### Headings

1 - Schematic files format: ................................................................. 3  
  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 ...................................................... 6  
  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  
  1.15 - Description of a Bitmap Image ............................................. 6  

2 - Schematic Libraries Files Format: ............................................... 8  
  2.1 - Units ....................................................................................... 8  
  2.2 - File header ............................................................................. 8  
  2.3 - Description of a component ................................................... 8  
    2.3.1 - Description of Aliases ..................................................... 9  
    2.3.2 - Description of the fields ................................................. 9  
      2.3.2.1 - Important Note 1: .................................................. 9  
      2.3.2.2 - Important Note 2: .................................................. 9  
    2.3.3 - Description of graphic elements .................................... 9  
      2.3.3.1 - Polygon : ............................................................ 10  
      2.3.3.2 - Rectangle ............................................................ 10  
      2.3.3.3 - Circle ................................................................. 10  
      2.3.3.4 - Arc of circle ......................................................... 10  
    2.3.4 - Description of pins ......................................................... 11  

3 - Board File Format (Format versions 1 and 2)............................... 13  
  3.1 - Information about V1 version ............................................... 13  
  3.2 - Information about V2 version ............................................... 14  
  3.3 - Information about new 'S expression' version ..................... 14  
  3.4 - Layer numbering: ............................................................... 15  
  3.5 - First line of description: ...................................................... 16  
  3.6 - $GENERAL ................................................................. 16  
  3.7 - $SHEETDESCR ............................................................. 16  
  3.8 - $SETUP block: ................................................................. 16  
  3.9 - $EQUIPOT ................................................................. 18  
  3.10 - $MODULE ................................................................. 18  
    3.10.1 - General description .................................................. 19  
    3.10.2 - Field Description ..................................................... 19  
    3.10.3 - Drawings: ............................................................... 20  
      3.10.3.1 - Draw segment: ............................................... 20  
      3.10.3.2 - Circle: ........................................................... 20  
      3.10.3.3 - Arc: ............................................................... 20  
      3.10.3.4 - Polygon: ......................................................... 20  
    3.10.4 - Pad Descriptions: ...................................................... 20  
    3.10.5 - $SHAPE3D ............................................................ 20  
    3.10.6 - $PAD ................................................................. 21  
  3.11 - Graphic items: ................................................................. 21  
    3.11.1 - $DRAWSegment ....................................................... 22  
      3.11.1.1 - Line: ............................................................. 22  
      3.11.1.2 - Circle: ........................................................... 22  
    3.11.2 - Arc: ................................................................. 22  
    3.11.3 - $TEXTPCB ............................................................ 22  
    3.11.4 - $MIRE ............................................................... 23
### Files Formats

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.11.5</td>
<td>$COTATION</td>
</tr>
<tr>
<td>3.12</td>
<td>Track, vias and Zone section</td>
</tr>
<tr>
<td>3.12.1</td>
<td>$TRACK</td>
</tr>
<tr>
<td>3.12.2</td>
<td>$ZONE</td>
</tr>
<tr>
<td>3.12.3</td>
<td>$CZONE_OUTLINE</td>
</tr>
<tr>
<td>3.13</td>
<td>$EndBOARD</td>
</tr>
</tbody>
</table>

### KiCad

4 - Pcbnew "S-expression" file format:

- 4.1 - File format syntax:
- 4.2 - Coordinates of objects and sizes:
- 4.3 - Keywords:
- 4.4 - Identifiers and Strings:
- 4.5 - Layer representation in files:
  - 4.5.1 - Layer capacity:
  - 4.5.2 - Layer names in files:
- 4.6 - Typical structure of the board file:
  - 4.6.1 - Description of an item:
  - 4.6.2 - The header line:
  - 4.6.3 - The general section:
  - 4.6.4 - The layers section (the mapping of layers):
  - 4.6.5 - The setup section:
  - 4.6.6 - The list of nets:
  - 4.6.7 - The list of net classes:
  - 4.6.8 - The list of modules:
  - 4.6.9 - The list of graphic items:
  - 4.6.10 - The list of tracks:
  - 4.6.11 - The list of zones:
  - 4.6.12 - Description of a module (footprint):
- 4.7 - Board Example:

5 - Page layout file format:

- 5.1 - General:
- 5.2 - Units and coordinates:
- 5.3 - basic items:
- 5.4 - Coordinates and reference corners:
- 5.5 - Items options:
- 5.6 - Keywords used in description:
- 5.7 - Comments:
- 5.8 - Coordinates:
- 5.9 - Items:
- 5.10 - Header and setup:
- 5.11 - Line and rect:
- 5.12 - Texts:
  - 5.12.1 - Basic texts:
  - 5.12.2 - Multi-line texts:
  - 5.12.3 - Font option:
  - 5.12.4 - Justification:
  - 5.12.5 - Constrained text:
  - 5.12.6 - Format symbols:
- 5.13 - Poly Polygons:
- 5.14 - File format:
  - 5.14.1 - Header section:
  - 5.14.2 - Setup section:
  - 5.14.3 - Data section:
  - 5.14.3.1 - Rect item:
  - 5.14.3.2 - Line item:
  - 5.14.3.3 - Text item:
  - 5.14.3.4 - Poly polygon item:
  - 5.14.4 - Example:
1 - Schematic files format:

Headings

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:

**SComp**

L name reference

U N mm time_stamp

P posx posy

**List of fields:**
**Schematic files format:**

**KiCad**

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.

- 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
$ 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.
```

---

Schematic files format: page 4/40
Files Formats

Schematic files format:

KiCad

form = I (input) O (output) B (BiDi) T (tri state) U (unspecified)
side = R (right), L (left), T (top), B (bottom)

Example:

S$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

1.15 - Description of a Bitmap Image
Bitmaps are considered to be 300x300 pixels per inch.
A scaling factor is applied by Eeschema to adjust the actual bitmap size on screen.
Format:

$Bitmap

Pos posx posy

Scale scale value (float). This is the user scalin factor used to display the bitmap.

Data

Bitmap data, PNG format, in hexadecimal.
Each byte is coded by 2 hexadecimal digits.
Bytes are separated by a space.

EndData

$EndBitmap

Example:

$Bitmap
Pos 7450 5600
Scale 1,000000
Data
89 50 4E 47 0D 0A 1A 0A 00 00 00 0D 49 48 44 52 00 00 01 00 00 01 00 08 06 00 00 00 5C 72 A8
66 00 00 00 04 73 42 49 54 08 08 08 08 7C 08 64 88 00 00 00 09 70 48 59 73 00 00 1B 58 00 00 1B
....
EndData
$EndBitmap
2 - Schematic Libraries Files Format:

Headings

2 - Schematic Libraries Files Format: ................................................................................................................. 8
  2.1 - Units ......................................................................................................................................................... 8
  2.2 - File header ............................................................................................................................................. 8
  2.3 - Description of a component .................................................................................................................. 8
    2.3.1 - Description of Aliases ..................................................................................................................... 9
    2.3.2 - Description of the fields ................................................................................................................ 9
      2.3.2.1 - Important Note 1 ..................................................................................................................... 9
      2.3.2.2 - Important Note 2 ..................................................................................................................... 9
    2.3.3 - Description of graphic elements ..................................................................................................... 9
      2.3.3.1 - Polygon ...................................................................................................................................... 10
      2.3.3.2 - Rectangle ................................................................................................................................... 10
      2.3.3.3 - Circle .......................................................................................................................................... 10
      2.3.3.4 - Arc of circle ............................................................................................................................... 10
      2.3.3.5 - Text field .................................................................................................................................... 11
    2.3.4 - Description of pins ........................................................................................................................... 11

2.1 - Units

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

2.2 - File header

format:

```
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) ou F (units are identical and therefore can be swapped) (Used only if unit_count > 1)
- **option_flag** = N (normal) ou 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
```
2.3.1 - Description of Aliases
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”
```

2.3.2.1 - Important Note 1:
The F1 field is the default component value and the component name in library.
**So the F1 field text should be the same as the name.**

2.3.2.2 - Important Note 2:
F0 is the reference prefix.
If the prefix starts b # (like #U) the component is not output to netlist or Bill Of Material.
This is a “virtual” component.
Mainly power symbols must have the prefix starting by #.

