2700 15 3EC0C28A 3EBF830C ~~ | <b>Po</b> Xpos Ypos Orientation(0.1deg) Layer TimeStamp<br>Attribut1Attribut2<br>Attribut1 = ~or 'F' for autoplacement (F = Fixed, ~= moveable)<br>Attribut2 = ~or 'P' for autoplacement (P = autoplaced) |
| Li bornier6                                 | <b>Li</b> <module lib name>   |
| Cd Bornier d'alimentation 4 pins            | <b>Cd</b> comment description (displayed when browsing libraries)   |



|                   |   |
|-------------------|---|
| \$MODULE bornier6 | <b>\$MODULE</b> <module lib name>   |
| Kw DEV            | <b>Kw</b> Keyword1 Keyword2 ... (for footprint selection by keywords)   |
| Sc 3EBF830C       | <b>Sc</b> TimeStampOp   |
| Op 0 0 0          | <b>Op</b> <rotation cost 90 deg> <rotation cost 180 deg> for auto place.<br>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**

**3.8.2 - 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"

| <b>Field</b> | <b>Units</b>                  | <b>Meaning</b>  |
|--------------|-------------------------------|---|
| 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?  |
| 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     |

**3.8.3 - Drawings:**

Tells how to draw module shape.

Drawings are segment, circle, arc.

|                                  |   |
|----------------------------------|---|
| DS -6000 -1500 -6000 1500 120 21 | DS is a <b>Draw Segment</b><br>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 <b>Draw Circle</b><br>DC Xcentre Ycentre Xpoint Ypoint Width Layer |
| DA x0 y0 x1 y1 angle width layer | DA is a <b>Draw Arc</b><br>X0,y0 = Start point x1,y1 = end point           |

### 3.8.4 - Pad Descriptions:

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

### 3.8.5 - \$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"     | <i>FileName</i> (default path is kicad/modules/packages3d/) |
| Sc 1.000000 1.000000 1.000000 | X Y Z <i>scale factor</i>                                   |
| Of 0.000000 0.000000 0.000000 | X Y Z <i>offset (move vector, in 3D units (0.1 inch))</i>   |
| Ro 0.000000 0.000000 0.000000 | X Y Z <i>rotation (in degree)</i>                           |
| \$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).

### 3.8.6 - \$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                            | <b>Shape:</b> <pad name> shape Xsize Ysize Xdelta Ydelta Orientation   |
| Dr 600 0 0<br>or (oblong hole)<br>Dr 600 0 0 O 600 650 | <b>Drill</b> <Pad drill> Xoffset Yoffset (round hole)<br>or (oblong hole)<br><b>Drill</b> <Pad drill.x> Xoffset Yoffset <Hole shape> <Pad drill.x> <Pad drill.y> |
| At STD N 00E0FFFF                                      | <b>Attributs:</b> <Pad type> N <layer mask>  |
| Ne 8 "GND"   | <b>Net reference of the pad:</b> <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 Oblong)

**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
```

### 3.9 - Graphic items:

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

#### 3.9.1 - \$DRAWSEGMENT

Draw segments are :

- segments (strait line)
- circles
- arcs

##### 3.9.1.1 - Line:

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

**Note:**

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

##### 3.9.1.2 - Circle:

|                                  |   |
|----------------------------------|---|
| \$DRAWSEGMENT                    | Start description                                     |
| Po 1 67500 39000 65500 39000 120 | <b>Position</b> shape Xcentre Ycentre Xend Yend width |
| De 28 0 900 0 0                  | <b>Description</b> 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)

### 3.9.2 - Arc:

|                                  |  |
|----------------------------------|--|
| \$DRAWSEGMENT                    | Start description                                    |
| Po 2 67500 39000 65500 39000 120 | <b>Position</b> shape Xstart Ystart Xend Yend width  |
| De 28 0 900 0 0                  | <b>Description</b> 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
```

### 3.9.3 - \$TEXTPCB



Example: **TDI**

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

**Example:**

```
$TEXTPCB
Te "TCK"
Po 57250 33500 600 600 150 0
De 15 1 B98C Normal
$EndTEXTPCB
```

### 3.9.4 - \$MIRE

|   |         |
|---|---------|
|  | shape 1 |
|  | shape 0 |

|                                       |  |
|---------------------------------------|--|
| \$MIREPCB                             | Start description                                    |
| Po 0 28 28000 51000 5000 150 00000000 | <b>Position</b> shape Xpos Ypos size width timestamp |
| \$EndMIREPCB                          | End description                                      |

### 3.9.5 - \$COTATION



|                                |   |
|--------------------------------|---|
| \$COTATION                     | Start description   |
| Ge 0 24 0                      | <b>General</b> shape layer timestamp<br>currently, shape = 0.                 |
| Te "4,5500""                   | <b>Text</b> "string"<br>string is the cotation value in inches or millimeters |
| Po 50250 5791 600 800 170 0 1  | <b>Position</b> (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   |

### 3.10 - Track, vias and Zone section:

#### 3.10.1 - \$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:

**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 layer (layer 0) end ending at component layer (layer 15) has the layer parameter 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 | <b>Position</b> shape Xstart Ystart Xend Yend width<br>width = diameter for a via                             |
| De 15 0 1 0 400                  | <b>Description</b> layer type netcode timestamp status<br>type = 0 for a track segment.<br>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   |

### 3.10.2 - \$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           |

### 3.10.3 - \$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><br>zone hatching option = N (none), E (edge hatching) or F (full hatching)   |
| ZClearance 200 T                    | <Zone clearance> <pads option = I, T or X><br>I = pads in zone<br>T = Thermal reliefs<br>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><br>fill mode = 0 (use solid polygons) or 1 (use segments)<br>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   |                                       |
| 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                       |

### 3.11 - \$EndBOARD

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