2.3.3 - Description of graphic elements
There are of 5 types:
- Polygon (succession of segments), filled or normal.
2.3.3.1 - Polygon :

Format: \[ P \text{ Nb parts convert thickness x0 y0 x1 y1 xi yi cc} \]

With:
- \( \text{Nb} \) = 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.
- thickness = line thickness.
- xi yi coordinates of end i.
- cc = N F or F (F = filled polygon; f = . filled polygon, N = 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: \[ S \text{ startx starty endx endy unit convert thickness cc} \]

With:
- unit = 0 if common to the parts; if not, number of part (1..n).
- convert = 0 if common to all parts. If not, number of the part (1..n).
- thickness = thickness of the outline.
- cc = N F or F (F = filled Rectangle; f = . filled Rectangle, N = transparent background)

Example:

```
S 0 50.90 0.90 0 1 0 f
```

2.3.3.3 - Circle

Format: \[ C \text{ posx posy radius unit convert thickness cc} \]

With:
- posx posy = circle center position
- unit = 0 if common to the parts; if not, number of part (1..n).
- convert = 0 if common to all parts. If not, number of the part (1..n).
- thickness = thickness of the outline.
- cc = N F or F (F = filled circle; f = . filled circle, N = 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: \[ A \text{ posx posy radius start end part convert thickness cc start_pointX start_pointY end_pointX end_pointY} \]

With:
- posx posy = arc center position
- start = angle of the starting point (in 0,1 degrees).
- end = angle of the end point (in 0,1 degrees).
- unit = 0 if common to all parts; if not, number of the part (1..n).
- convert = 0 if common to the representations, if not 1 or 2.
- thickness = thickness of the outline or 0 to use the default line thickness.
- cc = N F or F (F = filled arc; f = . filled arc, N = transparent background)
- start_pointX start_pointY = coordinate of the starting point (role similar to start)
Files Formats

Schematic Libraries Files Format:

KiCad

- end_pointX end_pointY = coordinate of the ending point (role similar to end)

Example:

| A -1 -200 49 900 -11 0 1 0 N -50 -200 0 -150 |
| A 0 -199 49 0 -911 0 1 0 N 0 -150 50 -200 |

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 if common to the parts. If not, the number of the 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 all parts. If not, number of the 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 O 1 0 0 R 40 40 1 1 W NC |
| X ~ 2 0 - 250 200 U 40 40 1 1 P |

Etype list:

| INPUT       | I |
| OUTPUT      | O |
| BIDI        | B |
| TRISTATE    | T |
| PASSIVE     | P |
| UNSPECIFIED | U |
| POWER INPUT | W |
| POWER OUTPUT| w |
| OPEN COLLECTOR | C |
| OPEN Emitter | E |
| NOT CONNECTED | N |
Shape list:
- If invisible pin, the shape identifier starts by N
- Next character is:

<table>
<thead>
<tr>
<th>Line</th>
<th>None (default)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inverted</td>
<td>I</td>
</tr>
<tr>
<td>Clock</td>
<td>C</td>
</tr>
<tr>
<td>Inverted clock</td>
<td>CI</td>
</tr>
<tr>
<td>Input low</td>
<td>L</td>
</tr>
<tr>
<td>Clock low</td>
<td>CL</td>
</tr>
<tr>
<td>Output low</td>
<td>V</td>
</tr>
<tr>
<td>Falling edge clock</td>
<td>F</td>
</tr>
<tr>
<td>Non Logic</td>
<td>X</td>
</tr>
</tbody>
</table>

Example:
A clock is coded C if visible, and NC if invisible.
3 - Board File Format (Format versions 1 and 2)

Headdings

3.1 - Information about V1 version: ................................................................. 13
3.2 - Information about V2 version: ................................................................. 13
3.3 - Information about new “S expression” version: ....................................... 14
3.4 - Layer numbering: .................................................................................. 15
3.5 - First line of description: .......................................................................... 16
3.6 - $GENERAL ............................................................................................ 16
3.7 - $SHEETDESCR ..................................................................................... 16
3.8 - $SETUP block: ...................................................................................... 16
3.9 - $EQUIPOT ............................................................................................. 18
3.10 - $MODULE .............................................................................................. 18
3.11 - Graphic items: ...................................................................................... 21
3.12 - Track, vias and Zone section: ................................................................. 24
3.13 - $EndBOARD ......................................................................................... 26

3.1 - Information about V1 version:

- Board files ( *.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 1 date 02/04/2011 15:04:20

All the following descriptions are like this:

$DESCRIPTION
some data

$endDESCRIPTION

when <DESCRIPTION> is an identifier which gives the meaning of the data between $DESCRIPTION and $endDESCRIPTION.

Example:

$GENERAL
encoding utf-8
LayerCount 2
Ly 1FFF8001
3.2 - Information about V2 version:

The file format is exactly the same format, and the extension is still .brd. However, dimensions are in mm (floating notation), except for the page size (in 1/1000 inch). Because the internal Pcbnew unit is now 1nm, the integer coordinates in 1/10000 inch cannot be used in files. Of course, the Pcbnew versions which are in nm are able to read the V1 version files, but can only write files in V2 version.

The V2 version should be seen as a temporary way to store boards without loss of resolution.

First line is something as:

```
PCBNEW-BOARD Version 2 date 22/02/2013 15:04:20
```

All the following descriptions are like this:

```
$DESCRIPTION
some data
...
$endDESCRIPTION
```

Example:

```
PCBNEW-BOARD Version 2 date 22/02/2013 10:33:30
# Created by Pcbnew(2013-02-20 BZR 3963)-testing
```

3.3 - Information about new “S expression” version:
For Pcbnew versions in nanometers, the default file format is now using “S expressions”. This new format uses mm for coordinates, fixes issues (like spaces in names) in V1 and V2 versions, and is more human readable than the older format. The new file extension is `.kicad_pcb`

Here is a sample:

```kicad_pcb
(kicad_pcb (version 3) (host pcbnew "(2013-01-12 BZR 3902)-testing")
  (general
   (links 200)
   (no_connects 0)
   (area 69.241669 24.89454 202.336401 196.2404)
   (thickness 1.6002)
   (drawings 19)
   (tracks 779)
   (zones 0)
   (modules 25)
   (nets 111)
  )
  (page A4)
  (title_block
   (title Demo)
   (rev 2.C)
   (company Kicad)
  )
)
```

### 3.4 - 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:

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

### 3.5 - 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).

### 3.6 - $GENERAL

This data is useful only when loading file. It is used by Pcbnew for displaying activity when loading data.

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

### 3.7 - $SHEETDESCR

This the page size and texts.

<table>
<thead>
<tr>
<th>$SHEETDESCR</th>
<th>Start description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sheet A4 11700 8267</td>
<td>&lt;Page size&gt; X_size Y_size in mils (1/1000 inch)</td>
</tr>
<tr>
<td>Title &quot;&quot;</td>
<td>Title text</td>
</tr>
<tr>
<td>Date &quot;23 feb 2004&quot;</td>
<td>Date text</td>
</tr>
<tr>
<td>Rev &quot;&quot;</td>
<td>Revision text</td>
</tr>
<tr>
<td>Comp &quot;&quot;</td>
<td>Company name text</td>
</tr>
<tr>
<td>Comment1 &quot;&quot;</td>
<td>Comment text, line 1</td>
</tr>
<tr>
<td>Comment2 &quot;&quot;</td>
<td>Comment text, line 2</td>
</tr>
<tr>
<td>Comment3 &quot;&quot;</td>
<td>Comment text, line 3</td>
</tr>
<tr>
<td>Comment4 &quot;&quot;</td>
<td>Comment text, line 4</td>
</tr>
<tr>
<td>$EndSHEETDESCR</td>
<td>End description</td>
</tr>
</tbody>
</table>

### 3.8 - $SETUP block:

This data block is used for design settings. This is useful only for board edition. Example:
### Files Formats

**Board File Format (Format versions 1 and 2)**

**KiCad**

```plaintext
$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
```

<table>
<thead>
<tr>
<th>Variable</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$SETUP</td>
<td>Start block &quot;SETUP&quot;</td>
</tr>
<tr>
<td>InternalUnit 0.000100 INCH</td>
<td>Internal unit for Pcbnew, all coordinates are in this unit</td>
</tr>
<tr>
<td>Layers 2</td>
<td>Number of layers (2 = double sided board) must be 1 to 16</td>
</tr>
<tr>
<td>Layer[0] Cuivre signal</td>
<td>layer name and type name = name given to the layer by the user (here: &quot;cuivre&quot;) type = signal (not current used in Pcbnew)</td>
</tr>
<tr>
<td>Layer[15] Composant signal</td>
<td></td>
</tr>
<tr>
<td>TrackWidth 250</td>
<td>Current track width</td>
</tr>
<tr>
<td>TrackWidthHistory 25</td>
<td>Last used track widths</td>
</tr>
<tr>
<td>TrackWidthHistory 170</td>
<td></td>
</tr>
<tr>
<td>TrackWidthHistory 250</td>
<td></td>
</tr>
<tr>
<td>TrackWidthHistory 400</td>
<td></td>
</tr>
<tr>
<td>TrackClearence 110</td>
<td>Isolation for DRC (Design rules check)</td>
</tr>
<tr>
<td>ZoneClearence 200</td>
<td>Isolation used in zone filling</td>
</tr>
<tr>
<td>DrawSegmWidth 120</td>
<td>Current segment width for drawings on technical layers</td>
</tr>
<tr>
<td>EdgeSegmWidth 120</td>
<td>Current segment width for drawings on &quot;edge layer&quot;</td>
</tr>
<tr>
<td>ViaSize 700</td>
<td>Current via size</td>
</tr>
<tr>
<td>ViaDrill 250</td>
<td>Via drill for this board</td>
</tr>
<tr>
<td>ViaSizeHistory 450</td>
<td></td>
</tr>
<tr>
<td>ViaSizeHistory 650</td>
<td>Last used via sizes</td>
</tr>
<tr>
<td>ViaSizeHistory 700</td>
<td></td>
</tr>
<tr>
<td>TextPcbWidth 120</td>
<td>Current text width for texts on copper or technical layers. This is not for text on footprints</td>
</tr>
</tbody>
</table>
```
3.9 - $EQUIPOT

$EQUIPOT describes a net name.

<table>
<thead>
<tr>
<th>$EQUIPOT</th>
<th>Start block</th>
</tr>
</thead>
<tbody>
<tr>
<td>Na 2 &quot;N-000026&quot;</td>
<td>Na &lt;internal net number&gt; « net name »</td>
</tr>
<tr>
<td>St ~</td>
<td></td>
</tr>
<tr>
<td>$EndEQUIPOT</td>
<td>End block</td>
</tr>
</tbody>
</table>

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

3.10 - $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.

3.10.1 - General description:

<table>
<thead>
<tr>
<th>Field</th>
<th>Units</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>field number</td>
<td>enumeration</td>
<td>0=&gt;reference, 1=&gt;value, etc.</td>
</tr>
<tr>
<td>Xpos</td>
<td>tenths of mils (.0001 inches)</td>
<td>The horizontal offset relative to the module's overall position</td>
</tr>
<tr>
<td>Ypos</td>
<td>tenths of mils (.0001 inches)</td>
<td>The vertical offset relative to the module's overall position</td>
</tr>
<tr>
<td>Xsize</td>
<td>tenths of mils (.0001 inches)</td>
<td>The horizontal size of the character 'M'</td>
</tr>
<tr>
<td>Ysize</td>
<td>tenths of mils (.0001 inches)</td>
<td>The vertical size of the character 'M'</td>
</tr>
<tr>
<td>rotation</td>
<td>tenths of degrees</td>
<td>Angular rotation from horizontal, counterclockwise</td>
</tr>
<tr>
<td>penWidth</td>
<td>tenths of mils (.0001 inches)</td>
<td>Width of the pen used to draw characters</td>
</tr>
<tr>
<td>N</td>
<td>none</td>
<td>flag for the parser?</td>
</tr>
<tr>
<td>visible</td>
<td>boolean</td>
<td>I=&gt; invisible, V=&gt; visible</td>
</tr>
<tr>
<td>layer</td>
<td>enumeration</td>
<td>see layer numbers above</td>
</tr>
</tbody>
</table>

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

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

3.10.2 - 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"
3.10.3 - Drawings:
Tells how to draw module shape.
They cannot be on a copper layer (DRC ignore them)
Drawings are segment, circle, arc, polygon.

3.10.3.1 - Draw segment:

| DS -6000 -1500 -6000 1500 120 21 | DS is a Draw Segment
|---------------------------------|-----------------------------------
| DS 6000 1500 6000 -1500 120 21  | An other Draw Segment

3.10.3.2 - Circle:

<table>
<thead>
<tr>
<th>DC Xcentre Ycentre Xpoint Ypoint Width Layer</th>
</tr>
</thead>
</table>
| DC is a Draw Circle \nXpoint, Ypoint is a point on the circle.

3.10.3.3 - Arc:

<table>
<thead>
<tr>
<th>DA Xcentre Ycentre Xstart_point Ystart_point angle width layer</th>
</tr>
</thead>
</table>
| DA is a Draw Arc \nangle is the arc angle in 0.1 degrees

3.10.3.4 - Polygon:

<table>
<thead>
<tr>
<th>DP 0 0 0 0 corners_count width layer</th>
</tr>
</thead>
</table>
| Draw Polygon \nThe polygon should be closed, otherwise this is a \npoly-line. \nwidth is the thickness of outlines.
| Di corner_posx corner_posy |
| Corner coordinate \n( corners_count lines like this)

3.10.4 - Pad Descriptions:
All the pads of this footprint are listed here (Many $PAD/$EndPAD sections here)..
See $PAD description.

3.10.5 - $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.

|$SHAPE3D$ |
|-----------------|-------------------|
| Na "device/bornier_6.wrl" | FileName (default path is kicad/modules/packages3d/)
| Sc 1.000000 1.000000 1.000000 | X Y Z scale factor
| Of 0.000000 0.000000 0.000000 | X Y Z offset (move vector, in 3D units (0.1 inch))
| Ro 0.000000 0.000000 0.000000 | X Y Z rotation (in degree)
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).

### 3.10.6 - $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

<table>
<thead>
<tr>
<th>$PAD</th>
<th>Start description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sh &quot;2&quot; C 1500 1500 0 0 2700</td>
<td>Shape: &lt;pad name&gt; shape Xsize Ysize Xdelta Ydelta Orientation</td>
</tr>
<tr>
<td>Dr 600 0 0</td>
<td>Drill &lt;Pad drill&gt; Xoffset Yoffset (round hole)</td>
</tr>
<tr>
<td>Dr 600 0 0 O 600 650</td>
<td>Drill &lt;Pad drill.x&gt; Xoffset Yoffset &lt;Hole shape&gt; &lt;Pad drill.x&gt; &lt;Pad drill.y&gt;</td>
</tr>
<tr>
<td>At STD N 00E0FFFF</td>
<td>Attributes: &lt;Pad type&gt; N &lt;layer mask&gt;</td>
</tr>
<tr>
<td>Ne 8 &quot;GND&quot;</td>
<td>Net reference of the pad: &lt;netnumber&gt; &lt;net name&gt;</td>
</tr>
<tr>
<td>Po -3000 0</td>
<td>X_pos Y_pos (relative to the module position)</td>
</tr>
<tr>
<td>$EndPAD</td>
<td>End description</td>
</tr>
</tbody>
</table>

Note:
```
$PAD
Sh "2" C 1500 1500 0 0 2700
Sh
ape: <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
Attributes: <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
```

### 3.11 - Graphic items:
There are drawing items like segments, circles, texts, targets and cotations.

### 3.11.1 - $DRAWSEGMENT

Draw segments are:
- segments (strait line)
- circles
- arcs

#### 3.11.1.1 - Line:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$DRAWSEGMENT</td>
<td>Start description</td>
</tr>
<tr>
<td>Po 0 67500 39000 65500 39000 120</td>
<td>Position shape Xstart Ystart Xend Yend width</td>
</tr>
<tr>
<td>De 28 0 900 0 0</td>
<td>Description layer type angle timestamp status</td>
</tr>
<tr>
<td>$EndDRAWSEGMENT</td>
<td>End description</td>
</tr>
</tbody>
</table>

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

#### 3.11.1.2 - Circle:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$DRAWSEGMENT</td>
<td>Start description</td>
</tr>
<tr>
<td>Po 1 67500 39000 65500 39000 120</td>
<td>Position shape Xcentre Ycentre Xend Yend width</td>
</tr>
<tr>
<td>De 28 0 900 0 0</td>
<td>Description layer type angle timestamp status</td>
</tr>
<tr>
<td>$EndDRAWSEGMENT</td>
<td>End description</td>
</tr>
</tbody>
</table>

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)

#### 3.11.2 - Arc:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$DRAWSEGMENT</td>
<td>Start description</td>
</tr>
<tr>
<td>Po 2 67500 39000 65500 39000 120</td>
<td>Position shape Xstart Ystart Xend Yend width</td>
</tr>
<tr>
<td>De 28 0 900 0 0</td>
<td>Description layer type angle timestamp status</td>
</tr>
<tr>
<td>$EndDRAWSEGMENT</td>
<td>End description</td>
</tr>
</tbody>
</table>

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
```

#### 3.11.3 - $TEXTPCB

**Example:** TDI
 Files Formats

Board File Format (Format versions 1 and 2)

KiCad

$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 "TDI"
Po 57250 35750 600 600 150 0
De 15 1 B98C Normal
$EndTEXTPCB

3.11.4 - $MIRE

| shape 1
| shape 0 |

$MIREPCB
Start description

Po 0 28 28000 51000 5000 150 0
Position shape Xpos Ypos size width timestamp

$EndMIREPCB
End description

3.11.5 - $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
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

Coordinates of segments (axis, arrows...)

$EndCOTATION
End description

3.12 - Track, vias and Zone section:
### 3.12.1 - $\text{TRACK}$

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) and 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 routingInfos).

```
$\text{TRACK}$
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
De 15 0 1 0 0
An other track

Po 3 53500 27000 53500 27000 650
De 15 1 14 0 0
This is a via (via "through") from layer 15 (component) to layer 0 (copper)

$\text{EndTRACK}$
End description
```

### 3.12.2 - $\text{ZONE}$

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.

```
$\text{ZONE}$
Start description

Po 0 67100 33700 67100 38600 100
Same as track description

De 0 0 2 3EDDB09D 0

$\text{EndZONE}$
End description
```

### 3.12.3 - $\text{CZONE\_OUTLINE}$

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:**

```
$\text{CZONE\_OUTLINE}$
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

---

Other example:

$CZONE_OUTLINE  
ZInfo 47B3E800 3 "VCC"  
ZLayer 1  
ZAux 8 F  
ZClearance 200 T  
ZMinThickness 190  
ZOptions 0 32 F 200 200  
ZCorner 49704 23032 0  

$POLYSCORNERS  
74655 51655 0 0  
74655 13345 0 0  

$endPOLYSCORNERS  
$endCZONE_OUTLINE

---

### Files Formats

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>ZInfo</td>
<td>Time stamp &lt;internal netcode&gt; &quot;net name&quot;</td>
<td>&lt;Time stamp&gt; &lt;internal netcode&gt; &quot;net name&quot;</td>
</tr>
<tr>
<td>ZLayer</td>
<td>Layer (0 = copper, 15 = component, 1 ..14 = inner layers)</td>
<td>Layer (0 = copper, 15 = component, 1 ..14 = inner layers)</td>
</tr>
<tr>
<td>ZClearance</td>
<td>&lt;Zone clearance&gt; &lt;pads option = I, T or X&gt; I = pads in zone T = Thermal reliefs X = pads not in zone.</td>
<td>&lt;Zone clearance&gt; &lt;pads option = I, T or X&gt; I = pads in zone T = Thermal reliefs X = pads not in zone.</td>
</tr>
<tr>
<td>ZMinThickness</td>
<td>Zone min thickness (for copper zone)</td>
<td>Zone min thickness (for copper zone)</td>
</tr>
<tr>
<td>ZOptions</td>
<td>&lt;fill mode&gt; &lt;arc approx&gt; &lt;antipad thickness&gt; &lt;thermal stubs width&gt; fill mode = 0 (use solid polygons) or 1 (use segments) arc approx = 16 or 32 (segments count to approximate a 360 arc)</td>
<td>&lt;fill mode&gt; &lt;arc approx&gt; &lt;antipad thickness&gt; &lt;thermal stubs width&gt; fill mode = 0 (use solid polygons) or 1 (use segments) arc approx = 16 or 32 (segments count to approximate a 360 arc)</td>
</tr>
<tr>
<td>ZCorner</td>
<td>First corner (external outline)</td>
<td>First corner (external outline)</td>
</tr>
<tr>
<td>ZCorner</td>
<td>Next corner</td>
<td>Next corner</td>
</tr>
<tr>
<td>ZCorner</td>
<td>End corner (flag = 1)</td>
<td>End corner (flag = 1)</td>
</tr>
<tr>
<td>$POLYSCORNERS</td>
<td>Start of filled areas outlines</td>
<td>Start of filled areas outlines</td>
</tr>
<tr>
<td>$endPOLYSCORNERS</td>
<td>First corner (first filled area outline)</td>
<td>First corner (first filled area outline)</td>
</tr>
<tr>
<td>$endCZONE_OUTLINE</td>
<td>End description</td>
<td>End description</td>
</tr>
</tbody>
</table>

---
Files Formats  Board File Format (Format versions 1 and 2)  KiCad

| 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 |
| ZCorner 48450 19900 0 |
| ZCorner 48450 20800 0 |
| ZCorner 47350 20800 0 |
| ZCorner 47250 19900 1 |
| $endCZONE_OUTLINE |

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

4 - Pcbnew “S-expression” file format

Headings

4 - Pcbnew “S-expression” file format ................................................................. 26
  4.1 - File format syntax ................................................................. 26
  4.2 - Coordinates of objects and sizes .............................................. 26
  4.3 - Keywords ................................................................. 27
  4.4 - Identifiers and Strings ............................................................. 27
  4.5 - Layer representation in files ................................................... 27
    4.5.1 - Layer capacity .......................................................... 27
    4.5.2 - Layer names in files .................................................. 27
  4.6 - Typical structure of the board file ........................................... 28
    4.6.1 - Description of an item ................................................ 28
    4.6.2 - The header line .......................................................... 28
    4.6.3 - The general section ..................................................... 28
    4.6.4 - The layers section (the mapping of layers) ....................... 28
    4.6.5 - The setup section ...................................................... 29
    4.6.6 - The list of nets .......................................................... 29
    4.6.7 - The list of net classes ................................................ 29
    4.6.8 - The list of modules ..................................................... 30
    4.6.9 - The list of graphic items .............................................. 30
    4.6.10 - The list of tracks ..................................................... 30
    4.6.11 - The list of zones ...................................................... 30
    4.6.12 - Description of a module (footprint) .............................. 31
  4.7 - Board Example ........................................................................ 31

4.1 - File format syntax:
Based on the Spectra syntax.
All tokens are lowercase.
Strings such as board text and copper layer names can have upper case characters, but these will often be quoted.
A special emphasis to readability considerations is made.

4.2 - Coordinates of objects and sizes:
Coordinates are relative to the origin of their containing object.
Values are given in millimeters.
Exponential floating point values are not allowed and instead are presented as:
"xxx.yyy" or "0.0x" or "0.x", or even "xy". This was done because values like 4e-2 are not as easy for a human to read.

4.3 - Keywords:
Use only lowercase ASCII words (and trailing digits if needed). ASCII characters are <= 127.

4.4 - Identifiers and Strings:
Identifiers are variables used within the file such as layer names, net names, etc. Strings are longer text sequences such as drawing labels. They are handled the same, and will be referred to as strings henceforth. From a purely syntactical perspective (and ignoring any limits imposed at a higher level by the user of the string), a string can contain embedded spaces and tabs, but may not contain an actual newline. A string is to be encoded in UTF8 format, meaning that it may be ASCII or international characters sequences, since ASCII is a subset of UTF8.

Note that this excludes LATIN characters >= 128.

To encode LATIN characters, a UTF8 sequence must be used.
A string may not contain an actual newline or carriage return, but it may use an escape sequence to encode a newline, such as \n.
If a string has any of the following conditions, then it must be quoted with a leading and trailing double quote character, otherwise it is acceptable to not quote the string:

1. has one or more of the following 4 separator bytes: ASCII space, tab, '{', or '}'.
2. has one or more of the following bytes: '%', '{', or '}'.
3. has a length of zero bytes, and you need a place holder for the field, then use "".
4. includes a byte of '-', and this byte is not in the first position of the string.

Examples:

• If the field has embedded spaces, tabs, '{' or '}', then it must be quoted like these samples: "this is a sample", "con(14)", "(19".
• If the field has an embedded #, then is must be quoted: "wire#123"
• This string does not need to be quoted: -CDC, but this one does: "C-DC"
• If your string needs to convey multiple lines, escape the new line character like this: "line 1\nline 2".
• Here is a legal string with an embedded quote: "leg"23
• Here is the same string quoted, and because it is quoted the internal quote must be duplicated: "leg""23"
• Here is a string quoted that does not need to be but is acceptable anyways: "R1"

4.5 - Layer representation in files

4.5.1 - Layer capacity:
- Copper layers 16
- Technical layers 13 ( 8 paired layers, 4 user layers and 1 layer for board edges )

4.5.2 - Layer names in files
In files the layers have a name, not a number.

Copper layers:
For copper layers the name can be set by the user.
A copper layer name is defined as <layer name set by user>.Cu
However, for pads and vias which are on all copper layers, the full set of copper layers is defined as *.Cu.

Paired technical layers:
The name is fixed and built on the form.
B.<layer name> for a layer on the back side of the board.
F.<layer name> for a layer on the front side of the board.
The layer name is one of
Adhes Paste Paste SilkS Mask.
Or the translated name of these layers for non English users.
Like for Copper layers, *.<layer name> can be used to represent the 2 paired layers

Other layers:
The name is <layer name>. User like:


Or the translated name of these layers for non English users.

**Board outlines:**

the name is *Edge.Cuts*

### 4.6 - Typical structure of the board file:

A board file includes different sections and list of board items:

- The header line
- The general section
- The layers section (the mapping of layers)
- The setup section
- The list of nets
- The list of net classes
- The list of modules
- The list of graphic items
- The list of tracks
- The list of zones

The order of lists is not critical, and some sections can be omitted.

#### 4.6.1 - Description of an item:

The basic item syntax is an opening brace “(“ followed by a keyword with one or more parameters associated with the keyword separated by one or more spaces followed by a closing brace “)” . Items can be nested but the opening and closing braces must be symmetrical.

An item is described by

( <keyword>  <parameter> )

Examples:

- *(via_drill 0.635)*
- *(area 57.924999 28.924999 74.075001 42.075001)*

#### 4.6.2 - The header line

*(kicad pcb (version 3) (host pcbnew "(2013-02-20 BZR 3963)-testing")*

#### 4.6.3 - The general section

*(general)*

* (links 2)
  *(no_connects 0)*
  *(area 57.924999 28.924999 74.075001 42.075001)*
  *(thickness 1.6)*
  *(drawings 5)*
  *(tracks 5)*
  *(zones 0)*
  *(modules 2)*
  *(nets 3)*

*(page A4)*

#### 4.6.4 - The layers section (the mapping of layers)

*layers*

* (15 top_side.Cu signal)
 (2 Inner2.Cu signal)
 (1 Inner1.Cu signal)
 (0 bottom_side.Cu signal)
 (16 B.Adhes user)
 (17 F.Adhes user)
 (18 B.Paste user)
 (19 F.Paste user)
 (20 B.SilkS user)
 (21 F.SilkS user)
 (22 B.Mask user)
This is an important section, because it defines the active layers, the layer types and attributes, the copper layer names (set by the user) and the numerical identifier used to associate the user defined layer names with the Pcbnew internal layer definition. All subsequent layer references are by name only.

### 4.6.5 - The setup section

```plaintext
(setup
   (last_trace_width 0.254)
   (trace_clearance 0.254)
   (zone_clearance 0.2)
   (zone45_only no)
   (trace_min 0.254)
   (segment_width 0.2)
   (edge_width 0.15)
   (via_size 0.889)
   (via_drill 0.635)
   (via_min_size 0.889)
   (via_min_drill 0.508)
   (uvia_size 0.508)
   (uvia_drill 0.127)
   (uvias_allowed no)
   (uvia_min_size 0.508)
   (uvia_min_drill 0.127)
   (pcb_text_width 0.3)
   (pcb_text_size 1.5 1.5)
   (mod_edge_width 0.15)
   (mod_text_width 0.15)
   (mod_text_size 1.5 1.5)
   (via_min_width 0.254)
   (pcbplotparams
      (layerselection 3178497)
      (usegerberextensions true)
      (excludedgelayer true)
      (linewidth 50000)
      (plotframeref false)
      (viasonmask false)
      (mode 1)
      (useauxorigin false)
      (hpglpenspeed 20)
      (hpglpenwidth 15)
      (hpglpenoverlay 2)
      (plotreference true)
      (plotvalue true)
      (plotothertext true)
      (plotinvisibletext false)
      (padonmask false)
      (subtractmaskfromsilk false)
      (chiponmask true)
      (mirror false)
      (drillshape 1)
      (scaleselection 1)
      (outputfile "")
   )
 )
```

This section stores the current settings (default item sizes) and options in use for this board.

### 4.6.6 - The list of nets

```plaintext
(net 0 "")
(net 1 /SIGNAL)
(net 2 GND)
```

This section includes the list of nets read from the schematic netlist. Each net has a net number and a name if in the schematic the net has a label.

### 4.6.7 - The list of net classes

```plaintext
(net_class Default "Ceci est la Netclass par défaut"
   (clearance 0.254)
   (trace_width 0.254)
   (via_dia 0.889)
   (via_drill 0.635)
   (uvia_dia 0.508)
   (uvia_drill 0.127)
   (add_net "")
)```
4.6.8 - The list of modules

(module R3 (layer top_side.Cu) (tedit 4E4C0E65) (tstamp 5127A136)
  (at 66.04 33.502)
  (tags R)
  (path /5127A011)
  (autoplace_cost180 10)
  (descr "Resistance 3 pas")
  (fp_text reference R1 (at 0 0.127) (layer F.SilkS) hide
    (effects (font (size 1.397 1.27) (thickness 0.2032))))
  (fp_text value 330K (at 0 0.127) (layer F.SilkS)
    (effects (font (size 1.397 1.27) (thickness 0.2032))))
  (fp_line (start -3.81 0) (end -3.302 0) (layer F.SilkS) (width 0.2032))
  (fp_line (start 3.81 0) (end 3.302 0) (layer F.SilkS) (width 0.2032))
  (fp_line (start 3.302 0) (end 3.302 -1.016) (layer F.SilkS) (width 0.2032))
  (fp_line (start 3.302 -1.016) (end -3.302 -1.016) (layer F.SilkS) (width 0.2032))
  (pad 1 thru_hole circle (at -3.81 0) (size 1.397 1.397) (drill 0.812799)
    (layers *.Cu *.Mask F.SilkS)
    (net 1 /SIGNAL))
  (pad 2 thru_hole circle (at 3.81 0) (size 1.397 1.397) (drill 0.812799)
    (layers *.Cu *.Mask F.SilkS)
    (net 2 GND))
  (model discret/resistor.wrl
    (at (xyz 0 0 0))
    (scale (xyz 0.3 0.3 0.3))
    (rotate (xyz 0 0 0)))
)

This is the description of all modules (footprints) on the board.

4.6.9 - The list of graphic items

(gr_text TEST (at 62 31) (layer top_side.Cu)
  (Effects (font (size 1.1 1.5) (thickness 0.3))))
)

This is the list of “graphical” items on the board.

4.6.10 - The list of tracks

(segment (start 61.0616 36.8808 (end 61.0616 34.5186) (width 0.254) (layer bottom_side.Cu) (net 1))
(segment (start 61.0616 34.5186 (end 69.85 33.3502) (width 0.254) (layer bottom_side.Cu) (net 1) (tstamp 5127A159))
(segment (start 69.85 33.3502 (end 70.993 33.3502) (width 0.5) (layer bottom_side.Cu) (net 2))
(segment (start 71.2216 33.5788 (end 71.2216 36.8808) (width 0.5) (layer bottom_side.Cu) (net 2) (tstamp 5127A156))
(segment (start 70.993 33.3502 (end 71.2216 33.5788) (width 0.5) (layer bottom_side.Cu) (net 2) (tstamp 5127A155))

This is the list of tracks and vias (obviously, only on copper layers) on the board.

4.6.11 - The list of zones

(zone (net 2) (net_name GND) (layer bottom_side.Cu) (tstamp 5127A1B2) (hatch edge 0.508)
  (connect_pads (clearance 0.2))
  (min_thickness 0.1778)
  (fill (arc_segments 16) (thermal_gap 0.254) (thermal_bridge_width 0.4064))
  (polygon
    (pts
      (xy 59 30) (xy 73 30) (xy 73 41) (xy 59 41)))
)

This section stores the net classes setup. Each netclass has a set of track and via and size and clearance settings and the name of net or nets assigned to the net class.
4.6.12 - Description of a module (footprint)

Here is an example:

```s-expression
(module R3 (layer top_side.Cu) (tedit 4E4C0E65) (tsnap 5127A136)
  (at 66.04 33.3502)
  (desc "Resistance 3 pas")
  (tags R)
  (path /5127A011)
  (autoplace_cost180 10)
  (fp_text reference R1 (at 0 0.127) (layer F.SilkS) hide
  (effects (font (size 1.397 1.27) (thickness 0.2032)))
  (fp_text value 330K (at 0 0.127) (layer F.SilkS)
  (effects (font (size 1.397 1.27) (thickness 0.2032)))
  (fp_line (start -3.81 0) (end -3.302 0) (layer F.SilkS) (width 0.2032))
  (fp_line (start -3.81 0) (end -3.302 0) (layer F.SilkS) (width 0.2032))
  (pad 1 thru_hole circle (at -3.81 0) (size 1.397 1.397) (drill 0.812799)
  (layers *.Cu *.Mask F.SilkS)
  (net 1 /SIGNAL)
  (pad 2 thru_hole circle (at 3.81 0) (size 1.397 1.397) (drill 0.812799)
  (layers *.Cu *.Mask F.SilkS)
  (model discret/resistor.wrl
    (at (xyz 0 0 0))
    (scale (xyz 0.3 0.3 0.3))
    (rotate (xyz 0 0 0)))
)
```

A module has:
- a reference
- a layer (Front or Back layer)
- a last edition time stamp (for user info)
- a time stamp from the schematic
- a position.

Its description includes:
- Text (at least reference and value)
- Graphic outlines
- Pads (with pad type, pad layers, pad size and position, net)
- A link to a 3D model, if exists, for the 3D viewer.

### 4.7 - Board Example:

```s-expression
(kicad_pcb (version 3) (host pcbnew "(2013-02-20 BZR 3963)-testing")

  (general
    (links 2)
    (no_connects 0)
    (area 57.924999 28.924999 74.075001 42.075001)
    (thickness 1.6)
    (drawings 5)
    (tracks 5)
    (zones 0)
    (modules 2)
    (nets 3)
  )

  (page A4)
  (layers
    (15 top_side.Cu signal)
    (2 Inner2.Cu signal)
    (1 Inner1.Cu signal)
    (0 bottom_side.Cu signal)
    (16 B.Adhes user)
    (17 F.Adhes user)
    (18 B.Paste user)
    (19 F.Paste user)
    (20 B.SilkS user)
    (21 F.SilkS user)
    (22 B.Mask user)
    (23 F.Mask user)
    (24 Dwgs.User user)
  )
```

Pcbnew “S-expression” file format
(25 Cmts.User user)
(26 Ecol1.User user)
(27 Eco2.User user)
(28 Edge.Cuts user)
)

(setup
(last_trace_width 0.254)
(trace_clearance 0.254)
(zone_clearance 0.2)
(zone_45_only no)
(trace_Mn 0.254)
(segment_width 0.2)
(edge_width 0.15)
(via_size 0.889)
(via_drill 0.635)
(via_min_size 0.889)
(via_min_drill 0.508)
(uvia_size 0.508)
(uvia_drill 0.127)
(uvias_allowed no)
(uvia_min_size 0.508)
(uvia_min_drill 0.127)
(pcb_text_width 0.3)
(pcb_text_size 1.5 1.5)
(mod_edge_width 0.15)
(mod_text_size 1.5 1.5)
(mod_text_width 0.15)
(pad_size 0.0005 0.0005)
(pad_drill 0)
(pad_to_mask_clearance 0.2)
(aux_axis_origin 0 0)
(white_elements ?FFFFFF)
(pcbplotparams
(layerselection 3178497)
(usegerberextensions true)
(excludeedgelayer true)
(linewidth 50000)
(plotframeref false)
(viasomask false)
(mode 1)
(useauxorigin false)
(hpglpenumber 1)
(hpglpenumber 20)
(hpglpendiameter 15)
(hpglpenumber 2)
(psnegative false)
(psa4output false)
(plotreference true)
(plotvalue true)
(plotothertext true)
(plot invisibletext false)
(padsonsilk false)
(subtractmaskfromsilk false)
(outputformat 1)
(mirror false)
(drillshape 1)
(scaleselection 1)
(outputdirectory "")
)

(net 0 **)
(net 1 /SIGNAL)
(net 2 GND)

(net_class Default "Ceci est la Netclass par dACfaut"
(clearance 0.254)
(trace_width 0.254)
(via dia 0.889)
(via_drill 0.635)
(uvia dia 0.508)
(uvia drill 0.127)
(add net "")
(add net /SIGNAL)
)

(net_class POWER ""
(clearance 0.254)
(trace_width 0.5)
(via dia 1.2)
(via_drill 0.635)
(uvia dia 0.508)
(uvia drill 0.127)
(add net GND)
)

(module R3 (layer top_side.Cu) (tedit 4E4CE65) (tstamp 5127A136)
(at 66.04 33.3502)
(descr "Resistance 3 pas")
(tags R)
(path /5127A011)
(autoplacement_cost 180 10)
(fp_text reference R1 (at 0 0.127) (layer F.SilkS) hide
(effects (font (size 1.397 1.27) (thickness 0.2032)))

Pcbnew “S-expression” file format

KiCad

page 32/40
Pcbnew “S-expression” file format

KiCad

Files Formats

Pcbnew “S-expression” file format

page 33/40
5 - Page layout file format:

Headings

5.1 - General
This document explains the file format used to describe the page layout and the title block of schematic and printed circuits pages.

5.2 - Units and coordinates
Sizes and coordinates are given in mm
Coordinates are always relative to a page corner (called the reference corner).
Some items use two coordinates. Each coordinate has its own reference corner

5.3 - basic items
Page layout use different basic items:
- Segment (two coordinates)
- Rectangle (two coordinates)
- Text (one coordinate)
- Poly-polygon (one coordinate)
Items can be repeated n times, and items having one coordinate can be rotated.
Texts are defined both by the page layout file (fixed text) and by user, at run time (date, title...).
Therefore texts can include format descriptors, which will be replaced by the corresponding string, inside a project.

5.4 - Coordinates and reference corners
Each coordinates is relative to a reference corner
The direction of coordinates depends on this reference corner:
5.5 - Items options

<table>
<thead>
<tr>
<th>Item</th>
<th>coordinates</th>
<th>line thickness</th>
<th>Rotation (in degree)</th>
<th>Repeat (X or/and Y)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Line (segment)</td>
<td>2</td>
<td>Yes or default</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Rectangle</td>
<td>2</td>
<td>Yes or default</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Text</td>
<td>1</td>
<td>Yes or default</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Poly polygon</td>
<td>1</td>
<td>Yes (default = 0)</td>
<td>Yes</td>
<td>Yes</td>
</tr>
</tbody>
</table>

5.6 - Keywords used in description

<table>
<thead>
<tr>
<th>page_layout</th>
<th>pos</th>
<th>justify</th>
</tr>
</thead>
<tbody>
<tr>
<td>setup</td>
<td>start</td>
<td>left</td>
</tr>
<tr>
<td>left_margin</td>
<td>end</td>
<td>center</td>
</tr>
<tr>
<td>right_margin</td>
<td>pts</td>
<td>right</td>
</tr>
<tr>
<td>top_margin</td>
<td>xy</td>
<td>top</td>
</tr>
<tr>
<td>bottom_margin</td>
<td>maxlen</td>
<td>bottom</td>
</tr>
<tr>
<td>linewidth</td>
<td>maxheight</td>
<td>rotate</td>
</tr>
<tr>
<td>textlinewidth</td>
<td>font</td>
<td>repeat</td>
</tr>
<tr>
<td>textsize</td>
<td>bold</td>
<td>incr</td>
</tr>
<tr>
<td>comment</td>
<td>italic</td>
<td>incry</td>
</tr>
<tr>
<td>line</td>
<td>size</td>
<td>incrlabel</td>
</tr>
<tr>
<td>rect</td>
<td>ltcornel</td>
<td></td>
</tr>
<tr>
<td>polygon</td>
<td>rtcorner</td>
<td></td>
</tr>
<tr>
<td>tbtext</td>
<td>lbcorner</td>
<td></td>
</tr>
<tr>
<td>name</td>
<td>rbcorner</td>
<td></td>
</tr>
</tbody>
</table>

5.7 - Comments

Comments are allowed. They are inside (), and start by the keyword comment.

Example:
(comment rect around the title block)

5.8 - Coordinates
A coordinate is the X pos, the Y pos, and the reference corner (coordinate origin)
the default is the bottom right corner
example:
\((\text{start } 10 0 \ \text{lcorner}) \text{ or } (\text{start } 20 10)\)
The direction depends on the corner: a positive coordinate define a point from the corner origin, to the opposite
corner.

5.9 - Items
Items are defined by a name, coordinates in mm and some attributes, and can be repeated.
for instance (\text{repeat} 2) (\text{incrx} 2) (\text{incry} 2) repeat the item 2 times, and coordinates are incremented by 2 on X
direction, and 2 on Y direction

5.10 - Header and setup
\(\text{(page\_layout} \\
    (\text{setup} (\text{textsize} 1.5 1.5) (\text{linewidth} 0.15) (\text{textlinewidth} 0.15)) \) \\
.....

5.11 - Line and rect
Lines and rectangles are defined by 2 coordinates start and end, and attributes.
Attributes are linewidth and repeat parameters.
Example:
\(\text{(line} (\text{start } 50 2 \ \text{lcorner}) (\text{end } 50 0 \ \text{lcorner}) (\text{repeat} 30) (\text{incrx} 50) \) \\
(\text{rect} (\text{comment rect around the title block}) (\text{linewidth} 0.15) (\text{start} 110 34) (\text{end} 2 2) \)

5.12 - Texts

5.12.1 - Basic texts
Texts are defined by the text (between quotes), the position, and optional attributes
Example
\(\text{(tbtext} "1" (\text{pos} 25 1 \ \text{lcorner}) (\text{font} (\text{size} 1.3 1.3)) (\text{repeat} 100) (\text{incrx} 50) \) (\text{inclabel} 1) \\
the text can be rotated by (\text{rotation} <\text{value}>) with value = rot angle in degrees, and the initial text value will be “2” “3” 
...

5.12.2 - Multi-line texts
Texts can include more than one line.
In files the new line symbol is the 2 chars sequence “\n”, like in C printf format: \\n’ followed by ‘\’.
Example:
\(\text{(tbtext} "\text{Multi line text\nline 2\nline 3\nline 4}" (\text{name text18:Text}) (\text{pos} 90.3 29)) \\
is the text:
\text{Multi line text} \\
\text{line 2} \\
\text{line 3} \\
\text{line 4} \\
Note also it is possible to insert a “\n” string in the actual text by inserting the “\\n” sequence, and a ‘\’ followed by ‘\’ 
sequence is converted to a single ‘\’ char.

5.12.3 - Font option
Font option is
\(\text{(font} <\text{size} <\text{size} X> <\text{size} Y>) <\text{bold} <\text{italic}>)} \\
Example \\
\(\text{(font} (\text{size} 1.3 1.3) \text{bold italic}) \) defines a specific size, with both bold and italic options

5.12.4 - Justification
\(\text{(justify} <\text{justif keyword}> \) controls the text justification (the default is left on X and centered on Y) 
justif keywords are center, left, right, top and bottom
(justify center top) is a text centered on X axis and top aligned on vertical axis

If center is used, it should be the first option, because it forces both horizontal and vertical justification to be centered.

5.12.5 - Constrained text

The text size can be constrained:

\(\text{maxlen <value>}\) and \(\text{maxheight <value>}\) force the actual text x and/or y size to be reduced to limit the text height to the maxheight value, and the full text x size to the maxlen value.

If the actual text size is smaller than limits, its size is not modified.

5.12.6 - Format symbols

Texts can include a format symbol, a la printf.

At run time these format symbols will be replaced by their actual value.

Known format symbols are:

<table>
<thead>
<tr>
<th>Format Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>%</td>
<td>replaced by %</td>
</tr>
<tr>
<td>%K</td>
<td>KiCad version</td>
</tr>
<tr>
<td>%Z</td>
<td>paper format name (A4, USLetter ...)</td>
</tr>
<tr>
<td>%Y</td>
<td>company name</td>
</tr>
<tr>
<td>%D</td>
<td>date</td>
</tr>
<tr>
<td>%R</td>
<td>revision</td>
</tr>
<tr>
<td>%S</td>
<td>sheet number</td>
</tr>
<tr>
<td>%N</td>
<td>number of sheets</td>
</tr>
<tr>
<td>%Cx</td>
<td>comment (x = 0 to 9 to identify the comment)</td>
</tr>
<tr>
<td>%F</td>
<td>File name</td>
</tr>
<tr>
<td>%P</td>
<td>sheet path (sheet full name)</td>
</tr>
<tr>
<td>%T</td>
<td>title</td>
</tr>
</tbody>
</table>

Example:

\((\text{tbtext Size: }%Z\ldots)\) displays something like "Size A4" or Size USLetter", depending on the selected page format.

5.13 - Poly Polygons

Set of filled polygons are supported.

The main purpose is to allow logos, or complex shapes

They support the repeat and rotation options

They are defined by

\((\text{polygon (position ... <rotation> <linewidth> <list of corners>})\)

the parameter linewidth defines the pen size used to draw/plot the polygon outlines (default = 0)

example:

\((\text{polygon (pos 134 18 rcorner) (rotate 20) (linewidth 0.00254)})\)

The list of corners is like


each sequence like


defines a polygon.

Each coordinate is relative to the polygon position.

Therefore a "polygon" is in fact a set of polygons, of a poly polygon

5.14 - File format

5.14.1 - Header section

A layout description starts by

\((\text{page_layout})\)
and ends by the corresponding closing parenthesis
)

5.14.2 - Setup section
Some default values for parameters can be defined in the setup section:
They are the default text size, the default linewidth, the default textlinewidth and margins.
Example:
( setup (textsize 1.5 1.5) (linewidth 0.15) (textlinewidth 0.15) )
(left_margin 10)(right_margin 10)(top_margin 10)(bottom_margin 10))
If these default values not defined here, Kicad will use internal default values.

5.14.3 - Data section
It contains the list of items (lines, rectangles, texts and poly polygons) to draw.

5.14.3.1 - Rect item
Here is a simple example:
( rect (comment rect around the title block) (linewidth 0.15) (start 110 34) (end 2 2) )
Here is a more complex example:
( rect (start 0 0 ltcorner) (end 0 0 rbcorner) (repeat 2) (incrx 2) (incry 2) )
2 rectangle will be drawn.
The first is drawn from the left top corner of the page, to the right bottom corner
The second is drawn inside the first (from 2 mm of the left top corner of the page, to 2 mm from the right bottom
corner of the page)

5.14.3.2 - Line item
Here is a simple example:
( line (start 110 5.5) end 2 5.5) )
This is a horizontal line near the right bottom corner (default), using the default line thickness.
Here is a more complex example:
( line (start 50 2 ltcorner) (end 50 0 ltcorner) (repeat 30) (incrx 50) )
This is a small vertical line (2mm height) at 50 mm of the left top corner, repeated 30 times, on the X axis, every
50mm.
In fact, only segments inside the page will be taken in account.

5.14.3.3 - Text item
A text description starts by
( tbtext “text to display” (<position>)
and can have attributes:
- font (bold, italic font size)
- rotation
- repeat parameters

Here is a simple text at 109 mm to the left of the right bottom corner and 14.3mm above it:
( tbtext "File: %F" (pos 109 14.3) )
This text has a format parameter (%F), which will be replaced by the filename of the sheet at
run time, and uses the default size and the default text line width.
It is left justified on the X axis, and center justified on the Y axis (default values)

Here is more complex texts
( tbtext "Title: %T" (pos 109 10.7)(font bold (size 2 2)) )
Uses a bold font, with a X size = 2mm, and a Y size = 2mm
( tbtext "1" (pos 25 1 ltcorner) (font (size 1.3 1.3))(repeat 100) (incrx 50) )
is a text repeated 100 times (in fact only if items are inside the page)
( tbtext "A" (pos 1 25 ltcorner) (font (size 1.3 1.3)) (justify center)(repeat 100) (incry 50) )
Repeated texts have 4 parameters to control duplicate items
repeat defines the repeat count
incry defines the step on X axis (default = 0)
incriy defines the step on Y axis (default = 0)
inclabel define a text modification (default = 1).
his value, if not 0 have meaning only for texts having only one letter. 
If this letter is a digit, if will be converted to a number, and each duplicated item will also be a number incremented by inclabel
If this letter is not a digit, each duplicated text will be the letter corresponding to the ascii code incremented by inclabel
( tbtext "1" creates duplicate texts "1", "2", … "9", "10", "11" …
( tbtext "A" creates duplicate texts "A", "B", …"Z" …

5.14.3.4 - Poly polygon item
The poly polygon description starts by (polygon (<position>)
(polygon (pos 134 18 rbcorner)
and can include a rotation, repeat parameters, and linewidth definition.
(rotate 20) (linewidth 0.00254)
Note: the linewidth is used to draw the outlines of the filled polygons.
If not defined, the value 0 is used.
The next info is the list of corners corresponding to filled polygons to draw:
defines the 3 first corners of a new polygon.
A corner is described by (xy <X pos> <Y pos>), with <X pos> and <Y pos> are coordinates relative to the poly polygon position.

Here is an example of a poly polygon:
(polygon (pos 134 18 rbcorner) (rotate 20) (linewidth 0.00254)
)

There are 2 polygons (2 blocks starting by "(pts")
The set of polygons is rotated by 20 degrees (CCW)

5.14.4 - Example: