

**PCB &#x7f16;&#x8f91;&#x5668;**

The KiCad Team

REVISION HISTORY			
NUMBER	DATE	DESCRIPTION	NAME

# Contents

<b>1</b>	<b>Pcbnew &amp;#x7b80;&amp;#x4ecb;</b>	<b>1</b>
1.1	&#x521d;&#x59cb;&#x914d;&#x7f6e; . . . . .	1
1.2	Pcbnew &#x7528;&#x6237;&#x754c;&#x9762; . . . . .	2
1.3	&#x5bfc;&#x822a;&#x7f16;&#x8f91;&#x753b;&#x5e03; . . . . .	2
1.4	&#x5feb;&#x6377;&#x952e; . . . . .	3
<b>2</b>	<b>&amp;#x663e;&amp;#x793a;&amp;#x548c;&amp;#x9009;&amp;#x62e9;&amp;#x63a7;&amp;#x4ef6;</b>	<b>4</b>
2.1	&#x677f;&#x5c42; . . . . .	4
2.1.1	&#x7535;&#x8def;&#x677f;&#x5c42;&#x7684;&#x663e;&#x793a;&#x987a;&#x5e8f; . . . . .	4
2.2	&#x5916;&#x89c2;&#x9762;&#x677f; . . . . .	4
2.2.1	&#x56fe;&#x5c42;&#x63a7;&#x4ef6; . . . . .	4
2.2.2	&#x5bf9;&#x8c61;&#x63a7;&#x4ef6; . . . . .	5
2.2.3	&#x56fe;&#x5c42;&#x9884;&#x8bbe; . . . . .	5
2.2.4	&#x7f51;&#x7edc;&#x548c;&#x7f51;&#x7edc;&#x7c7b;&#x63a7;&#x4ef6; . . . . .	5
2.3	&#x9009;&#x62e9;&#x548c;&#x9009;&#x62e9;&#x7b5b;&#x9009;&#x5668; . . . . .	6
2.4	&#x7f51;&#x7edc;&#x9ad8;&#x4eae; . . . . .	6
2.5	&#x4ece;&#x539f;&#x7406;&#x56fe;&#x4ea4;&#x53c9;&#x63a2;&#x6d4b; . . . . .	6
2.6	&#x5de6;&#x4fa7;&#x5de5;&#x5177;&#x680f;&#x663e;&#x793a;&#x63a7;&#x4ef6; . . . . .	7
<b>3</b>	<b>&amp;#x521b;&amp;#x5efa; PCB</b>	<b>9</b>
3.1	&#x57fa;&#x672c; PCB &#x6982;&#x5ff5; . . . . .	9
3.2	&#x6027;&#x80fd; . . . . .	9
3.3	&#x4ece;&#x539f;&#x7406;&#x56fe;&#x5f00;&#x59cb; . . . . .	9
3.3.1	&#x9009;&#x9879; . . . . .	10
3.4	&#x4ece;&#x5934;&#x5f00;&#x59cb; . . . . .	11
3.5	&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e; . . . . .	11
3.5.1	&#x914d;&#x7f6e;&#x7535;&#x8def;&#x677f;&#x538b;&#x5c42;&#x548c;&#x7269;&#x7406;&#x53c2;&#x6579; . . . . .	12
3.5.2	&#x914d;&#x7f6e;&#x9ed8;&#x8ba4;&#x6587;&#x672c;&#x548c;&#x56fe;&#x5f62;&#x8bbe;&#x7f6e; . . . . .	12
3.5.3	&#x914d;&#x7f6e;&#x8bbe;&#x8ba1;&#x89c4;&#x5219; . . . . .	13
3.5.3.1	&#x7ea6;&#x675f; . . . . .	13
3.5.3.2	&#x9884;&#x5b9a;&#x4e49;&#x5927;&#x5c0f; . . . . .	14

3.5.3.3	151; 7edc; 7c7b; . . . . .	14
3.5.3.4	81ea; 5b9a; 4e49; 89c4; 5219; . . . . .	15
3.5.3.5	8fdd; 89c4; 4e25; 91cd; 7a0b; 5ea6; . . . . .	16
3.5.3.6	6b63; 5728; 5bfc; 5165; 8bbe; 7f6e; . . . . .	16
<b>4</b>	<b>16; 91; 7535; 8def; 677f;</b>	<b>18</b>
4.1	653e; 7f6e; 548c; 7ed8; 5236; 64cd; 4f5c; . . . . .	18
4.2	6355; 6349; . . . . .	19
4.3	7f16; 8f91; 5bf9; 8c61; 5c5e; 6027; . . . . .	19
4.4	4f7f; 7528; 5c01; 88c5; . . . . .	20
4.5	4f7f; 7528; 710a; 76d8; . . . . .	20
4.6	4f7f; 7528; 533a; 57df; . . . . .	22
4.7	56fe; 5f62; 5bf9; 8c61; . . . . .	24
4.7.1	6b63; 5728; 521b; 5efa; 56fe; 5f62; 5f62; 72b6; . . . . .	24
4.7.2	6b63; 5728; 521b; 5efa; 6587; 672c; 5bf9; 8c61; . . . . .	25
4.7.3	7535; 8def; 677f; 8f6e; 5ed3; ( 8fb9; 7f18; 5207; 5272;) . . . . .	26
4.8	6807; 6ce8; . . . . .	26
4.8.1	6807; 6ce8; 683c; 5f0f; 9009; 9879; . . . . .	28
4.8.2	6807; 6ce8; 6587; 672c; 9009; 9879; . . . . .	28
4.8.3	6807; 6ce8; 7ebf; 9009; 9879; . . . . .	29
4.8.4	5f15; 7ebf; 9009; 9879; . . . . .	29
4.9	5e03; 7ebf; . . . . .	29
4.9.1	5e03; 7ebf; 5f62; 6001; . . . . .	30
4.9.2	5e03; 7ebf; 8f6c; 89d2; 6a21; 5f0f; . . . . .	31
4.9.3	5e03; 7ebf; 5bbd; 5ea6; . . . . .	31
4.9.4	653e; 7f6e; 8fc7; 5b54; . . . . .	32
4.9.5	5dee; 5206; 5bf9; 5e03; 7ebf; . . . . .	32
4.9.6	4fee; 6539; 5e03; 7ebf; . . . . .	33
4.9.7	957f; 5ea6; 8c03; 6574; . . . . .	33
4.9.8	4ea4; 4e92; 5f0f; 5e03; 7ebf; 8bbe; 7f6e; . . . . .	34
4.10	5411; 524d; 548c; 5411; 540e; 6279; 6ce8; . . . . .	36
4.10.1	4f4d; 7f6e; 91cd; 65b0; 6279; 6ce8; . . . . .	36
4.11	9501; 5b9a; . . . . .	36
4.12	6279; 91cf; 7f16; 8f91; 5de5; 5177; . . . . .	37
4.13	6e05; 7406; 5de5; 5177; . . . . .	37
4.14	6b63; 5728; 5bfc; 5165; 56fe; 5f62; . . . . .	37
4.14.1	4ece; DXF 548c; SVG 6587; 4ef6; 5bfc; 5165; 77e2; 91cf; 56fe; . . . . .	37
4.14.2	6b63; 5728; 5bfc; 5165; 4f4d; 56fe; 56fe; 50cf; . . . . .	37

<b>5</b>	<b>&amp;#x68c0;&amp;#x67e5;&amp;#x7535;&amp;#x8def;&amp;#x677f;</b>	<b>38</b>
5.1	&#x6d4b;&#x91cf;&#x5de5;&#x5177;	38
5.2	&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x68c0;&#x67e5;	38
5.2.1	&#x95f4;&#x9699;&#x548c;&#x7ea6;&#x675f;&#x89e3;&#x6790;	40
5.3	Find tool	42
5.4	3D &#x67e5;&#x770b;&#x5668;	43
5.4.1	Navigating the 3D view	44
5.4.2	Generating images with the 3D Viewer	44
5.4.3	3D viewer controls	44
5.5	&#x7f51;&#x7edc;&#x68c0;&#x67e5;	45
<b>6</b>	<b>&amp;#x751f;&amp;#x6210;&amp;#x8f93;&amp;#x51fa;</b>	<b>47</b>
6.1	&#x5236;&#x9020;&#x8f93;&#x51fa;&#x548c;&#x7ed8;&#x5236;	47
6.1.1	&#x7ed8;&#x5236;&#x9009;&#x9879;	48
6.1.2	Gerber &#x9009;&#x9879;	49
6.1.3	Postscript &#x9009;&#x9879;	49
6.1.4	SVG &#x9009;&#x9879;	50
6.1.5	DXF &#x9009;&#x9879;	50
6.1.6	HPGL &#x9009;&#x9879;	50
6.2	&#x94bb;&#x5b54;&#x6587;&#x4ef6;	50
6.3	&#x5143;&#x4ef6;&#x653e;&#x7f6e;&#x6587;&#x4ef6;	51
6.4	&#x989d;&#x5916;&#x7684;&#x5236;&#x9020;&#x4ea7;&#x51fa;	52
6.5	&#x6253;&#x5370;	53
6.6	&#x6b63;&#x5728;&#x5bfc;&#x51fa;&#x6587;&#x4ef6;	53
6.6.1	IDF Exporter	54
<b>7</b>	<b>&amp;#x5c01;&amp;#x88c5;&amp;#x548c;&amp;#x5c01;&amp;#x88c5;&amp;#x5e93;</b>	<b>56</b>
7.1	&#x7ba1;&#x7406;&#x5c01;&#x88c5;&#x5e93;	56
7.1.1	Initial Configuration	57
7.1.2	Managing Table Entries	57
7.1.3	Environment Variable Substitution	58
7.1.4	Using the GitHub Plugin	58
7.2	&#x521b;&#x5efa;&#x548c;&#x7f16;&#x8f91;&#x5c01;&#x88c5;	58
7.2.1	&#x81ea;&#x5b9a;&#x4e49;&#x710a;&#x76d8;&#x5f62;&#x72b6;	58
7.2.2	&#x5c01;&#x88c5;&#x5c5e;&#x6027;	58
7.2.3	&#x5c01;&#x88c5;&#x5411;&#x5bfc;	59

<b>8</b>	<b>&amp;#x9ad8;&amp;#x7ea7;&amp;#x4e3b;&amp;#x9898;</b>	<b>60</b>
8.1	&#x914d;&#x7f6e;&#x548c;&#x81ea;&#x5b9a;&#x4e49;	60
8.1.1	&#x663e;&#x793a;&#x9009;&#x9879;	60
8.1.2	&#x7f16;&#x8f91;&#x9009;&#x9879;	61
8.1.3	&#x989c;&#x8272;	62
8.1.4	&#x64cd;&#x4f5c;&#x63d2;&#x4ef6;	63
8.1.5	&#x539f;&#x70b9;&#x548c;&#x8f74;	63
8.2	&#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;	64
8.2.1	&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x7f16;&#x8f91;&#x5668;	64
8.2.2	&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x8bed;&#x6cd5;	64
8.2.2.1	&#x56fe;&#x5c42;&#x5b50;&#x53e5;	65
8.2.2.2	&#x6761;&#x4ef6;&#x5b50;&#x53e5;	65
8.2.2.3	&#x7ea6;&#x675f;	66
8.2.3	&#x5bf9;&#x8c61;&#x5c5e;&#x6027;&#x548c;&#x51fd;&#x6570;&#x53c2;&#x8003;	68
8.2.3.1	&#x5e38;&#x89c1;&#x5c5e;&#x6027;	68
8.2.3.2	&#x8fde;&#x63a5;&#x7684;&#x5bf9;&#x8c61;&#x5c5e;&#x6027;	69
8.2.3.3	&#x5c01;&#x88c5;&#x5c5e;&#x6027;	69
8.2.3.4	&#x710a;&#x76d8;&#x5c5e;&#x6027;	69
8.2.3.5	&#x5e03;&#x7ebf;&#x548c;&#x5706;&#x5f27;&#x5c5e;&#x6027;	70
8.2.3.6	&#x8fc7;&#x5b54;&#x5c5e;&#x6027;	70
8.2.3.7	&#x8986;&#x94dc;&#x548c;&#x89c4;&#x5219;&#x533a;&#x57df;&#x5c5e;&#x6027;	70
8.2.3.8	&#x56fe;&#x5f62;&#x5f62;&#x72b6;&#x5c5e;&#x6027;	71
8.2.3.9	&#x6587;&#x672c;&#x5c5e;&#x6027;	71
8.2.3.10	&#x8868;&#x8fbe;&#x5f0f;&#x51fd;&#x6570;	71
8.2.4	&#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x793a;&#x4f8b;	72
8.3	&#x811a;&#x672c;	73
8.3.1	Python &#x811a;&#x672c;&#x4f4d;&#x7f6e;	73
8.3.2	&#x64cd;&#x4f5c;&#x63d2;&#x4ef6;	73
8.3.3	&#x5c01;&#x88c5;&#x5411;&#x5bfc;	73
8.3.4	&#x4f7f;&#x7528;&#x811a;&#x672c;&#x63a7;&#x5236;&#x53f0;	74
8.3.5	&#x7f16;&#x5199;&#x5916;&#x90e8;&#x811a;&#x672c;	74
8.3.6	&#x7f16;&#x5199;&#x64cd;&#x4f5c;&#x63d2;&#x4ef6;	74
8.4	Working With IDF Component Outlines	74
8.4.1	Specifying component models for use by the exporter	74
8.4.2	Creating a component outline file	76
8.4.3	Guidelines for creating outlines	77
8.4.3.1	Package naming	77
8.4.3.2	Comments	77
8.4.3.3	Geometry and Part Number entries	77

8.4.3.4	Pin orientation and positioning . . . . .	77
8.4.3.5	Tips on dimensions . . . . .	78
8.4.4	IDF Component Outline Tools . . . . .	78
8.4.4.1	idfcyl . . . . .	78
8.4.4.2	idfrect . . . . .	79
8.4.4.3	dx2idf . . . . .	80
8.4.4.4	idf2vrm . . . . .	81
9	&#x64cd;&#x4f5c;&#x53c2;&#x8003;	82
9.1	PCB &#x7f16;&#x8f91;&#x5668; . . . . .	82
9.2	3D &#x67e5;&#x770b;&#x5668; . . . . .	87
9.3	Common . . . . .	89

53c2; 8003; 624b; 518c;

---

#### Note

This manual is in the process of being revised to cover the latest stable release version of KiCad. It contains some sections that have not yet been completed. We ask for your patience while our volunteer technical writers work on this task, and we welcome new contributors who would like to help make KiCad's documentation better than ever.

---

#### 7248; 6743;

This document is Copyright © 2010-2022 by its contributors as listed below. You may distribute it and/or modify it under the terms of either the GNU General Public License (<http://www.gnu.org/licenses/gpl.html>), version 3 or later, or the Creative Commons Attribution License (<http://creativecommons.org/licenses/by/3.0/>), version 3.0 or later.

672c; 6307; 5357; 4e2d; 7684; 6240; 6709; 5546; 6807; 5747; 5c5e; 4e8e; 5176; 8d21; 732e; 4eba; 5458;

#### 8d21; 732e; 4eba; 5458;

Jean-Pierre Charras, Fabrizio Tappero, Wayne Stambaugh, Cirilo Bernardo, Jon Evans, Graham Keeth

#### 7ffb; 8bd1; 4eba; 5458;

taotieren <[admin@taotieren.com](mailto:admin@taotieren.com)>, 2019, 2020, 2021.

Telegram 7b80; 4f53; 4e2d; 6587; 4ea4; 6d41; 7fa4;: [https://t.me/KiCad\\_zh\\_CN](https://t.me/KiCad_zh_CN)

8bd1; 8005; 6ce8; fff1a; 82f1; 6587; 53cc; 5f15; 53f7; 5305; 542b; 7684; 4e2d; 6587; 53cd; 9988;

#### 53cd; 9988;

The KiCad project welcomes feedback, bug reports, and suggestions related to the software or its documentation. For more information on how to submit feedback or report an issue, please see the instructions at <https://www.kicad.org/help/report-an-issue/>

---

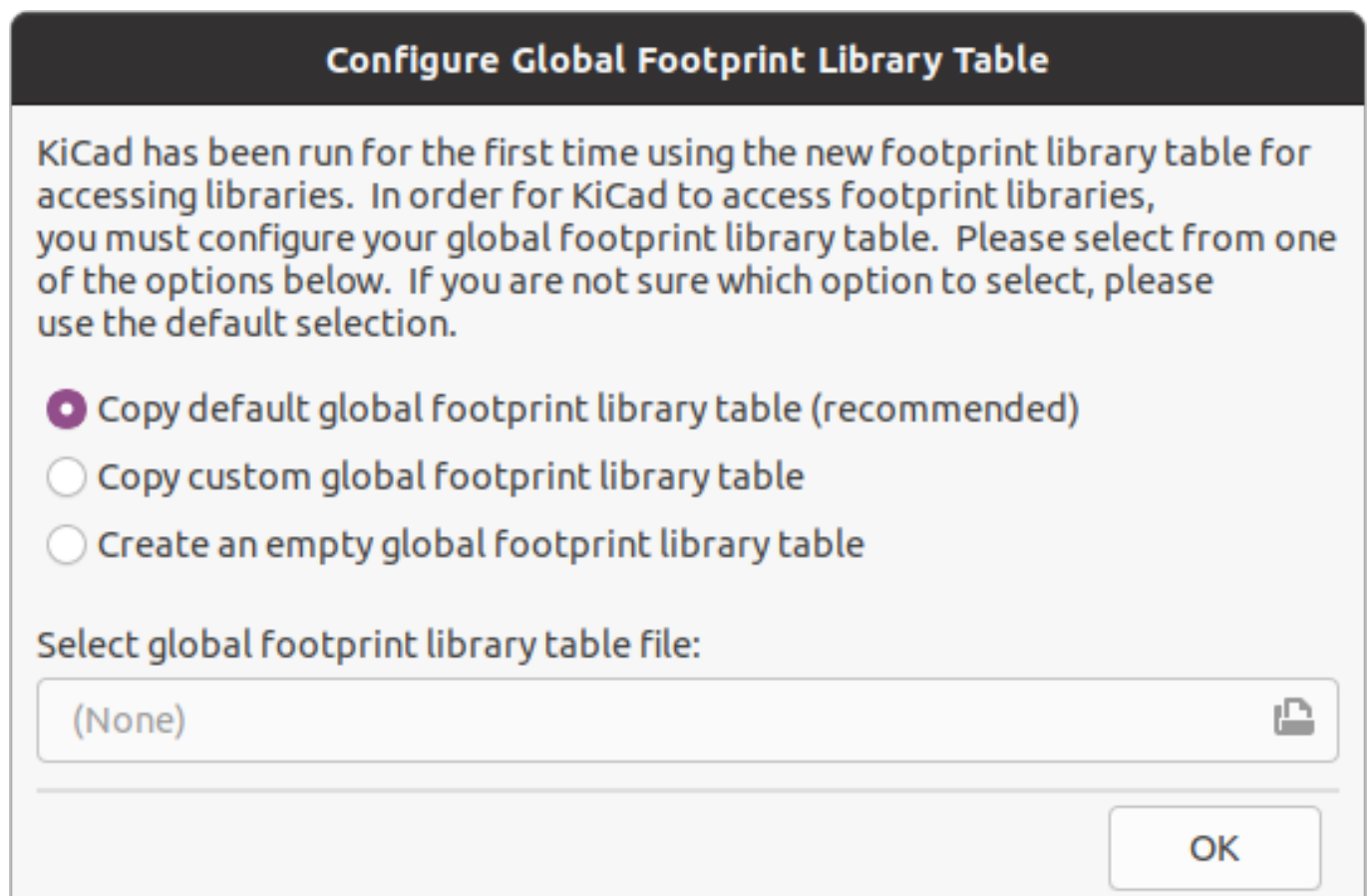


## Chapter 1

# Pcbnew

### 1.1

Pcbnew; KiCad footprint library table; Pcbnew



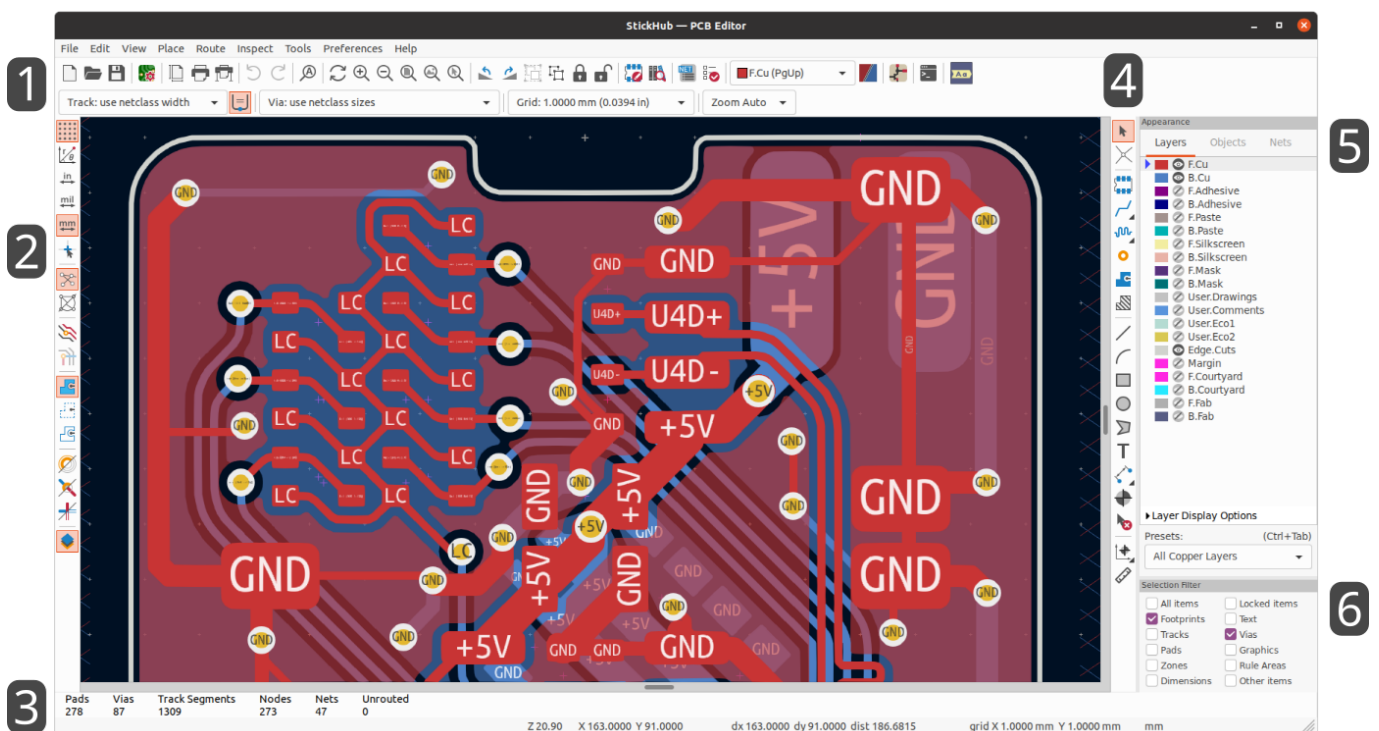
The first option is recommended (**Copy default global footprint library table (recommended)**). The default footprint library table includes all of the standard footprint libraries that are installed as part of KiCad.

If this option is disabled, KiCad was unable to find the default global footprint library table. This probably means you did not install the standard footprint libraries with KiCad, or they are not installed where KiCad expects to find them. On some systems the KiCad libraries are installed as a separate package.

- If you have installed the standard KiCad footprint libraries and want to use them, but the first option is disabled, select the second option and browse to the `fp-lib-table` file in the directory where the KiCad libraries were installed.
- If you already have a custom footprint library table that you would like to use, select the second option and browse to your `fp-lib-table` file.
- If you want to construct a new footprint library table from scratch, select the third option.

Footprint library management is described in more detail [later](#).

## 1.2 Pcbnew



1. [Pcbnew](#)






1. [Pcbnew](#)
2. [Pcbnew](#)
3. [Pcbnew](#)
4. [Pcbnew](#)
5. [Pcbnew](#)
6. [Pcbnew](#)

## 1.3

[Pcbnew](#)

By default, dragging with the middle or right mouse button will pan the canvas view and scrolling the mouse wheel will zoom the view in or out. You can change this behavior in the Mouse and Touchpad section of the preferences (see [Configuration and Customization](#) for details).

Several other zoom tools are available in the top toolbar:

-  zooms in on the center of the viewport.
-  zooms out from the center of the viewport.
-  zooms to fit the frame around the drawing sheet.
-  zooms to fit the items within the drawing sheet.
-  allows you to draw a box to determine the zoomed area.

The cursor's current position is displayed at the bottom of the window (X and Y), along with the current zoom factor (Z), the cursor's relative position (dx, dy, and dist), the grid setting, and the display units.

The relative coordinates can be reset to zero by pressing `kbd:[Space]`. This is useful for measuring distance between two points or aligning objects.

## 1.4 &#x5feb;&#x6377;&#x952e;

The `kbd:[Ctrl+F1]` shortcut displays the current hotkey list. The default hotkey list is included in the [Actions Reference](#) section of the manual.

The hotkeys described in this manual use the key labels that appear on a standard PC keyboard. On an Apple keyboard layout, use the `kbd:[Cmd]` key in place of `kbd:[Ctrl]`, and the `kbd:[Option]` key in place of `kbd:[Alt]`.

Many actions do not have hotkeys assigned by default, but hotkeys can be assigned or redefined using the hotkey editor (**Preferences** → **Preferences...** → **Hotkeys**).

---

### Note

&#x901a;&#x8fc7;&#x70ed;&#x952e;&#x53ef;&#x7528;&#x7684;&#x8bb8;&#x591a;&#x64cd;&#x4f5c;&#x4e5f;&#x53ef;&#x5728;&#x

---

Hotkeys are stored in the file `user.hotkeys` in KiCad's configuration directory. The location is platform-specific:

- Windows: `%APPDATA%\kicad\6.0\user.hotkeys`
- Linux: `~/ .config/kicad/6.0/user.hotkeys`
- macOS: `~/Library/Preferences/kicad/6.0/user.hotkeys`

KiCad can import hotkey settings from a `user.hotkeys` file using the **Import Hotkeys** button in the hotkey editor.

---

## Chapter 2

# &#x663e;&#x793a;&#x548c;&#x9009;&#x62e9;&#x6

### 2.1 &#x677f;&#x5c42;

Pcbnew &#x4e2d;&#x7684;&#x5c42;&#x8868;&#x793a;&#x7ebf;&#x8def;&#x677f;&#x4e0a;&#x7684;&#x7269;&#x7406;&#x94  
&#x5728;&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x59cb;&#x7ec8;&#x6709;&#x4e00;&#x4e2a;&#x5904;&#x4e8e;&#x6d3b;&#x5  
&#x6d3b;&#x52a8;&#x56fe;&#x5c42;&#x88ab;&#x753b;&#x5728;&#x5176;&#x4ed6;&#x5c42;&#x4e4b;&#x4e0a;&#xff0c;&#x5  
&#x6d3b;&#x52a8;&#x56fe;&#x5c42;&#x5728;&#x9876;&#x90e8;&#x5de5;&#x5177;&#x680f;&#x7684;&#x5c42;&#x9009;&#x6  
&#x82e5;&#x8981;&#x53d8;&#x66f4;&#x6d3b;&#x52a8;&#x56fe;&#x5c42;&#xff0c;&#x53ef;&#x4ee5;&#x5de6;&#x952e;&#x53  
&#x53ef;&#x4ee5;&#x9690;&#x85cf;&#x56fe;&#x5c42;&#x4ee5;&#x7b80;&#x5316;&#x7535;&#x8def;&#x677f;&#x89c6;&#x56

#### 2.1.1 &#x7535;&#x8def;&#x677f;&#x5c42;&#x7684;&#x663e;&#x793a;&#x987a;&#x5e8f;

---

**Note**

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;&#x3002;

---

### 2.2 &#x5916;&#x89c2;&#x9762;&#x677f;

&#x5916;&#x89c2;&#x9762;&#x677f;&#x63d0;&#x4f9b;&#x7528;&#x4e8e;&#x7ba1;&#x7406; Pcbnew &#x7ed8;&#x56fe;&#x75

#### 2.2.1 &#x56fe;&#x5c42;&#x63a7;&#x4ef6;

&#x5728;&#x5916;&#x89c2;&#x9762;&#x677f;&#x7684;&#x56fe;&#x5c42;&#x9009;&#x9879;&#x5361;&#x4e2d;&#xff0c;&#x6  
&#x6d3b;&#x52a8;&#x56fe;&#x5c42;&#x5728;&#x8272;&#x5757;&#x7684;&#x5de6;&#x8fb9;&#x6709;&#x4e00;&#x4e2a;&#x7  
&#x5de6;&#x952e;&#x70b9;&#x51fb;&#x4e00;&#x4e2a;&#x56fe;&#x5c42;&#x6765;&#x9009;&#x62e9;&#x5b83;&#x4f5c;&#x4  
&#x5de6;&#x952e;&#x5355;&#x51fb;&#x76f8;&#x5e94;&#x7684;&#x53ef;&#x89c1;&#x6027;&#x56fe;&#x6807;&#xff0c;&#x57  
&#x53cc;&#x51fb;&#x6216;&#x4e2d;&#x51fb;&#x8272;&#x5757;&#x6765;&#x6539;&#x53d8;&#x8be5;&#x56fe;&#x5c42;&#x7

---

**Note**

&#x5fc5;&#x987b;&#x5148;&#x5728;&#x9996;&#x9009;&#x53c2;&#x6570;&#x4e2d;&#x521b;&#x5efa;&#x81ea;&#x5b9a;&#x4e49;&#x  
&#x7136;&#x540e;&#x624d;&#x80fd;&#x5728;&#x5916;&#x89c2;&#x9762;&#x677f;&#x4e2d;&#x66f4;&#x6539;&#x56fe;&#x5c42;&#x7

---

&#x5728;&#x56fe;&#x5c42;&#x5217;&#x8868;&#x4e0b;&#x65b9;&#x662f;&#x4e00;&#x4e2a;&#x5305;&#x542b;&#x56fe;&#x5  
"&#x6697;&#x663e;" &#x6216; "&#x9690;&#x85cf;" &#x65f6;&#xff0c;&#x4e0d;&#x80fd;&#x9009;&#x62e9;&#x975e;&#x6d3b;&#x6  
kbd:[Ctrl+H] &#x5feb;&#x901f;&#x5207;&#x6362;&#x8fd9;&#x4e9b;&#x663e;&#x793a;&#x6a21;&#x5f0f;&#x3002;

---

&#x7ffb;&#x8f6c;&#x7535;&#x8def;&#x677f;&#x89c6;&#x56fe; &#xc06;&#x663e;&#x793a;&#x7535;&#x8def;&#x677f;&#xf  
Y &#x8f74;&#x955c;&#x50cf;)&#x3002;&#x6b64;&#x9009;&#x9879;&#x4e5f;&#x53ef;&#x5728;&#x89c6;&#x56fe;&#x83dc;&#x

### Note

&#x7fb;&#x86c;&#x7535;&#x8def;&#x677f;&#x89c6;&#x56fe;&#x4e0d;&#x4f1a;&#x66f4;&#x6539;&#x53ef;&#x89c6;&#x5c42;&#x98

### 2.2.2 **2.2.2**

&#x5916;&#x89c2;&#x9762;&#x677f;&#x7684;"&#x5bf9;&#x8c61;"&#x9009;&#x9879;&#x5361;&#x4e0e;"&#x56fe;&#x5c42;"&#x9009;&#x9879;&#x5361;&#x7c7b;&#x4f3c;&#x3002;&#x4e3b;&#x8981;&#x533a;&#x522b;&#x5728;&#x4e8e;&#xff0c;&#x6876;&#x8fd9;&#x91cc;&#x7684;&#x4e0d;&#x900f;&#x660e;&#x5ea6;&#x8bbe;&#x7f6e;&#x5c06;&#x4e0e;&#x56fe;&#x5c42;&#x9876;&#x9ed8;&#x8ba4;&#x60c5;&#x51b5;&#x4e0b;&#xff0c;&#x6240;&#x6709;&#x5bf9;&#x8c61;&#x90fd;&#x662f;&#x5b8c;&#x51

### 2.2.3 $\mathbb{R}^2$ and $\mathbb{R}^3$

&#x56fe;&#x5c42;&#x9884;&#x8bbe;&#x5b58;&#x50a8;&#x4e86;&#x54ea;&#x4e9b;&#x56fe;&#x5c42;&#x548c;&#x5bf9;&#x884e;&#x6709;&#x51e0;&#x4e2a;&#x5185;&#x7f6e;&#x7684;&#x56fe;&#x5c42;&#x9884;&#x8bbe;&#xff0c;&#x60a8;&#x53ef;&#x4e86;&#x81ea;&#x5b9a;&#x4e49;&#x9884;&#x8bbe;&#x5b58;&#x50a8;&#x5728;&#x4e00;&#x4e2a;&#x7535;&#x8def;&#x677f;&#x7684;&#x8981;&#x52a0;&#x8f7d;&#x4e00;&#x4e2a;&#x9884;&#x8bbe;&#xff0c;&#x8bf7;&#x4ece;&#x5916;&#x89c2;&#x9762;&#x677d;&#x5e76;&#x6309; kbd:[Tab] &#x6765;&#x4f7f;&#x7528;&#x5feb;&#x901f;&#x5207;&#x6362;&#x5668;&#x3002;&#x4e00;&#x65e6;&#x5feb;&#x901f;&#x5207;&#x6362;&#x5668;&#x7a97;&#x53e3;&#x51fa;&#x73b0;&#xff0c;&#x4f60;&#x53ef;&#x4e86;&#x548c; kbd:[Shift+Tab] &#x6765;&#x5faa;&#x73af;&#x6d4f;&#x89c8;&#x53ef;&#x7528;&#x7684;&#x9884;&#x8bbe;&#x5f53;&#x4f60;&#x653e;&#x5f00; kbd:[Ctrl] &#x952e;&#x65f6;&#xff0c;&#x9ad8;&#x4eae;&#x663e;&#x793a;&#x7684;&#x9884;&#x8981;&#x4fdd;&#x5b58;&#x4e00;&#x4e2a;&#x81ea;&#x5b9a;&#x4e49;&#x7684;&#x9884;&#x8bbe;&#xff0c;&#x9996;&#x57ed9;&#x60a8;&#x7684;&#x9884;&#x8bbe;&#x4e00;&#x4e2a;&#x540d;&#x5b57;&#xff0c;&#x5b83;&#x73b0;&#x5728;&#x5f53;&#x8981;&#x4fee;&#x6539;&#x4e00;&#x4e2a;&#x81ea;&#x5b9a;&#x4e49;&#x9884;&#x8bbe;&#xff0c;&#x8bf7;&#x9075;&#x5f53;&#x8981;&#x5220;&#x9664;&#x4e00;&#x4e2a;&#x81ea;&#x5b9a;&#x4e49;&#x9884;&#x8bbe;&#xff0c;&#x4ece;&#x4e0b;&#x677d;

#### 2.2.4 **2.2.4**

&#x5916;&#x89c2;&#x9762;&#x677f;&#x7684;&#x7f51;&#x7edc;&#x9009;&#x9879;&#x5361;&#x663e;&#x793a;&#x7535;&#xa  
&#x6bcf;&#x4e2a;&#x7f51;&#x7edc;&#x548c;&#x7f51;&#x7edc;&#x7c7b;&#x8fd8;&#x53ef;&#x4ee5;&#x6307;&#x5b9a;&#x4e  
(&#x6216;&#x7f51;&#x7edc;&#x7c7b;&#x4e2d;&#x7684;&#x6240;&#x6709;&#x7f51;&#x7edc;) &#x7684;&#x98de;&#x7ebf;&#x

### Note

&#x9ed8;&#x8ba4;&#x7f51;&#x7edc;&#x7c7b;&#x4e0d;&#x80fd;&#x5206;&#x914d;&#x989c;&#x8272;&#xff0c;  
&#x56e0;&#x4e3a;&#x8be5;&#x7c7b;&#x4e2d;&#x7684;&#x7f51;&#x7edc;&#x5c06;&#x4ec5;&#x4f7f;&#x7528;&#x989c;&#x8272;&#

&#x60a8;&#x8fd8;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x5916;&#x89c2;&#x9762;&#x677f;&#x9009;&#x62e9;&#x5e76;&#x9  
&#x7f51;&#x7edc;&#x7c7b;&#x5217;&#x8868;&#x4e0b;&#x9762;&#x662f;&#x4e00;&#x4e2a;&#x5305;&#x542b;&#x7f51;&#x7  
(&#x710a;&#x76d8;&#x3001;&#x5e03;&#x7ebf;&#x3001;&#x8fc7;&#x5b54;&#x548c;&#x533a;&#x57df.)&#x90fd;&#x5c06;&#x  
&#x5f53;&#x9009;&#x62e9;“&#x6ca1;&#x6709;”&#x65f6;&#xff0c;&#x7f51;&#x7edc;&#x548c;&#x7f51;&#x7edc;&#x7c7b;&#x9  
&#x7b2c;&#x4e8c;&#x4e2a;&#x9009;&#x9879;&#x63a7;&#x5236;&#x5982;&#x4f55;&#x7ed8;&#x5236;&#x98de;&#x7ebf;&#x3

## 2.3 2.3.2.3 2.3.2.3.2.3 2.3.2.3.2.3.2.3

Selecting items in the editing canvas is done with the left mouse button. Single-clicking on an object will select it and dragging will perform a box selection. A box selection from left to right will only select items that are fully inside the box. A box selection from right to left will select any items that touch the box. A left-to-right selection box is drawn in yellow, with a cursor that indicates exclusive selection, and a right-to-left selection box is drawn in blue with a cursor that indicates inclusive selection.

&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x5728;&#x5355;&#x51fb;&#x6216;&#x62d6;&#x52a8;&#x7684;&#x540c;&#x65f6;&#x6

Modifier Keys (Windows)	Modifier Keys (Linux)	Modifier Keys (macOS)	Selection Effect
kbd:[Shift]	kbd:[Shift]	kbd:[Shift]	Add the item to the existing selection.
kbd:[Ctrl + Shift]	kbd:[Ctrl + Shift]	kbd:[Cmd + Shift]	Remove the item from the existing selection.
long click	long click or kbd:[Alt]	long click or kbd:[Option]	Clarify selection from a pop-up menu.
kbd:[Ctrl]	kbd:[Ctrl]	kbd:[Cmd]	Highlight the net of the selected copper item.

&#x62d6;&#x52a8;&#x4ee5;&#x6267;&#x884c;&#x9009;&#x6846;&#x65f6;&#xff0c;&#x5c06;&#x5e94;&#x7528;&#x4ee5;&#x4

Modifier Keys (Windows)	Modifier Keys (Linux)	Modifier Keys (macOS)	Selection Effect
kbd:[Shift] or kbd:[Ctrl]	kbd:[Shift] or kbd:[Ctrl]	kbd:[Shift] or kbd:[Ctrl]	Add item(s) to the existing selection.
kbd:[Ctrl + Shift]	kbd:[Ctrl + Shift]	kbd:[Cmd + Shift]	Remove item(s) from the existing selection.

Pcbnew &#x7a97;&#x53e3;&#x53f3;&#x4e0b;&#x89d2;&#x7684;&#x9009;&#x62e9;&#x8fc7;&#x6ee4;&#x5668;&#x9762;&#x67  
&#x5f53;&#x4e00;&#x4e2a;&#x8fde;&#x63a5;&#x7684;&#x94dc;&#x7ebf;&#x9879;&#x76ee;&#x88ab;&#x9009;&#x4e2d;&#x63  
"&#x6269;&#x5c55;&#x9009;&#x62e9;" &#x547d;&#x4ee4;&#x6216;&#x70ed;&#x952e; kbd:[U] &#x5c06;&#x9009;&#x62e9;&#  
&#x7b2c;&#x4e00;&#x6b21;&#x8fd0;&#x884c;&#x8fd9;&#x4e2a;&#x547d;&#x4ee4;&#x65f6;&#xff0c;&#x9009;&#x62e9;&#x5c  
&#x7b2c;&#x4e8c;&#x6b21;&#xff0c;&#x9009;&#x62e9;&#x5c06;&#x88ab;&#x6269;&#x5c55;&#x5230;&#x6240;&#x6709;&#x5

Selecting an object displays information about the object in the message panel at the bottom of the window. Double-clicking an object opens a window to edit the object's properties.

&#x6309; kbd:[Esc] &#x5c06;&#x59cb;&#x7ec8;&#x53d6;&#x6d88;&#x5f53;&#x524d;&#x5de5;&#x5177;&#x6216;&#x64cd;&#x524d; kbd:[Esc] &#x5c06;&#x6e05;&#x9664;&#x5f53;&#x524d;&#x9009;&#x62e9;&#x3002;

## 2.4 $\mathbb{R}^n$ and $\mathbb{C}^n$

```
&#x7535;&#x6c14;&#x7f51;&#x7edc;&#xff08;&#x6216;&#x4e00;&#x7ec4;&#x7f51;&#x7edc;&#xff09;&#x53ef;&#x4ee5;&#x572
PCB&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x88ab;&#x9ad8;&#x4eae;&#x663e;&#x793a;&#xff0c;&#x4ee5;&#x663e;&#x793a;&
PCB&#x4e0a;&#x5e03;&#x7ebf;&#x7684;&#x3002;&#x901a;&#x8fc7;&#x5728; PCB&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x
&#x5f53;&#x7f51;&#x7edc;&#x9ad8;&#x4eae;&#x6fc0;&#x6d3b;&#x65f6;&#xff0c;&#x9ad8;&#x4eae;&#x7684;&#x7f51;&#x621
&#x6709;&#x4e09;&#x79cd;&#x65b9;&#x6cd5;&#x53ef;&#x4ee5;&#x70b9;&#x51fb;&#x4e00;&#x4e2a;&#x6216;&#x591a;&#x4
PCB&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x9ad8;&#x4eae;&#xff1a;&#x70b9;&#x51fb;&#x94dc;&#x5bf9;&#x8c61;&#x540e;&
kbd[ ]&#xff0c;&#x4f7f;&#x7528;&#x4efb;&#x4f55;&#x94dc;&#x5bf9;&#x8c61;&#x7684;&#x4e0a;&#x4e0b;&#x6587;&#x83dc;
(&#x70ed;&#x952e; kbd[~])&#x6e05;&#x9664;&#x9ad8;&#x4eae;&#x3002;
```

&#x9009;&#x62e9;&#x4e00;&#x4e2a;&#x6216;&#x591a;&#x4e2a;&#x7f51;&#x7edc;&#x8fdb;&#x884c;&#x9ad8;&#x4eae;&#x6  
 kbd: [Ctrl+`]&#x8bbf;&#x95ee;)&#x3002;&#x6b64;&#x64cd;&#x4f5c;&#x5c06;&#x6253;&#x5f00;&#x6216;&#x5173;&#x95ed;&

2.5



KiCad &#x5141;&#x8bb8;&#x5728;&#x539f;&#x7406;&#x56fe;&#x548c; PCB &#x4e4b;&#x95f4;&#x8fdb;&#x884c;&#x53cc;&#x6709;&#x51e0;&#x79cd;&#x4e0d;&#x540c;&#x7c7b;&#x578b;&#x7684;&#x4ea4;&#x53c9;&#x63a2;&#x6d4b;&#x3002;

&#x9009;&#x62e9;&#x4ea4;&#x53c9;&#x63a2;&#x6d4b;&#x5141;&#x8bb8;&#x60a8;&#x5728;&#x539f;&#x7406;&#x56fe;&#x5982;&#x679c;&#x5b58;&#x5728;)&#x4e2d;&#x70b9;&#x51fb;&#x76f8;&#x5e94;&#x7684;&#x5c01;&#x88c5;&#x6212;&#x9ed8;&#x8ba4;&#x60c5;&#x51b5;&#x4e0b;&#xff0c;&#x4ea4;&#x53c9;&#x63a2;&#x67e5;&#x5c06;&#x5bfc;&#x81f4;&#x6666;&#x53ef;&#x4ee5;&#x5728;&#x504f;&#x597d;&#x8bbe;&#x7f6e;&#x5bf9;&#x8bdd;&#x6846;&#x7684;&#x663e;&#x793a;&#x9009;

&#x9ad8;&#x4eae;&#x4ea4;&#x53c9;&#x63a2;&#x6d4b; &#x5141;&#x8bb8;&#x60a8;&#x540c;&#x65f6;&#x9ad8;&#x4eae;&#x4e2d;&#x7684;&#x7f51;&#x7edc;&#x3002; &#x5982;&#x679c;&#x5728;&#x504f;&#x597d;&#x8bbe;&#x7f6e;&#x5bf9;  
" &#x9ad8;&#x4eae;&#x4ea4;&#x53c9;&#x7f51;&#x7edc;" &#xff0c; &#x5219;&#x5728;&#x539f;&#x7406;&#x56fe;&#x7f16;&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x9ad8;&#x4eae;&#x3002;  
PCB &#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x9ad8;&#x4eae;&#x3002;

## 2.6

&#x5de6;&#x4fa7;&#x5de5;&#x5177;&#x680f;&#x63d0;&#x4f9b;&#x4e86;&#x66f4;&#x6539;Pcbnew &#x4e2d;&#x9879;&#x76

	&#x6253;&#x5f00;/&#x5173;&#x95ed;&#x6805;&#x683c;&#x663e;&#x793a;&#x3002; &#x6ce8;&#x610f;&#xff1a; &#x9ed8;&#x8ba4;&#x60c5;&#x51b5;&#x4e0b;&#xff0c;&#x9690;&#x85cf;&#x7f51;&#x683c;&#x5c06;&#x7981;&#x &#x53ef;&#x4ee5;&#x5728;&#x504f;&#x597d;&#x8bbe;&#x7f6e;&#x7684;&#x663e;&#x793a;&#x9009;&#x9879;&#x
	&#x5728;&#x72b6;&#x6001;&#x680f;&#x4e2d;&#x7684;&#x6781;&#x5750;&#x6807;&#x548c;&#x7b1b;&#x5361;&#x
	&#x4ee5;&#x82f1;&#x5bf8;&#x3001;&#x5bc6;&#x8033;&#x6216;&#x6beb;&#x7c73;&#x4e3a;&#x5355;&#x4f4d;&#x
	&#x5728;&#x5168;&#x5c4f;&#x548c;&#x5c0f;&#x7f16;&#x8f91;&#x5149;&#x6807; (&#x5341;&#x5b57;&#x5149;&#x6807;) &#x4e4b;&#x95f4;&#x5207;&#x6362;&#x3002;
	&#x6253;&#x5f00;/&#x5173;&#x95ed;&#x98de;&#x7ebf;&#x663e;&#x793a;&#x3002;
	&#x5728;&#x76f4;&#x7ebf;&#x578b;&#x548c;&#x5f27;&#x7ebf;&#x578b;&#x98de;&#x7ebf;&#x4e4b;&#x95f4;&#x
	&#x5728;&#x6b63;&#x5e38;&#x548c;&#x6697;&#x663e;&#x4e4b;&#x95f4;&#x5207;&#x6362;&#x975e;&#x6d3b;&#x &#x6ce8;&#x610f;&#xff1a; &#x5f53;&#x975e;&#x6d3b;&#x52a8;&#x5c42;&#x663e;&#x793a;&#x6a21;&#x5f0f;&#x4e3a;&#x6697;&#x663e;&#x &#x5728;&#x8fd9;&#x4e24;&#x79cd;&#x60c5;&#x51b5;&#x4e0b;&#xff0c;&#x6309;&#x4e0b;&#x6309;&#x94ae;&#x &#x9690;&#x85cf;&#x6a21;&#x5f0f;&#x53ea;&#x80fd;&#x901a;&#x8fc7;&#x5916;&#x89c2;&#x9762;&#x677f;&#x kbd:[Ctrl+H] &#x8fdb;&#x5165;&#x3002;
	&#x9009;&#x62e9;&#x8981;&#x9ad8;&#x4eae;&#x7684;&#x7f51;&#x7edc;&#x65f6;&#xff0c;&#x4f1a;&#x6253;&#x &#x6ce8;&#x610f;&#xff1a; &#x5f53;&#x6ca1;&#x6709;&#x9ad8;&#x4eae;&#x4efb;&#x4f55;&#x7f51;&#x7edc;&#x65f6;&#xff0c;&#x6b64;&#x6 &#x8981;&#x9ad8;&#x4eae;&#x7f51;&#x7edc;&#xff0c;&#x53ef;&#x4f7f;&#x7528;&#x70ed;&#x952e; kbd:[ ]&#xff0c; &#x53f3;&#x51fb;&#x7f51;&#x7edc;&#x4e2d;&#x7684;&#x4efb;&#x4f55;&#x94dc;&#x5bf9;&#x8c61;&#x5e76;&#x &#x6216;&#x53f3;&#x51fb;&#x5916;&#x89c2;&#x9762;&#x677f;&#x7684;&#x7f51;&#x7edc;&#x9009;&#x9879;&#x
	&#x663e;&#x793a;&#x5206;&#x533a;&#x586b;&#x5145;&#x533a;&#x57df;&#x3002;

	&#x4ec5;&#x663e;&#x793a;&#x533a;&#x57df;&#x8f6e;&#x5ed3;&#x3002;
	&#x5c06;&#x5206;&#x533a;&#x586b;&#x5145;&#x533a;&#x57df;&#x663e;&#x793a;&#x4e3a;&#x8f6e;&#x5ed3;&#x
	&#x5728;&#x586b;&#x5145;&#x6a21;&#x5f0f;&#x548c;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e4b;&#x95f4;&#x
	&#x5728;&#x586b;&#x5145;&#x6a21;&#x5f0f;&#x548c;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e4b;&#x95f4;&#x
	&#x5728;&#x586b;&#x5145;&#x6a21;&#x5f0f;&#x548c;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e4b;&#x95f4;&#x
	&#x663e;&#x793a;&#x6577;&#x94dc;&#x586b;&#x5145;&#x533a;&#x57df;&#x3002;
	&#x4ec5;&#x663e;&#x793a;&#x6577;&#x94dc;&#x8f6e;&#x5ed3;&#x3002;
	&#x5c06;&#x6577;&#x94dc;&#x586b;&#x5145;&#x533a;&#x57df;&#x663e;&#x793a;&#x4e3a;&#x8f6e;&#x5ed3;&#x
	&#x5728;&#x586b;&#x5145;&#x6a21;&#x5f0f;&#x548c;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e4b;&#x95f4;&#x
	&#x5728;&#x586b;&#x5145;&#x6a21;&#x5f0f;&#x548c;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e4b;&#x95f4;&#x
	&#x5728;&#x586b;&#x5145;&#x6a21;&#x5f0f;&#x548c;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e4b;&#x95f4;&#x
	&#x663e;&#x793a;&#x6216;&#x9690;&#x85cf;&#x7f16;&#x8f91;&#x5668;&#x53f3;&#x4fa7;&#x7684;&#x5916;&#x



## Chapter 3

## PCB

### 3.1 PCB

KiCad &#x4e2d;&#x7684;&#x5370;&#x5237;&#x7535;&#x8def;&#x677f;&#x901a;&#x5e38;&#x7531;&#x4ee3;&#x8868;&#x7531;&#x5c01;&#x88c5;&#x3001;&#x5b9a;&#x4e49;&#x8fd9;&#x4e9b;&#x710a;&#x76d8;&#x5982;&#x4f55;&#x5f7c;&#x6b64;&#x8868;&#x7f51;&#x7edc;&#x3001;&#x5f62;&#x6210;&#x6bcf;&#x4e2a;&#x7f51;&#x7edc;&#x4e2d;&#x710a;&#x76d8;&#x4e4b;&#x901a;&#x5e03;&#x7ebf;&#x3001;&#x8fc7;&#x5b54;&#x548c;&#x586b;&#x5145;&#x533a;&#x4ee5;&#x53ca;&#x5b9a;&#x4e49;&

KiCad &#x901a;&#x5e38;&#x4f1a;&#x5c06; PCB &#x4e0a;&#x7684;&#x7f51;&#x7edc;&#x4fe1;&#x606f;&#x4e0e;&#x76f8;&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x521b;&#x5efa;&#x548c;&#x7f16;&#x8f91;&#x7f51;&#x7edc;&#x3002;

### 3.2 $\times 6027; \times 80fd;$


KiCad &#x0fd;&#x591f;&#x521b;&#x5efa;&#x591a;&#x8fbe; 32 &#x4e2a;&#x94dc;&#x5c42;&#x3001;14 &#x4e2a;&#x6280;&#x4e1d;&#x5370;&#x3001;&#x963b;&#x710a;&#x3001;&#x5143;&#x4ef6;&#x7c98;&#x5408;&#x5242;&#x3001;&#x710a;&#x548c; 13 &#x4e2a;&#x901a;&#x7528;&#x7ed8;&#x56fe;&#x5c42;&#x7684;&#x5370;&#x5237;&#x7535;&#x8def;&#x677f;&#x

The internal measurement resolution of all objects in KiCad is 1 nanometer, and measurements are stored as 32-bit integers. This means it is possible to create boards up to approximately 4 meters by 4 meters.

KiCad &#x76ee;&#x524d;&#x652f;&#x6301;&#x6bcf;&#x4e2a;&#x5de5;&#x7a0b;/&#x539f;&#x7406;&#x56fe;&#x4e00;&#x4e2a

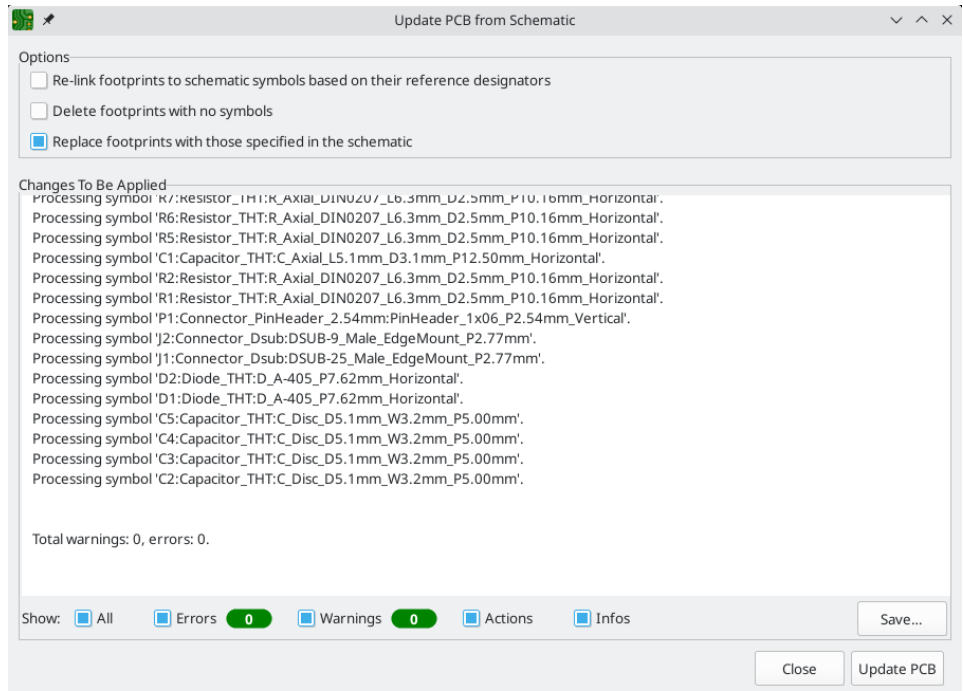
### 3.3 $\times 4\text{ece}; \times 539\text{f}; \times 7406; \times 56\text{fe}; \times 5\text{f00}; \times 59\text{cb};$

Creating a board from a schematic is the recommended workflow for KiCad. When you create a new project, KiCad will generate an empty board file with the same name as the project. To start designing the board after you have created a schematic, simply open the board file. You can do this either from the KiCad project manager, or by clicking the "Open PCB in board editor" button in the schematic editor. To import the schematic design information into the board editor, including footprints and net

connections, use the **Tools** → **Update Schematic from PCB...** action (kbd:[F8]). You can also use the  icon in the top toolbar.

## Note

Update PCB from Schematic is the preferred way to transfer design information from the schematic to the PCB. In older versions of KiCad, the equivalent process was to export a netlist from the Schematic Editor and import it into the Board Editor. It is no longer necessary to use a netlist file.



The tool adds the footprint for each symbol to the board and transfers updated schematic information to the board. In particular, the board's net connections are updated to match the schematic.

The changes that will be made to the PCB are listed in the *Changes To Be Applied* pane. The PCB is not modified until you click the **Update PCB** button.

You can show or hide different types of messages using the checkboxes at the bottom of the window. A report of the changes can be saved to a file using the **Save...** button.

### 3.3.1

The tool has several options to control its behavior.


Option	Description
Re-link footprints to schematic symbols based on their reference designators	Footprints are normally linked to schematic symbols via a unique identifier created when the symbol is added to the schematic. A symbol's unique identifier cannot be changed. If checked, each footprint in the PCB will be re-linked to the symbol that has the same reference designator as the footprint. If unchecked, footprints and symbols will be linked by unique identifier as usual, rather than by reference designator. Each footprint's reference designator will be updated to match the reference designator of its linked symbol. This option should generally be left unchecked. It is useful for specific workflows that rely on changing the links between schematic symbols and footprints, such as refactoring a schematic for easier layout or replicating layout between identical channels of a design.
Delete footprints with no symbols	If checked, any footprint in the PCB without a corresponding symbol in the schematic will be deleted from the PCB. Footprints with the "Not in schematic" attribute will be unaffected. If unchecked, footprints without a corresponding symbol will not be deleted.
Replace footprints with those specified in the schematic	If checked, footprints in the PCB will be replaced with the footprint that is specified in the corresponding schematic symbol. If unchecked, footprints that are already in the PCB will not be changed, even if the schematic symbol is updated to specify a different footprint.

### 3.4 $\times 5934; \times 5f00; \times 59cb;$

&#x4e5f;&#x53ef;&#x4ee5;&#x521b;&#x5efa;&#x6ca1;&#x6709;&#x5339;&#x914d;&#x539f;&#x7406;&#x56fe;&#x7684;&#x75  
PCB&#x7f16;&#x8f91;&#x5668;(&#x800c;&#x4e0d;&#x662f;&#x4ece; KiCad &#x5de5;&#x7a0b;&#x7ba1;&#x7406;&#x5668;&#x  
"&#x53e6;&#x5b58;&#x4e3a;..."&#x4ece;&#x6587;&#x4ef6;&#x83dc;&#x5355;&#x9009;&#x62e9;&#x4fdd;&#x5b58;&#x7535;

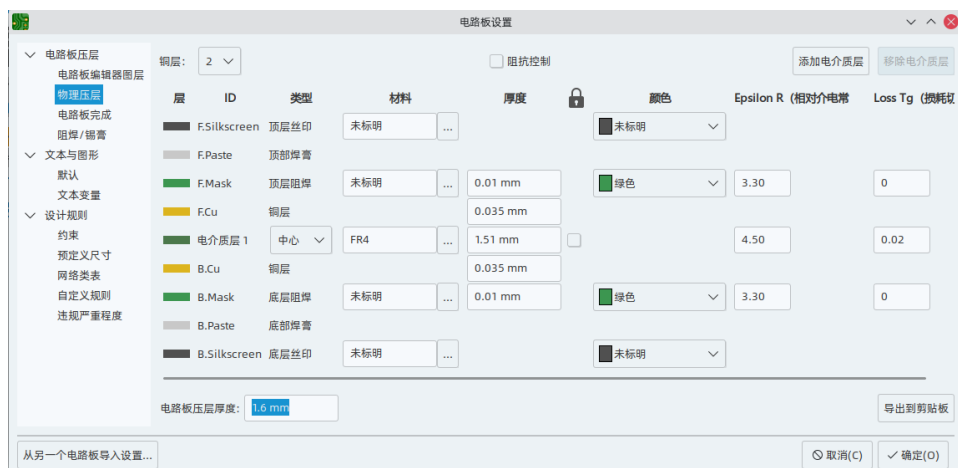
### 3.5 $\int_{-\infty}^{\infty} \delta(x) dx = 1$

&#x5728;&#x5f00;&#x59cb;&#x60a8;&#x7684;&#x7ebf;&#x8def;&#x677f;&#x8bbe;&#x8ba1;&#x4e4b;&#x524d;&#xff0c;&#x4f

 &#x56fe;&#x6807;&#x6216;&#x9009;&#x62e9; "&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;..." &#x4ece;&#x6587;&#x

3.5.1

&#x5728;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#x4e2d;&#x6709;&#x4e24;&#x4e2a;&#x90e8;&#x5206;&#x7528;&#x4  
(&#x975e;&#x94dc;) &#x5c42;&#xff0c;&#x5e76;&#x5728;&#x9700;&#x8981;&#x65f6;&#x4e3a;&#x5c42;&#x6307;&#x5b9a;&#x  
&#x8981;&#x914d;&#x7f6e;&#x7535;&#x8def;&#x677f;&#x7684;&#x538b;&#x5c42;&#xff0c;&#x4ece;&#x7269;&#x7406;&#x53



&#x5728;&#x5de6;&#x4e0a;&#x89d2;&#x8bbe;&#x7f6e;&#x94dc;&#x5c42;&#x7684;&#x6570;&#x91cf;&#xff0c;&#x7136;&#x5-  
&#x8fd9;&#x4e9b;&#x53c2;&#x6570;&#x53ef;&#x4ee5;&#x4fdd;&#x7559;&#x5176;&#x9ed8;&#x8ba4;&#x503c;&#xff0c;&#x4f-  
3D&#x6a21;&#x578b;&#x65f6;&#xff0c;&#x5c06;&#x4f7f;&#x7528;&#x7535;&#x8def;&#x677f;&#x7684;&#x539a;&#x5ea6;&#x7

### Note

KiCad &#x76ee;&#x524d;&#x4ec5;&#x652f;&#x6301;&#x94dc;&#x5c42;&#x6570;&#x4e3a;&#x5076;&#x6570;&#x7684;&#x538b;&#x8981;&#x521b;&#x5efa;&#x5177;&#x6709;&#x5947;&#x6570;&#x5c42;&#x7684;&#x8bbe;&#x8ba1;  
(&#x4f8b;&#x5982;&#xff0c;&#x67d4;&#x6027;&#x5370;&#x5237;&#x7535;&#x8def;&#x677f;&#x6216;&#x91d1;&#x5c5e;&#x82af;&#x53ea;&#x9700;&#x9009;&#x62e9;&#x4e0b;&#x4e00;&#x4e2a;&#x6700;&#x9ad8;&#x7684;&#x5076;&#x6570;&#xff0c;&#x800c;&

&#x63a5;&#x4e0b;&#x6765;&#xff0c;&#x5982;&#x679c;&#x9700;&#x8981;&#xff0c;&#x53ef;&#x4ee5;&#x4f7f;&#x7528;&#x7528;  
 B. Silkreen &#x5c42;&#x65c1;&#x8fb9;&#x7684;&#x590d;&#x9009;&#x6846;&#x3002:



Note

PCB

PCB

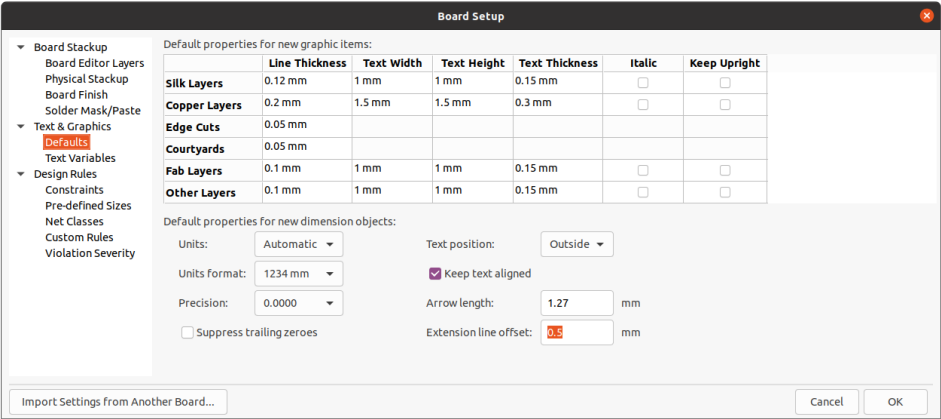
Warning



PCB

3.5.2

PCB



&#x53ef;&#x4ee5;&#x4e3a;&#x5bf9;&#x8bdd;&#x6846;&#x4e2d;&#x663e;&#x793a;&#x7684;&#x516d;&#x79cd;&#x4e0d;&#x53ef;&#x672c;&#x66ff;&#x6362;&#x53d8;&#x91cf;&#x53ef;&#x4ee5;&#x5728;&#x6587;&#x672c;&#x53d8;&#x91cf;&#x90&#x8fd9;&#x4e9b;&#x53d8;&#x91cf;&#x5141;&#x8bb8;&#x4f60;&#x5c06;&#x53d8;&#x91cf;&#x540d;&#x79f0;&#x66ff;&#x63&#x8fd9;&#x79cd;&#x66ff;&#x6362;&#x53d1;&#x751f;&#x5728;&#x53d8;&#x91cf;&#x540d;&#x79f0;&#x5728; \$ {VARIABLEN&#x7684;&#x53d8;&#x91cf;&#x66ff;&#x6362;&#x8bed;&#x6cd5;&#x5185;&#x7684;&#x4efb;&#x4f55;&#x5730;&#x65b9;&#x30&#x4f8b;&#x5982;&#xff0c;&#x60a8;&#x53ef;&#x4ee5;&#x521b;&#x5efa;&#x4e00;&#x4e2a;&#x540d;&#x4e3a; VERSION&#x7684;&#x53d8;&#x91cf;&#xff0c;&#x5e76;&#x5c06;&#x6587;&#x672c;&#x66ff;&#x6362;&#x8bbe;&#x7f6e;&#x4e3a; 1 . 0 &#x73b0;&#x5728;&#xff0c;&#x5728; PCB &#x4e0a;&#x7684;&#x4efb;&#x4f55;&#x6587;&#x672c;&#x5bf9;&#x8c61;&#x4e2d;d\$ {VERSION} &#xff0c;KiCad &#x5c06;&#x66ff;&#x4ee3; 1 . 0 ' &#x3002; &#x5982; &#x679c; &#x4f60; &#x628a; &#x66` 2 . 0 ' &#xff0c; &#x6bcf; &#x4e2a; &#x5305; &#x62ec; ` \$ {VERSION} &#x7684;&#x6587;&#x672c;&#x5bf9;&#x8c61&#x4f60;&#x4e5f;&#x53ef;&#x4ee5;&#x6df7;&#x5408;&#x4f7f;&#x7528;&#x666e;&#x901a;&#x6587;&#x672c;&#x548c;&#x53&#x4f8b;&#x5982;&#xff0c;&#x4f60;&#x53ef;&#x4ee5;&#x521b;&#x5efa;&#x4e00;&#x4e2a;&#x6587;&#x672c;&#x5bf9;&#x8c6&#x7248; &#x672c; &#xff1a; \$ {VERSION} &#xff0c;&#x5b83;&#x5c06;&#x88ab;&#x66ff;&#x6362;&#x4e3a; &#x7248; &#

3.5.3

&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x63a7;&#x5236;&#x4ea4;&#x4e92;&#x5f0f;&#x5e03;&#x7ebf;&#x5668;&#x7684;&#x8

3.5.3.1

&#x57fa;&#x672c;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x662f;&#x5728;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#x5b&#x8fd9;&#x4e00;&#x90e8;&#x5206;&#x7684;&#x7ea6;&#x675f;&#x6761;&#x4ef6;&#x9002;&#x7528;&#x4e8e;&#x6574;&#x4&#x8fd9;&#x91cc;&#x8bbe;&#x7f6e;&#x7684;&#x4efb;&#x4f55;&#x6700;&#x5c0f;&#x503c;&#x90fd;&#x662f;&#x4e00;&#x4e2&#x7edd;&#x5bf9;(absolute) &#x7684;&#x6700;&#x5c0f;&#x503c;&#xff0c;&#x4e0d;&#x80fd;&#x88ab;&#x66f4;&#x5177;&#x4f&#x4f8b;&#x5982;&#xff0c;&#x5982;&#x679c;&#x4f60;&#x9700;&#x8981;&#x7535;&#x8def;&#x677f;&#x7684;&#x4e00;&#x900.2mm&#xff0c;&#x5176;&#x4f59;&#x90e8;&#x5206;&#x4e3a; 0.3mm&#xff0c;&#x4f60;&#x5fc5;&#x987b;&#x5728;&#x7ea6;&#x&#x5176;&#x7684;&#x6700;&#x5c0f;&#x94dc;&#x95f4;&#x9699;&#xff0c;&#x5e76;&#x4f7f;&#x7528;&#x7f51;&#x7edc;&#x7c7b;&#x5176;&#x7684;&#x6700;&#x5c0f;&#x94dc;&#x95f4;&#x9699;&#x3002;

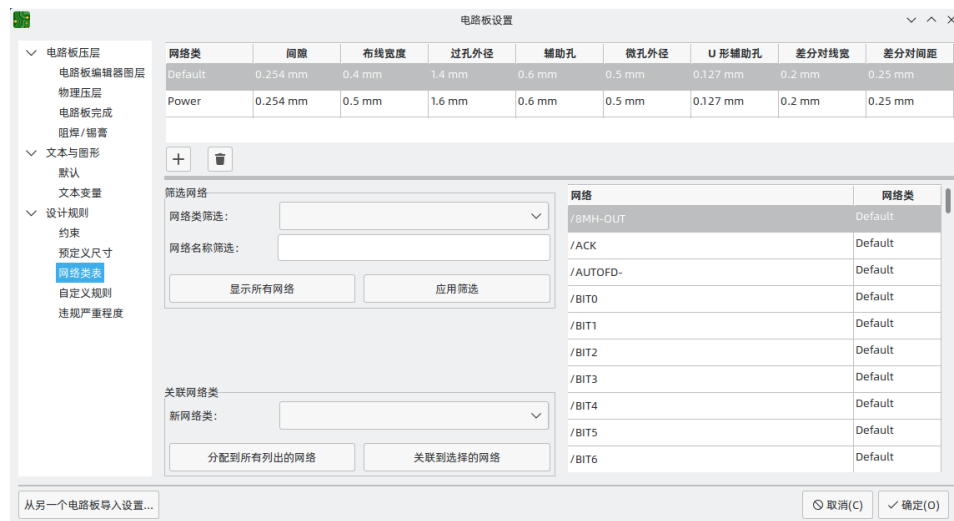


&#x9664;&#x4e86;&#x8bbe;&#x7f6e;&#x6700;&#x5c0f;&#x95f4;&#x9699;&#x5916;&#xff0c;&#x8fd8;&#x53ef;&#x4ee5;&#x5728;

&#x8bbe;&#x7f6e;	&#x63cf;&#x8ff0;
&#x5141;&#x8bb8;&#x76f2;&#x54c8;&#x5987;&#x5154;&#x542f;&#x7528;&#x6b64;&#x8bbe;&#x7f6e;&#xff0c;&#x7136;&#x76f2;&#x5b54;&#x662f;&#x673a;&#x68b0;&#x94bb;&#x5b54;&#xff0c;&#x4ece;&#x5916;&#x57cb;&#x5b54;&#x662f;&#x673a;&#x68b0;&#x94bb;&#x5b54;&#xff0c;&#x5728;&#x5185;	
&#x5141;&#x8bb8;&#x5fae;&#x5b57;&#x5728;&#x4f7f;&#x7528;&#x5e03;&#x7ebf;&#x5668;&#x653e;&#x7f6e;&#x5fae;&#x5b54;&#x5fae;&#x5b54;&#x662f;&#x5178;&#x578b;&#x7684;&#x6fc0;&#x5149;&#x94bb;&#x5b54;&#x5176;&#x7684;&#x6700;&#x5c0f;&#x94dc;&#x95f4;&#x9699;&#x3002;	



&#x5982;&#x679c;&#x4f60;&#x4e0d;&#x628a;&#x4e00;&#x4e2a;&#x7f51;&#x7edc;&#x6dfb;&#x52a0;&#x5230;&#x4e00;&#x4e00;&#x98de;&#x7ebf;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x539f;&#x7406;&#x56fe;&#x8bbe;&#x7f6e;&#x548c;&#x7535;&#x8d00;&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x6dfb;&#x52a0;&#x5230;&#x7f51;&#x7edc;&#x7c7b;&#x3002;



&#x7f51;&#x7edc;&#x7c7b;&#x90e8;&#x5206;&#x7684;&#x4e0a;&#x90e8;&#x5305;&#x542b;&#x4e00;&#x4e2a;&#x8868;&#x6bcf;&#x4e2a;&#x7f51;&#x7edc;&#x7c7b;&#x90fd;&#x6709;&#x94dc;&#x95f4;&#x9699;&#x3001;&#x5e03;&#x7ebf;&#x5b8d9;&#x4e9b;&#x503c;&#x5c06;&#x5728;&#x521b;&#x5efa;&#x5e03;&#x7ebf;&#x548c;&#x8fc7;&#x5b54;&#x65f6;&#x4f

### Note

0.1 mm&#xff0c;&#x4f6;&#x7ea6;&#x675f;&#x6761;&#x4ef6;&#x90e8;&#x5206;&#x4e2d;&#x7684;&#x6700;&#x5c0f;&#x95f4;&#x80.2 mm&#xff0c;&#x5219;&#x8be5;&#x7c7b;&#x7f51;&#x7edc;&#x7684;&#x95f4;&#x8ddd;&#x5c06;&#x4e3a; 0.2 mm&#x3002;

[illegible]

#### 3.5.3.4 $\frac{1}{2} \frac{1}{2} \frac{1}{2} \frac{1}{2} \frac{1}{2}$

&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x90e8;&#x5206;&#x5305;&#x542b;&#x4e00;&#x4e2a;&#x6587;&#x672c;&#x  
&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x7528;&#x4e8e;&#x521b;&#x5efa;&#x57fa;&#x672c;&#x7ea6;&#x675f;&#x62  
&#x53ea;&#x6709;&#x5728;&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x5b9a;&#x4e49;&#x4e2d;&#x6ca1;&#x6709;&#x9  
&#x5728;&#x5173;&#x95ed;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#x4e4b;&#x524d;&#xff0c;&#x4f7f;&#x7528;&#x68  
&#x8bf7;&#x53c2;&#x9605;&#x9ad8;&#x7ea7;&#x4e3b;&#x9898;&#x4e00;&#x7ae0;&#x4e2d;&#x7684;&#x81ea;&#x5b9a;&#x4





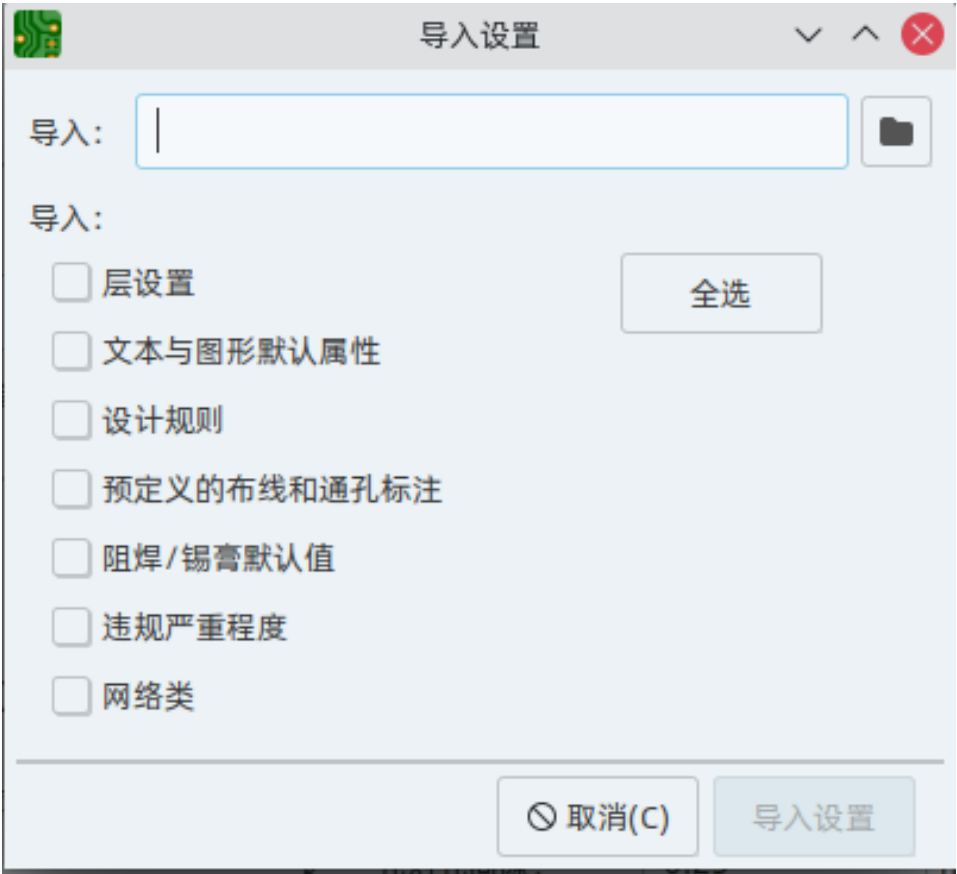
3.5.3.5

Note



3.5.3.6












PCB 文件  
kicad\_pcb













# Chapter 4

## 



### 4.1

This section describes the tools used in the PCB design process. The tools are categorized into several groups: Selection tool, Local ratsnest tool, Footprint placement tool, Route tracks / route differential pairs, Tune length, Add vias, and Add filled zone. Each tool is described in detail, including its function and how to use it.

	Selection tool (the default tool).
	Local ratsnest tool: when the board ratsnest is hidden, selecting footprints with this tool will show the ratsnest for the selected footprint only. Selecting the same footprint again will hide its ratsnest. The local ratsnest setting for each footprint will remain in effect even after the local ratsnest tool is no longer active.
	Footprint placement tool: click on the board to open the footprint chooser, then click again after choosing a footprint to confirm its location.
	Route tracks / route differential pairs: These tools activate the interactive router and allow placing tracks and vias. The interactive router is described in more detail in the Routing Tracks section below.
	Tune length: These tools allow you to tune the length of single tracks or the length or skew of differential pairs, after they have been routed. See the Routing Tracks section for details.
	Add vias: allows placing vias without routing tracks. Vias placed on top of tracks using this tool will take on the net of the closest track segment and will become part of that track (the via net will be updated if the pads connected to the tracks are updated). Vias placed anywhere else will take on the net of a copper zone at that location, if one exists. These vias will not automatically take on a new net if the net of the copper zone is changed.
	Add filled zone: Click to set the start point of a zone, then configure its properties before drawing the rest of the zone outline. Zone properties are described in more detail below.

	Add rule area: Rule areas, formerly known as keepouts, can restrict the placement of items and the filling of zones and can also define named areas to apply specific custom design rules to.
	Draw lines. <b>Note:</b> Lines are graphical objects and are not the same as tracks placed with the Route Tracks tool. Graphical objects cannot be assigned to a net.
	Draw arcs: pick the center point of the arc, then the start and end points.
	Draw rectangles. Rectangles can be filled or outlines.
	Draw circles. Circles can be filled or outlines.
	Draw graphical polygons. Polygons can be filled or outlined. <b>Note:</b> Filled graphical polygons are not the same as filled zones: graphical polygons cannot be assigned to a net and will not keep clearance from other items.
	Add text.
	Add dimensions. Dimension types are described in more detail below.
	Add layer alignment mark.
	Deletion tool: click objects to delete them.
	Set drill/place origin. Used for fabrication outputs.
	Set grid origin.

## 4.2

	
kbd:[Ctrl]	
kbd:[Shift]	

## 4.3

8981;60f3;4e00;6b21;7f16;8f91;4e0d;540c;7c7b;578b;7684;9879;76ee;x7
 5728;5c5e;6027;5bf9;8bdd;6846;4e2d;ff0c;4efb;4f55;5305;542b;6570;51
 4f8b;5982;xff0c;4e00;4e2a;5c3a;5bf8;53ef;4ee5;88ab;8f93;5165;4e3a; 2
 \* 2mm;ff0c;7ed3;679c;662f; 4mm;7684;503c;3002; 652f;6301;57fa;672c;7b97;#

4.4

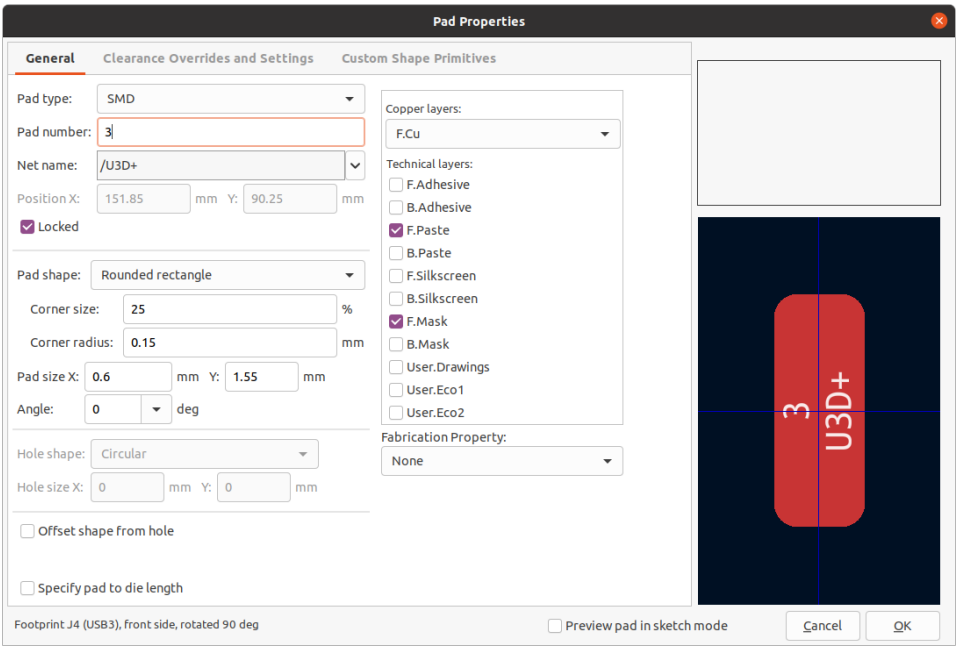
**Note**  
 TODO: 7f16;5199;8fd9;4e00;8282;—5305;62ec;5c01;88c5;5c5e;6027;xff0c;4ece;#

4.5

The properties of each individual pad of a footprint can be inspected and edited after placing the footprint on the board. In other words, it is possible to override the design of an individual footprint pad in a specific instance of the footprint on the board, if the footprint design in the library is not appropriate. For example, you may wish to remove the solder paste aperture for a pad that needs to remain unsoldered in a specific design, or you may wish to move the location of a through-hole pad for an axial-lead resistor in order to fit a specific design.

**Note**  
 By default, the position of all footprint pads are locked, so it is possible to edit the pad properties but not move the pad's location relative to the rest of the footprint. Pads may be unlocked to allow free movement, which can be useful for certain applications (such as through-hole footprints with varying lead positions) but is generally never recommended for surface-mount footprints.

The pad properties dialog is opened through the context menu or default hotkey kbd:[E] when a pad is selected. Note that KiCad assumes that if you click near a pad, you are probably trying to select the entire footprint rather than a single pad. To select a single pad, make sure to click inside the pad area, or turn off the Footprints setting in the selection filter (and make sure the Pads setting is turned on) to prevent accidental selection of the entire footprint rather than a specific pad.



The General tab of the pad properties dialog shows the physical properties of the pad, including its geometry, shape, and layer settings.

**Pad type:** this setting controls which features are enabled for the pad:

**SMD** pads are electrically-connected and have no hole. In other words, they exist on a single copper layer.

**Through-hole** pads are electrically-connected and have a plated hole. The hole exists on every layer, and the copper pad exists on multiple layers (see **Copper layers** setting below).

**Edge Connector** pads are SMD pads that are allowed to overlap the board outline on the Edge.Cuts layer.

**NPTH, Mechanical** pads are non-plated through holes that do not have an electrical connection.

**SMD Aperture** pads are pads that have no hole and no electrical connection. These can be used to add specific designs to a technical layer, for example a paste or solder mask aperture.

The **Copper layers** setting controls which copper layers will have a shape associated with the pad.

For SMD pads, the options are F.Cu or B.Cu, controlling whether the pad sits on the front or the back of the board *relative to the footprint's location*. In other words, if a pad is set to exist on B.Cu in its properties, and the footprint is flipped to the back of the board, *that pad will now exist on F.Cu, because it also has been flipped*.

For through-hole pads, it is possible to remove the pad shape from copper layers where the pad is not electrically connected to other copper (tracks or filled zones). Setting the copper layers to **connected layers only** will remove the pad shape from any unconnected layers, and setting to **F.Cu, B.Cu, and connected layers** will remove the pad shape from any internal unconnected layers. This can be useful in dense board designs to increase the routable area on internal layers.

The **Technical layers** checkboxes control which technical layers will have an aperture added with the pad's shape. By default, pads have apertures on the paste and mask layers matching their copper layer.

---

#### Note

It is not possible to define a different pad shape or size on different copper layers in the current version of KiCad.

---

**Specify pad to die length:** This setting allows a length to be associated with this pad that will be added to the routed track length by the track length tuning tools and the Net Inspector. This can be used to specify internal bondwire lengths for more accurate length matching, or in other situations where the electrical length of a net is longer than the length of the routed tracks on the board.

**Pad Properties**

General **Clearance Overrides and Settings** Custom Shape Primitives

**Clearances**  
Set values to 0 to use parent footprint or netclass values.  
Positive clearance means area bigger than the pad (usual for mask clearance).  
Negative clearance means area smaller than the pad (usual for paste clearance).

Pad clearance:  mm

Solder mask clearance:  mm

Solder paste absolute clearance:  mm

Solder paste relative clearance:  %

Note: solder mask and paste values are used only for pads on copper layers.  
Note: solder paste clearances (absolute and relative) are added to determine the final clearance.

**Connection to Copper Zones**

Pad connection:

Thermal relief spoke width:  mm

Thermal relief gap:  mm

Custom pad shape in zone:

Footprint: J4 (USB3), front side, rotated 90 deg

☐ Preview pad in sketch mode

Cancel OK

The aperture appearing on any technical layer will have the same shape and size as the pad shape on the copper layer(s). In the PCB manufacturing process, the manufacturer will often change the relative size of mask and paste apertures relative to the copper pad size, but since this size change is specific to a manufacturing process, most manufacturers expect the design data to be provided with the apertures set to the same size as the copper pads. For specific situations where you need to oversize or undersize a technical layer aperture in the design data, you can use the settings in the Overrides tab.

**Pad clearance** controls the minimum clearance between the pad and any copper shape (tracks, vias, pads, zones) on a different net. This value is normally set to 0 which will cause the pad clearance to be inherited from any clearance override set on the footprint, or the board's design rules and netclass rules if the footprint clearance is also set to 0.

**Solder mask clearance** controls the size difference between the pad shape and the aperture shape on the F.Mask and B.Mask layers. A positive number means the solder mask aperture will be larger than the copper shape. This number is an inflation applied to all directions. For example, a value of 0.1mm here will cause the solder mask aperture to be inflated by 0.1mm, meaning that there will be an 0.1mm border on all sides of the pad and the solder mask opening will be 0.2mm wider than the pad when measured along a given axis.

**Solder paste absolute clearance** controls the size difference between the pad shape and the aperture shape on the F.Paste and B.Paste layers. Its behavior is otherwise identical to the behavior of the **solder mask clearance** setting.

**Solder paste relative clearance** allows setting a solder paste clearance value as a percentage of the pad size rather than an absolute distance value. If both relative and absolute clearances are specified, they are added together to determine the solder paste aperture size.

The Overrides tab also has controls for how the pad connects to any copper zone that overlaps it and shares its net.

**Pad connection** controls whether the pad will have a solid, thermal relief, or no connection to the zone. Like the other overrides, this one may be set for an individual pad or for an entire footprint. The default setting for this control is *From parent footprint*, and the default footprint setting is to use the connection mode specified in the zone properties.

**Thermal relief spoke width** controls the width of the spokes generated when the zone connection mode is Thermal Relief.

**Thermal relief gap** controls the length of the thermal spokes, or the gap between the pad's shape and the filled copper area of the zone.

**Custom pad shape in zone** controls the behavior of the zone filler when the pad uses a custom shape rather than one of the default shapes. This can be used to achieve different results when using thermal reliefs and custom pad shapes.

## 4.6 &#x4f7f;&#x7528;&#x533a;&#x57df;

Copper zones, also sometimes called copper pours or fills by other EDA tools, are solid or hatched areas of copper assigned to a particular net that automatically keep clearance from other copper objects. Zones are commonly used to fill in all free space on a board layer (or a portion of a layer) in order to create ground and power planes, carry high currents, or to provide shielding.

---

### Note

Some EDA tools have separate tools for creating "plane layers" and for creating copper zones on signal layers. In KiCad, the Copper Zone tool is used for both these applications.

---

Zones are defined by a polygonal **outline** that defines the maximum extent of the filled copper area. This outline does not represent physical copper and will not appear in exported manufacturing data. The actual copper areas of the zone must be **filled** each time the outline, or any objects inside the outline, are modified. The filling process may be run on a single zone, or on all zones in a board (default hotkey kbd:[B]). Zones may be **unfilled** (default hotkey kbd:[Ctrl+B]) to improve performance and reduce visual clutter while editing large boards.


---

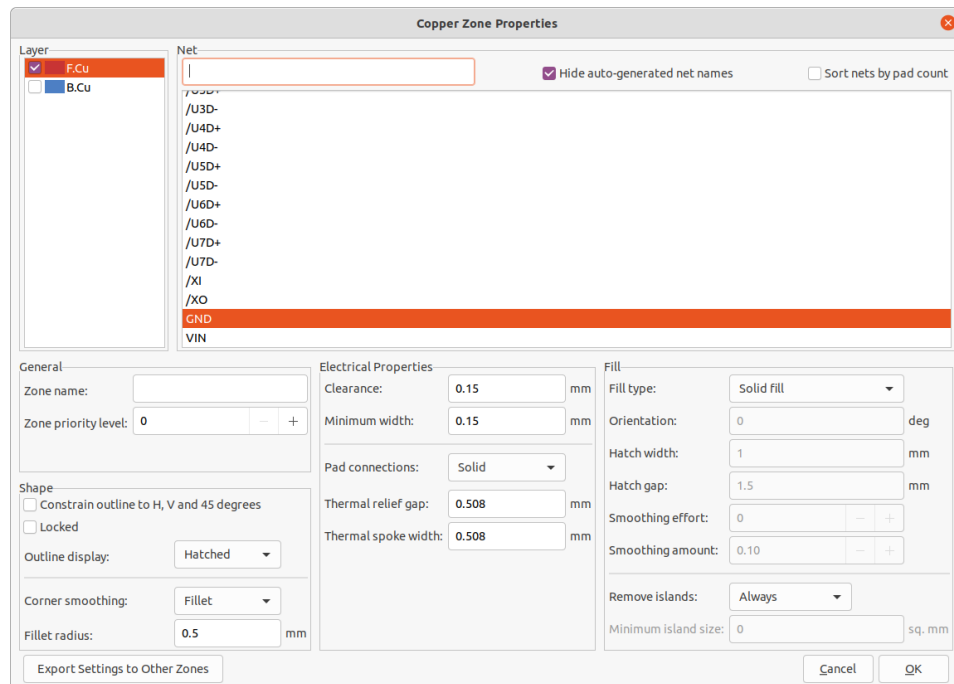
### Note

Zone filling is a manual process rather than occurring every time an object changes that would result in a change to the zone copper. This is because zone filling can be a slow process on older computers or very large designs. It is important to make sure zone fills are up-to-date before generating outputs. KiCad will check that zones have been updated and warn you before generating outputs or running DRC when zones have not yet been refilled.

---



To draw a zone, click the Add Filled Zone tool (  ) on the right toolbar, or use default hotkey kbd:[Ctrl+Shift+Z]. Click to choose the first point of the zone outline. The Zone Properties dialog will appear, allowing you to choose the zone net and other properties. These properties may be edited at any time, so it is not critical to choose them all correctly at first. Accept the dialog and continue placing points to define the zone outline. To finish the zone, double-click to set the last point. Zone outline points may be modified like graphic polygons, by dragging the square handles to move a corner or dragging the circular handles to move an edge. To edit the zone's properties, use hotkey kbd:[E] or select Properties from the context menu.



The image shows the 'Copper Zone Properties' dialog box. It has several sections: 'Layer' with checkboxes for F.Cu and B.Cu; 'Net' with a list of nets including /U3D-, /U4D+, /U4D-, /U5D+, /U5D-, /U6D+, /U6D-, /U7D+, /U7D-, /X1, /X0, GND (highlighted), and VIN; 'General' with fields for Zone name and Zone priority level; 'Shape' with checkboxes for 'Constrain outline to H, V and 45 degrees' and 'Locked', and dropdowns for 'Outline display' (set to Hatched) and 'Corner smoothing' (set to Fillet); 'Electrical Properties' with fields for Clearance, Minimum width, Pad connections (set to Solid), Thermal relief gap, and Thermal spoke width; and 'Fill' with a dropdown for Fill type (set to Solid fill), Orientation, Hatch width, Hatch gap, Smoothing effort, Smoothing amount, Remove islands (set to Always), and Minimum island size. There are 'Export Settings to Other Zones', 'Cancel', and 'OK' buttons at the bottom.

**Layer:** A single zone object can create filled copper on one or more copper layers. Check the box next to each copper layer that this zone outline should fill on. The copper on each layer will be filled independently, but all layers will share the same net.

**Net:** Select the electrical net that the zone copper should be connected to. It is possible to create zones with no net assignment. Zones with no net will keep clearance from any copper objects on any net.

**Zone name** can be used to assign a specific name to a zone. This name can be used to refer to the zone in custom DRC rules.

**Zone priority level** determines the order in which multiple zones on a single layer are filled. The highest priority level zone on a given layer will be filled first. Lower-priority zones will keep clearance to the filled areas of higher-priority zones. Two zones on the same layer with the same priority level will overlap (short-circuit) with each other.



#### Warning

Zones with the same priority level will never keep clearance from each other, even if they are assigned to different nets! The design rule checker will report these short-circuits, but they will not be prevented by the zone filler.

**Constrain outline to H, V and 45 degrees** controls the *initial* behavior of the zone outline drawing tool. When this option is enabled, the zone outline will be restricted to 45-degree angles. Note that after the zone outline has been created, this option has no effect. Outline points may be modified freely after creation.

**Locked** controls whether or not the zone outline object is locked. Locked objects may not be manipulated or moved, and cannot be selected unless the **Locked Items** option is enabled in the Selection Filter panel.

**Outline display** controls how the zone outline is drawn on screen. In **Line** mode, only the border lines of the outline are drawn. In **Hatched** mode, hatch lines are drawn on the inside of the outline border for a short distance, to make the zone outline more apparent. In **Fully Hatched** mode, hatch lines are drawn across the entire inside of the zone outline.

**Corner smoothing** controls the behavior of the filled copper areas at corners of the outline. Corners can be smoothed by a chamfer or fillet, or can extend all the way to the outline corner if smoothing is disabled.

---

#### Note

By default, chamfers and fillets are not added to **inside corners** of the zone outline, because this would result in filled copper extending *outside* the outline. If smooth inside corners are desired, enable the **Allow fillets outside zone outline** option in the Constraints section of the Board Setup dialog.

---

**Clearance** controls the minimum clearance the filled areas of this zone will keep from other copper objects. Note that if two clearance values are in conflict, the larger clearance value will be used. For example, if a zone is set to use 0.2mm clearance but its netclass is set to use 0.3mm clearance, the result will be an 0.3mm clearance.

**Minimum width** controls the minimum size of narrow necks of copper created inside the zone. Any copper areas that would be below this minimum width are removed during the filling process.

**Pad connection** controls the way that the filled zone areas will connect to footprint pads on the same net. **Solid** connections will result in the copper completely overlapping the pads. **Thermal reliefs** will result in small copper spokes connecting the pad to the rest of the copper zone, increasing the thermal resistance between the pad and the rest of the zone. This can be useful for hand soldering. **Reliefs for PTH** will apply thermal reliefs to plated through-hole pads and use solid connections for surface mount pads. **None** will result in the zone not connecting to any pads on the same net.

**Thermal relief gap** controls the distance maintained between any pad and the copper zone when the pad connection mode is set to generate thermal reliefs.

**Thermal spoke width** controls the width of the "spokes", or short copper segments connecting the pad to the rest of the copper zone.

**Fill type** controls how the copper zone is filled: the default is **solid fill**, which will result in copper filling in all available space within the zone outline. The zone can also be set to fill a **hatch pattern**, which will fill the area with a pattern that contains less copper. This can be useful for flexible printed circuits and other specialty applications.

**Orientation** controls the angle of the hatch pattern lines. An orientation of 0 degrees will result in the hatch pattern using horizontal and vertical lines.

**Hatch width** controls the width of each line in the hatch pattern.

**Hatch gap** controls the distance between each line in the hatch pattern.

**Smoothing effort** controls the style of smoothing applied to the hatch pattern. A value of 0 will result in no smoothing, and a value of 3 will result in the finest smoothing. Higher values will result in longer processing time and larger Gerber files.

**Smoothing amount** is a ratio that controls the size of the smoothing chamfers or fillets that are generated when **smoothing effort** is set to a value other than 0. An amount of 0.0 results in no smoothing, and a value of 1.0 results in maximum smoothing (in other words, a chamfer or fillet equal to half of the hatch gap).

**Remove islands** controls the behavior of isolated copper areas, also called islands, after the initial zone fill. When this is set to **always**, isolated areas inside the zone are removed. When set to **never**, isolated areas are left alone, and will result in copper areas that are not connected to the rest of the net. When set to **below area limit**, a **minimum island size** can be specified, and islands below this threshold will be removed.






## 4.7 &#x56fe;&#x5f62;&#x5bf9;&#x8c61;

&#x56fe;&#x5f62;&#x5bf9;&#x8c61; (&#x76f4;&#x7ebf;&#x3001;&#x5706;&#x5f27;&#x3001;&#x77e9;&#x5f62;&#x3001;&#x5668;&#x7981;&#x7528;&#x8f6e;&#x5ed3;&#x3002;

### 4.7.1 &#x6b63;&#x5728;&#x521b;&#x5efa;&#x56fe;&#x5f62;&#x5f62;&#x72b6;

---



The right toolbar can be used to create lines ( , default hotkey kbd:[Ctrl+Shift+L]), arcs ( , default hotkey kbd:[Ctrl+Shift+A]), rectangles ( , circles ( , default hotkey kbd:[Ctrl+Shift+C]), and polygons ( , default hotkey kbd:[Ctrl+Shift+P]).

Rectangles, circles, and polygons can be filled shapes or outlines. The **line width** option controls the width of the outline. The outline width extends on both sides of the "ideal" shape of the graphic object. For example, a graphic circle that is defined to have 2mm radius and 0.2mm line width will consist of a torus with an outer radius of 2.2mm and inner radius of 1.8mm. If the **filled shape** option is enabled and the line width is set to 0, the shape will be a filled circle with 2mm radius.

Line Segment Properties

Start Point

End Point

X: 148.33 mm

X: 151.68 mm

Y: 77.28 mm

Y: 77.28 mm

☐ Locked


Line width: 0.2 mm

Layer: F.Cu

Cancel

OK

4.7.2

Graphical text may be placed by using the ( ) icon in the right toolbar or by keyboard shortcut kbd:[Ctrl+Shift+T]. Click to place the text origin, and then edit the text and its properties in the dialog that will appear:



Text may be placed on any layer, but note that text on copper layers cannot be associated with a net and cannot form connections to tracks or pads. Copper zones will fill around the rectangular bounding box of text objects.

#### 4.7.3 Text Properties (Text Properties)

KiCad Edge . Cuts &#x5c42;&#x4e0a;&#x7684;&#x56fe;&#x5f62;&#x5b9;&#x8c61;&#x6765;&#x5b9a;&#x4e0a;&#x8be5;&#x8f6e;&#x5ed3;&#x5fc5;&#x987b;&#x662f;&#x4e00;&#x4e2a;&#x8fde;&#x7eed;&#xff08;&#x5c01;&#x95ed;&#xff09;&#x5982;&#x679c;&#x6ca1;&#x6709;&#x5b9a;&#x4e49;&#x7535;&#x8def;&#x677f;&#x7684;&#x8f6e;&#x5ed3;&#xff0c;&#x623D&#x67e5;&#x770b;&#x5668;&#x548c;&#x4e00;&#x4e9b;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x7684;&#x68c0;&#x67e5;&#x6807;&#x6ce8;

## 4.8 Text Properties

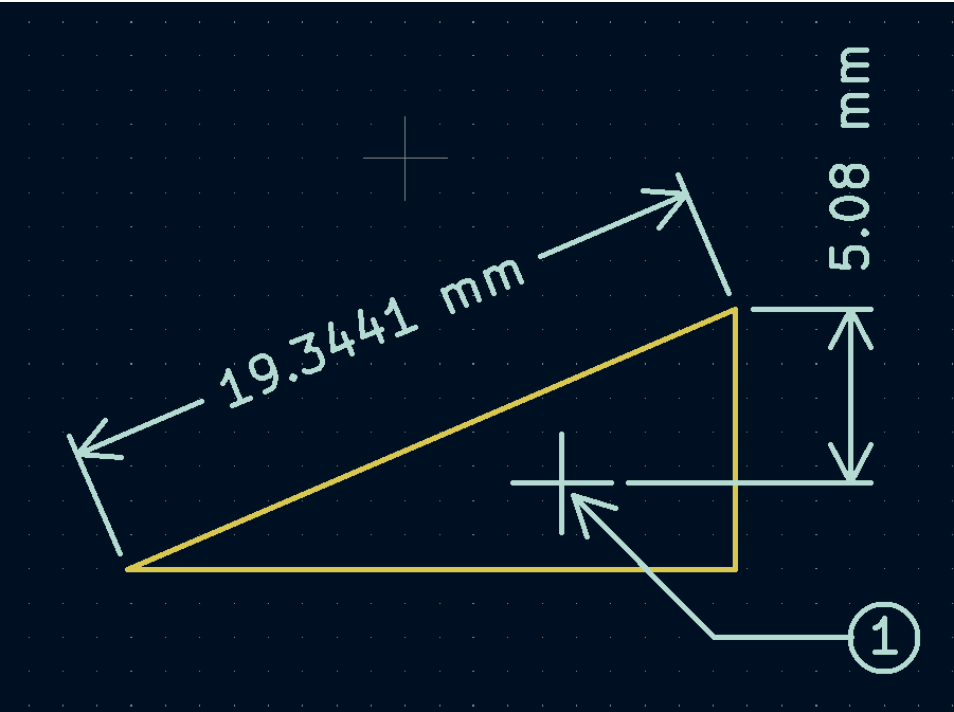
&#x6807;&#x6ce8;&#x662f;&#x7528;&#x4e8e;&#x663e;&#x793a;&#x6d4b;&#x91cf;&#x503c;&#x6216;&#x7535;&#x8def;&#x6b83;&#x4eec;&#x53ef;&#x4ee5;&#x88ab;&#x6dfb;&#x52a0;&#x5230;&#x4efb;&#x4f55;&#x7ed8;&#x56fe;&#x5c42;&#x4e0a;&#x76ee;&#x524d;&#x652f;&#x6301;&#x56db;&#x79cd;&#x4e0d;&#x540c;&#x7c7b;&#x578b;&#x7684;&#x6807;&#x6ce8;

&#x5bf9;&#x9f50; &#x6807;&#x6ce8; ( ) &#x8868;&#x793a;&#x4e24;&#x70b9;&#x4e4b;&#x95f4;&#x8ddd;&#x79bb;&#x7684;&#x6d4b;&#x91cf;&#x8f74;&#x662f;&#x8fde;&#x63a5;&#x8fd9;&#x4e24;&#x4e2a;&#x70b9;&#x7684;&#x7ebf;&#xff0c;&#x6807;&#x6ce8;

&#x6b63;&#x4ea4; &#x6807;&#x6ce8; ( ) &#x4e5f;&#x6d4b;&#x91cf;&#x4e24;&#x70b9;&#x4e4b;&#x95f4;&#x7684;&#x8868;&#x8f74;&#x6216; Y &#x8f74;&#x3002; &#x6362;&#x53e5;&#x8bdd;&#x8bf4;&#xff0c;&#x8fd9;&#x4e9b;&#x6807;&#x6ce8;&#x521b;&#x5efa;&#x6b63;&#x4ea4;&#x6807;&#x6ce8;&#x65f6;&#xff0c;&#x5728;&#x9009;&#x62e9;&#x8981;&#x6d4b;&#x91cf;&#x503c;&#x6216; &#x7528;&#x4e8e;&#x663e;&#x793a;&#x6d4b;&#x91cf;&#x503c;&#x6216; &#x7535;&#x8def;&#x677f;&#x7684;&#x8f6e;&#x5ed3;&#xff0c;&#x623D&#x67e5;&#x770b;&#x5668;&#x548c;&#x4e00;&#x4e9b;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x7684;&#x68c0;&#x67e5;&#x6807;&#x6ce8;

&#x4e2d;&#x5fc3; &#x6807;&#x6ce8; ( ) &#x521b;&#x5efa;&#x4e00;&#x4e2a;&#x5341;&#x5b57;&#x6807;&#x8bb0;&#x6807;&#x6ce8;

1. ( ) 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. 13. 14. 15. 16. 17. 18. 19. 20. 21. 22. 23. 24. 25. 26. 27. 28. 29. 30. 31. 32. 33. 34. 35. 36. 37. 38. 39. 40. 41. 42. 43. 44. 45. 46. 47. 48. 49. 50. 51. 52. 53. 54. 55. 56. 57. 58. 59. 60. 61. 62. 63. 64. 65. 66. 67. 68. 69. 70. 71. 72. 73. 74. 75. 76. 77. 78. 79. 80. 81. 82. 83. 84. 85. 86. 87. 88. 89. 90. 91. 92. 93. 94. 95. 96. 97. 98. 99. 100.



1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. 13. 14. 15. 16. 17. 18. 19. 20. 21. 22. 23. 24. 25. 26. 27. 28. 29. 30. 31. 32. 33. 34. 35. 36. 37. 38. 39. 40. 41. 42. 43. 44. 45. 46. 47. 48. 49. 50. 51. 52. 53. 54. 55. 56. 57. 58. 59. 60. 61. 62. 63. 64. 65. 66. 67. 68. 69. 70. 71. 72. 73. 74. 75. 76. 77. 78. 79. 80. 81. 82. 83. 84. 85. 86. 87. 88. 89. 90. 91. 92. 93. 94. 95. 96. 97. 98. 99. 100.

**Note**

1. 2. 3. 4. 5. 6. 7. 8. 9. 10. 11. 12. 13. 14. 15. 16. 17. 18. 19. 20. 21. 22. 23. 24. 25. 26. 27. 28. 29. 30. 31. 32. 33. 34. 35. 36. 37. 38. 39. 40. 41. 42. 43. 44. 45. 46. 47. 48. 49. 50. 51. 52. 53. 54. 55. 56. 57. 58. 59. 60. 61. 62. 63. 64. 65. 66. 67. 68. 69. 70. 71. 72. 73. 74. 75. 76. 77. 78. 79. 80. 81. 82. 83. 84. 85. 86. 87. 88. 89. 90. 91. 92. 93. 94. 95. 96. 97. 98. 99. 100.

标注属性

标注格式

值:115.5700

☐ 覆盖值

单位:mm

前缀:

单位格式:1234 mm

后缀:

精度:0.0000

层:User.Eco1

☐ 隐藏尾随零

预览:115.5700 mm

标注文本

宽度:1.524 mm

位置 X:137.16 mm

高度:2.032 mm

位置 Y:26.20772 mm

粗细:0.21082 mm

位置模式:外部

☐ 斜体

方向:0.0

☐ 镜像

☒ 与标注保持对齐

对齐:居中

标注线

线粗细:0.21082 mm

箭头长度:1.27 mm

尺寸界线偏移:0 mm

确定

取消

4.8.1

Figure 4-8-1: Dimension Properties dialog box. The dialog box is divided into three main sections: Dimension Format, Dimension Text, and Dimension Line. The Dimension Format section includes fields for Value (115.5700), Unit (mm), Prefix, Suffix, Layer (User.Eco1), and a checkbox for Hide trailing zeros. The Dimension Text section includes fields for Width (1.524 mm), Height (2.032 mm), Thickness (0.21082 mm), Position X (137.16 mm), Position Y (26.20772 mm), Position Mode (External), Direction (0.0), and checkboxes for Italic, Mirror, and Align with dimension (checked). The Dimension Line section includes fields for Line thickness (0.21082 mm), Arrow length (1.27 mm), and Dimension line offset (0 mm). The dialog box has 'OK' and 'Cancel' buttons at the bottom right.

4.8.2

Figure 4-8-2: Dimension Properties dialog box. The dialog box is divided into three main sections: Dimension Format, Dimension Text, and Dimension Line. The Dimension Format section includes fields for Value (115.5700), Unit (mm), Prefix, Suffix, Layer (User.Eco1), and a checkbox for Hide trailing zeros. The Dimension Text section includes fields for Width (1.524 mm), Height (2.032 mm), Thickness (0.21082 mm), Position X (137.16 mm), Position Y (26.20772 mm), Position Mode (External), Direction (0.0), and checkboxes for Italic, Mirror, and Align with dimension (checked). The Dimension Line section includes fields for Line thickness (0.21082 mm), Arrow length (1.27 mm), and Dimension line offset (0 mm). The dialog box has 'OK' and 'Cancel' buttons at the bottom right.

**Position mode:** Choose whether to position the dimension text manually, or to automatically keep it aligned with the dimension measurement lines.

Figure 4-8-3: Dimension Properties dialog box. The dialog box is divided into three main sections: Dimension Format, Dimension Text, and Dimension Line. The Dimension Format section includes fields for Value (115.5700), Unit (mm), Prefix, Suffix, Layer (User.Eco1), and a checkbox for Hide trailing zeros. The Dimension Text section includes fields for Width (1.524 mm), Height (2.032 mm), Thickness (0.21082 mm), Position X (137.16 mm), Position Y (26.20772 mm), Position Mode (External), Direction (0.0), and checkboxes for Italic, Mirror, and Align with dimension (checked). The Dimension Line section includes fields for Line thickness (0.21082 mm), Arrow length (1.27 mm), and Dimension line offset (0 mm). The dialog box has 'OK' and 'Cancel' buttons at the bottom right.

### 4.8.3 $\mathbb{R}^n$ as a vector space

&#x7ebf;&#x6761;&#x7c97;&#x7ec6;&#xff1a; &#x8bbe;&#x7f6e;&#x6784;&#x6210;&#x6807;&#x6ce8;&#x5f62;&#x72b6;&#x  
&#x6807;&#x6ce8;&#x754c;&#x7ebf;&#x504f;&#x79fb;&#xff1a; &#x8bbe;&#x7f6e;&#x6d4b;&#x91cf;&#x70b9;&#x5230;&#x  
&#x7bad;&#x5934;&#x957f;&#x5ea6;&#xff1a; &#x8bbe;&#x7f6e;&#x6807;&#x6ce8;&#x7684;&#x5f62;&#x72b6;&#x7684;&#x

#### 4.8.4 $\frac{1}{2} \times \frac{1}{2} \times \frac{1}{2} \times \frac{1}{2}$

引線格式

值:

文本框架: 圖形

层: User.Eco1

&#x503c;&#xff1a; &#x8f93;&#x5165;&#x8981;&#x5728;&#x5f15;&#x7ebf;&#x884c;&#x672b;&#x5c3e;&#x663e;&#x793a;&#x7  
&#x6587;&#x672c;&#x6846;&#x67b6;&#xff1a; &#x9009;&#x62e9;&#x6240;&#x9700;&#x7684;&#x6587;&#x672c;&#x5468;&#x

#### 4.9 $\times 10^3$ ; $\times 10^7$ ;

KiCad &#x177;&#x6709;&#x4ea4;&#x4e92;&#x5f0f;&#x5e03;&#x7ebf;&#x5668;&#x7684;&#x529f;&#x80fd;&#xff1a;

- &#x5141;&#x8bb8;&#x5bf9;&#x5355;&#x8f68;&#x548c;&#x5dee;&#x5206;&#x5bf9;&#x8fdb;&#x884c;&#x624b;&#x52a8;&#x534a;&#x81ea;&#x52a8;) &#x5e03;&#x7ebf;&#x3002;
- &#x53ef;&#x901a;&#x8fc7;&#x4ee5;&#x4e0b;&#x65b9;&#x5f0f;&#x4fee;&#x6539;&#x73b0;&#x6709;&#x8bbe;&#x8ba1;&#x5982;&#x679c;&#x9700;&#x8981;&#x7684;&#x8bdd;&#xff0c;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x4f7f;&#x7528;&#x9a00;&#x8fdd;&#x89c4;&#x9009;&#x9879;&#x6765;&#x7981;&#x7528;&#x8fd9;&#x79cd;&#x884c;&#x4e3a;&#xff08;&#x89c1;&#x5e03;&#x7ebf;&#x5668;&#x6709;&#x4e09;&#x79cd;&#x6a21;&#x5f0f;&#xff0c;&#x53ef;&#x4ee5;&#x968f;&#x65f6;&#x9000;&#x5e03;&#x7ebf;&#x5668;&#x6a21;&#x5f0f;&#x7528;&#x4e8e;&#x5e03;&#x7ebf;&#x65b0;&#x5e03;&#x7ebf;&#xff0c;&#x4f4bkd:[D]&#xff09;&#x547d;&#x4ee4;&#x62d6;&#x52a8;&#x73b0;&#x6709;&#x5e03;&#x7ebf;&#x65f6;&#x3002;&#x8fd9;&#x4e9b;
- &#x9ad8;&#x4eae;&#x51b2;&#x7a81;&#xff1a;&#x5728;&#x6b64;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x5927;&#x90e8;&#x5e03;&#x7ebf;&#x65f6;&#xff0c;&#x51b2;&#x7a81; (&#x95f4;&#x9699;&#x8fdd;&#x89c4;) &#x5c06;&#x4ee5;&#x7eff;&#x5982;&#x679c;&#x9700;&#x8981;&#x7684;&#x8bdd;&#xff0c;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x4f7f;&#x7528;&#x9a00;&#x8fdd;&#x89c4;&#x9009;&#x9879;&#x6765;&#x7981;&#x7528;&#x8fd9;&#x79cd;&#x884c;&#x4e3a;&#xff08;&#x89c1;&#x5e03;&#x7ebf;&#x5668;&#x6709;&#x4e09;&#x79cd;&#x6a21;&#x5f0f;&#xff0c;&#x53ef;&#x4ee5;&#x968f;&#x65f6;&#x9000;&#x5e03;&#x7ebf;&#x5668;&#x6a21;&#x5f0f;&#x7528;&#x4e8e;&#x5e03;&#x7ebf;&#x65b0;&#x5e03;&#x7ebf;&#xff0c;&#x4f4bkd:[D]&#xff09;&#x547d;&#x4ee4;&#x62d6;&#x52a8;&#x73b0;&#x6709;&#x5e03;&#x7ebf;&#x65f6;&#x3002;&#x8fd9;&#x4e9b;
- &#x63a8;&#x6324;&#xff1a;&#x5728;&#x6b64;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x88ab;&#x5e03;&#x7ebf;&#x7684;&#x4f8b;&#x5982;&#xff0c;&#x710a;&#x76d8;&#x548c;&#x9501;&#x5b9a;&#x7684;&#x5e03;&#x7ebf;/&#x5b54;&#x5b54;)&#x63a8;&#x6324;&#x969c;&#x788d;&#xff0c;&#x8fd9;&#x4e9b;&#x969c;&#x788d;&#x7269;&#x53ef;&#x4ee5;&#x79fb;&#x5982;&#x679c;&#x65e0;&#x6cd5;&#x5e03;&#x7ebf;&#x5230;&#x4e0d;&#x8fdd;&#x53cd; DRC &#x7684;&#x4e9b;
- &#x7ed5;&#x8d70;&#xff1a;&#x5728;&#x6b64;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x5e03;&#x7ebf;&#x5668;&#x7684;&#x4e9b;

&#x4f7f;&#x7528;&#x54ea;&#x79cd;&#x6a21;&#x5f0f;&#x662f;&#x4e00;&#x4e2a;&#x504f;&#x597d;&#x95ee;&#x9898;&#x3002;&#x5bf9;&#x4e8e;&#x5927;&#x591a;&#x6570;&#x7528;&#x6237;&#xff0c;&#x6211;&#x4eec;&#x5efa;&#x8bae;&#x4f7f;&#x7528;&#x45;&#x5ea6; (H/V/45) &#x5e03;&#x7ebf;&#x6bb5;&#x3002;&#x5982;&#x679c;&#x9700;&#x8981;&#x4f7f;&#x7528; H/V/45 &#x4ee5;&#x5916;&#x7684;&#x89d2;&#x5ea6;&#x5e03;&#x7ebf;&#x6bb5;&#xff0c;&#x5219;&#x5fc5;&#x987b;&#x4f7f;&#x7528;&#x6709;&#x4e94;&#x4e2a;&#x4e3b;&#x8981;&#x7684;&#x5e03;&#x7ebf;&#x529f;&#x80fd;&#x3002;&#x5355;&#x8f68;&#x504f; (&#x76f8;&#x4fd4;) &#x3002; &#x6240;&#x6709;&#x8fd9;&#x4e9b;&#x90fd;&#x5b58;&#x5728;&#x4e8e;&#x9876;&#x90e8;&#x504f; &#x4e0a;&#x9762;&#x4ecb;&#x7ecd;&#x4e86;&#x91cd;&#x8f7d;&#x56fe;&#x6807;&#x7684;&#x4f7f;&#x7528;&#x3002; &#x4e00;&#x4e2a;&#x7528;&#x4e8e;&#x4e24;&#x4e2a;&#x5e03;&#x7ebf;&#x529f;&#x80fd;&#xff0c;&#x4e00;&#x4e2a;&#x7528;&#x6b64;&#x5916;&#xff0c;&#x5e03;&#x7ebf;&#x83dc;&#x5355;&#x5141;&#x8bb8;&#x9009;&#x62e9;&#x8bbe;&#x7f6e;&#x504f; &#x8981;&#x5e03;&#x7ebf;&#xff0c;&#x8bf7;&#x70b9;&#x51fb;&#x5e03;&#x7ebf; &#x56fe;&#x6807;&#xff08;&#x4ece;&#x504f; &#x504f; &#x7ebf; &#x4e0b;&#xff09;&#x6216;&#x4f7f;&#x7528;&#x70ed;&#x952e; kbd:[X]&#x3002; &#x70b9;&#x51fb;&#x4e0a;&#x88ab;&#x5e03;&#x7ebf;&#x7684;&#x7f51;&#x7edc;&#x4f1a;&#x81ea;&#x52a8;&#x9ad8;&#x4eae;&#x663e;&#x793a;&#xff0c; &#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x6539;&#x53d8; " &#x504f;&#x597d;&#x8bbe;&#x7f6e;" &#x5bf9;&#x8bdd;&#x6846;&#x504f; " &#x95f4;&#x9699;&#x8f6e;&#x5ed3;" &#x8bbe;&#x7f6e;&#x6765;&#x7981;&#x7528;&#x95f4;&#x9699;&#x8f6e;&#x5ed3;&#x504f;

**Note**

&#x95f4;&#x9699;&#x8f6e;&#x5ed3;&#x663e;&#x793a;&#x4ece;&#x5e03;&#x7ebf;&#x7f51;&#x7edc;&#x5230; PCB  
 &#x4e0a;&#x4efb;&#x4f55;&#x5176;&#x4ed6;&#x94dc;&#x7ebf;&#x7684;&#x6700;&#x5927;&#x95f4;&#x9699;&#x3002;  
 &#x53ef;&#x4ee5;&#x4f7f;&#x7528;&#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x4e3a;&#x4e0d;&#x540c;&#x5e03;&#x7ebf;&#x5668;&#x5c06;&#x8003;&#x8651;&#x8fd9;&#x4e9b;&#x95f4;&#x9699;&#xff0c;&#x4f46;&#x4ec5;&#x76f4;&#x8f6e;&#x504f;

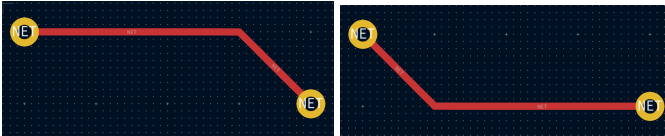
&#x5f53;&#x5e03;&#x7ebf;&#x5668;&#x5904;&#x4e8e;&#x6d3b;&#x52a8;&#x72b6;&#x6001;&#x65f6;&#xff0c;&#x5c06;&#x4e00; (unfixed temporary) &#x5bf9;&#x8c61;&#xff0c;&#x5b83;&#x4eec;&#x663e;&#x793a;&#x5f53;&#x60a8;&#x4f7f;&#x7528;&#x504f; kbd:[Enter] &#x952e;&#x6765;&#x56fa;&#x5b9a;&#x5e03;&#x7ebf; (fix the route) &#x65f6;&#x5c06;&#x521b;&#x5efa;&#x54ea; kbd:[Esc] &#x952e;&#x6216;&#x901a;&#x8fc7;&#x9009;&#x62e9;&#x53e6;&#x4e00;&#x4e2a;&#x5de5;&#x5177;&#x9000;&#x504f; kbd:[End]) &#x5c06;&#x56fa;&#x5b9a;&#x6240;&#x6709;&#x5e03;&#x7ebf;&#x5e76;&#x9000;&#x51fa;&#x5e03;&#x7ebf;&#x7528;&#x5e03;&#x7ebf;&#x65f6;&#xff0c;&#x53ef;&#x4ee5;&#x4f7f;&#x7528;&#x64a4;&#x6d88;&#x6700;&#x540e;&#x4e00; (&#x70ed;&#x952e; kbd:[Backspace]) &#x53d6;&#x6d88;&#x56fa;&#x5b9a;&#x6700;&#x8fd1;&#x56fa;&#x5b9a;&#x7684;&#x5e03;&#x7528;&#x4ee5;&#x524d;&#x7684; KiCad &#x7248;&#x672c;&#x4e2d;&#xff0c;&#x4f7f;&#x7528;&#x9f20;&#x6807;&#x5def; kbd:[Enter] &#x6765;&#x56fa;&#x5b9a;&#x5df2;&#x5e03;&#x7ebf;&#x7684;&#x7ebf;&#x6bb5;&#x4f1a;&#x56fa;&#x5b9a;&#x6700; &#x5728; KiCad 6 &#x4e2d;&#xff0c;&#x8fd9;&#x79cd;&#x884c;&#x4e3a;&#x73b0;&#x5728;&#x662f;&#x53ef;&#x9009;&#x7684; &#x5305;&#x62ec; &#x5728;&#x9f20;&#x6807;&#x5149;&#x6807;&#x4f4d;&#x7f6e;&#x7ed3;&#x675f;&#x7684;&#x7ebf;&#x6700; &#x901a;&#x8fc7;&#x5728;&#x4ea4;&#x4e92;&#x5f0f;&#x5e03;&#x7ebf;&#x5668;&#x8bbe;&#x7f6e;&#x5bf9;&#x8bdd;&#x6807; " &#x70b9;&#x51fb;&#x540e;&#x56fa;&#x5b9a;&#x6240;&#x6709;&#x7ebf;&#x6bb5;" &#x9009;&#x9879;&#xff0c;&#x53ef;&#x504f; &#x5e03;&#x7ebf;&#x65f6;&#xff0c;&#x53ef;&#x4ee5;&#x6309;&#x4f4f; kbd:[Ctrl] &#x952e;&#x7981;&#x7528;&#x7f51;&#x6807; kbd:[Shift] &#x952e;&#x7981;&#x7528;&#x5bf9;&#x710a;&#x76d8;&#x548c;&#x8fc7;&#x5b54;&#x7b49;&#x5bf9;&#x8c61;&#x504f;

**Note**

&#x4e5f;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x66f4;&#x6539;&#x9996;&#x9009;&#x9879;&#x5bf9;&#x8bdd;&#x6846;&#x7684;&#x504f; &#x6211;&#x4eec;&#x5efa;&#x8bae;&#x60a8;&#x5728;&#x4e00;&#x822c;&#x60c5;&#x51b5;&#x4e0b;&#x4fdd;&#x6301;&#x542f;&#x504f;

**4.9.1 &#x5e03;&#x7ebf;&#x5f62;&#x6001;**

&#x5728;&#x5782;&#x76f4;(H) / &#x6c34;&#x5e73; (V) / 45 &#x5ea6;&#x6a21;&#x5f0f;&#x4e0b;&#x5e03;&#x7ebf;&#x65f6;&#x662f;&#x6307;&#x4e00;&#x7ec4;&#x4e24;&#x4e2a;&#x7ebf;&#x6bb5;&#x5982;&#x4f55;&#x8fde;&#x63a5;&#x5355;&#x4e00; H/V/45 &#x5ea6;&#x7ebf;&#x6bb5;&#x65e0;&#x6cd5;&#x5230;&#x8fbe;&#x7684;&#x4e24;&#x4e2a;&#x70b9;&#x3002; &#x5728;&#x8fd9;&#x79cd;&#x60c5;&#x51b5;&#x4e0b;&#xff0c;&#x8fd9;&#x4e9b;&#x70b9;&#x5c06;&#x7531;&#x4e00;&#x6700; (45&#x5ea6;) &#x7ebf;&#x6bb5;&#x8fde;&#x63a5;&#x3002;&#x5f62;&#x6001;&#x6307;&#x7684;&#x662f;&#x8fd9;&#x4e9b;&#x504f;



KiCad &#x7684;&#x5e03;&#x7ebf;&#x5668;&#x8bd5;&#x56fe;&#x6839;&#x636e;&#x4e00;&#x7cfb;&#x5217;&#x56e0;&#x7d2d  
" &#x574f;" &#x62d0;&#x89d2; (&#x5982;&#x9510;&#x89d2;)&#x3002;&#x5f53;&#x4ece;&#x710a;&#x76d8;&#x5e03;&#x7ebf;&#x5c06;&#x9009;&#x62e9;&#x5c06;&#x8def;&#x7ebf;&#x4e0e;&#x710a;&#x76d8;&#x6700;&#x957f;&#x8fb9;&#x7f18;&#x5b  
&#x5728;&#x67d0;&#x4e9b;&#x60c5;&#x51b5;&#x4e0b;&#xff0c;KiCad &#x65e0;&#x6cd5;&#x6b63;&#x786e;&#x731c;&#x6d4  
(&#x70ed;&#x952e; kbd:[/])&#x3002;

&#x5728;&#x6c11;&#x6709;&#x660e;&#x663e;&#x7684;“&#x6700;&#x4f73;”&#x5f62;&#x6001;&#x7684;&#x60c5;&#x51b5;&#x5c06;&#x4f7f;&#x7528;&#x9f20;&#x6807;&#x5149;&#x6807;&#x7684;&#x79fb;&#x52a8;&#x6765;&#x9009;&#x62e9;&#x5546;&#x6c34;&#x5e73;&#x6216;&#x5782;&#x76f4;)&#x7ebf;&#x6bb5;&#x5f00;&#x59cb;&#xff0c;&#x8bf7;&#x5728;&#x6c34;&#x

### Note

&#x5982;&#x679c;&#x4f7f;&#x7528;&#x5207;&#x6362;&#x5e03;&#x7ebf;&#x5f62;&#x6001;&#x547d;&#x4ee4;&#x8986;&#x76d6;  
KiCad &#x9009;&#x62e9;&#x7684;&#x5f62;&#x6001;&#xff0c;&#x5219;&#x5728;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x64cd;&#x4f

#### 4.9.2 $\frac{d}{dt} \left( \frac{\partial L}{\partial \dot{x}} \right) = \frac{\partial L}{\partial x}$

&#x5f53;&#x4ee5; H/V/45 &#xa21;&#x5f0f;&#x5e03;&#x7ebf;&#x65f6;&#xff0c;KiCad &#x7684;&#x5e03;&#x7ebf;&#x5668;&#x70ed;&#x952e; kbd:[Ctrl+/]&#x3002;&#x4f7f;&#x7528;&#x5706;&#x89d2;&#x5e03;&#x7ebf;&#x65f6;&#xff0c;&#x6bcf;&#x9009;&#x62e9;&#x6240;&#x9700;&#x5e03;&#x7ebf;&#x540e;&#xff0c;&#x8fd8;&#x53ef;&#x4ee5;&#x5728;&#x5e03;&#x7e

### Note

&#x5c1a;&#x4e0d;&#x652f;&#x6301;&#x4f7f;&#x7528;&#x5706;&#x5f27;&#x62d6;&#x52a8;&#x5e03;&#x7ebf;&#x3002;&#x5f53;&#x

### 4.9.3 $\mathbb{R}^n$ and $\mathbb{C}^n$

&#x88ab;&#xe03;&#x7ebf;&#x7684;&#x7ebf;&#x6bb5;&#x7684;&#x5bbd;&#xea6;&#x662f;&#x901a;&#x8fc7;&#xae5;&#x4e3a;  
= &#x6309;&#x94ae;&#x88ab;&#x542f;&#x7528;&#xff0c;&#x5bbd;&#xea6;&#xc06;&#x88ab;&#x8bbe;&#x7f6e;&#x4e3a;  
&#x5426;&#x5219;&#xff0c;&#x5982;&#x679c;&#x9876;&#x90e8;&#x5de5;&#x5177;&#x680f;&#x4e2d;&#x7684;&#xe03;&#x7684;&#xf7f;  
"&#x4f7f;&#x7528;&#x7f51;&#x7edc;&#x7c7b;&#x5bbd;&#xea6;"&#xff0c;&#x5219;&#x5bbd;&#xea6;&#xc06;&#x53d6;&#x7684;&#x6700;&#x540e;&#xff0c;&#x5982;&#x679c;&#xe03;&#x7ebf;&#x5bbd;&#xea6;&#x4e0b;&#x62c9;&#x83dc;&#x5355;&#x88ab;

### Note

&#x5e03;&#x7ebf;&#x5bbd;&#x5ea6;&#x6c38;&#x8fdc;&#x4e0d;&#x80fd;&#x4f4e;&#x4e8e;&#x5728;&#x7535;&#x8def;&#x677f;&#x

K;iCad &#x7684;&#x5e03;&#x7ebf;&#x5668;&#x652f;&#x6301;&#x6d3b;&#x52a8;&#x8def;&#x7ebf;&#x7684;&#x5355;&#x4e00  
&#x6362;&#x53e5;&#x8bdd;&#x8bf4;&#xff0c;&#x8981;&#x5728;&#x7ebf;&#x6bb5;&#x4e2d;&#x95f4;&#x6539;&#x53d8;&#x55  
&#x8981;&#x6539;&#x53d8;&#x6d3b;&#x52a8;&#x7ebf;&#x6bb5;&#x7684;&#x5bbd;&#x5ea6;&#xff0c;&#x53ef;&#x4f7f;&#x755  
kbd:[W] &#x548c; kbd:[Shift+W]&#xff0c;&#x5728;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#x5bf9;&#x8bdd;&#x6846;&#x



#### 4.9.4

&#x5728;&#x5e03;&#x7ebf;&#x7ebf;&#x6bb5;&#x65f6;&#xff0c;&#x5207;&#x6362;&#x5c42;&#x4f1a;&#x5728;&#x5f53;&#x52  
&#x4e00;&#x65e6;&#x4f60;&#x653e;&#x7f6e;&#x4e86;&#x8fc7;&#x5b54;&#xff0c;&#xe03;&#x7ebf;&#x5c06;&#x7ee7;&#x7ee  
&#x6709;&#x51e0;&#x79cd;&#x65b9;&#x6cd5;&#x53ef;&#x4ee5;&#x9009;&#x62e9;&#x4e00;&#x4e2a;&#x65b0;&#x5c42;&#x5

- &#x4f7f;&#x7528;&#x70ed;&#x952e;&#x9009;&#x62e9;&#x7279;&#x5b9a;&#x7684;&#x56fe;&#x5c42;&#xff0c;&#x5982;  
kbd:[PgUp] &#x9009;&#x62e9; F . Cu &#x6216; kbd:[PgDn] &#x9009;&#x62e9; B . Cu&#x3002;
- &#x901a;&#x8fc7;&#x4f7f;&#x7528; "&#x4e0b;&#x4e00;&#x5c42;" &#x6216; "&#x4e0a;&#x4e00;&#x5c42;" &#x70ed;&#x952e;  
(kbd:[+] &#x548c; kbd:[-])&#x3002;
- &#x901a;&#x8fc7;&#x4f7f;&#x7528; "&#x653e;&#x7f6e;&#x8fc7;&#x5b54;" &#x70ed;&#x952e; (kbd:[V])&#xff0c;&#x5b83;&#x3002;
- &#x901a;&#x8fc7;&#x4f7f;&#x7528; "&#x9009;&#x62e9;&#x56fe;&#x5c42;&#x5e76;&#x901a;&#x8fc7;&#x901a;&#x5b54;&#x64cd;&#x4f5c; (&#x70ed;&#x952e; kbd:[<])&#xff0c;&#x5c06;&#x6253;&#x5f00;&#x4e00;&#x4e2a;&#x5bf9;&#x8bdd;&#x3002;

&#x8fc7;&#x5b54;&#x7684;&#x5c3a;&#x5bf8;&#x5c06;&#x4ece;&#x6fc0;&#x6d3b;&#x7684;&#x8fc7;&#x5b54;&#x5c3a;&#x5b54;(kbd:[''])&#x548c;&#x51cf;&#x5c0f;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;(kbd:[\ ])&#x70ed;&#x952e;&#x8bbf;&#x95ee;&#x300&#x65f6;&#xff0c;&#x5c06;&#x4f7f,&#x7528;"&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;"&#x7684;"&#x7f51;&#x7edc;&#x&#x90e8;&#x5206;&#x4e2d;&#x914d;&#x7f6e;&#x7684;&#x8fc7;&#x5b54;&#x5927;&#x5c0f;(&#x9664;&#x975e;&#x88ab;&#x&#x5982;&#x679c;&#x5728;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#x5bf9;&#x8bdd;&#x6846;&#x7684;&#x7ea6;&#x6&#x4f7f;&#x7528;&#x70ed;&#x952e;kbd:[Ctrl+V]&#x6765;&#x653e;&#x7f6e;&#x5fae;&#x5b54;&#xff0c;kbd:[Alt+Shift+V]  
&#x6765;&#x653e;&#x7f6e;&#x76f2;/&#x57cb;&#x5b54;&#x3002;&#x5fae;&#x5b54;&#x53ea;&#x80fd;&#x88ab;&#x653e;&#x7&#x76f2;&#x5b54;/&#x57cb;&#x5b54;&#x53ef;&#x4ee5;&#x653e;&#x7f6e;&#x5728;&#x4efb;&#x4f55;&#x4e00;&#x5c42;&#x4  
&#x5e03;&#x7ebf;&#x5668;&#x653e;&#x7f6e;&#x7684;&#x8fc7;&#x5b54;&#x88ab;&#x8ba4;&#x4e3a;&#x662f;&#x5df2;&#x5e  
&#x8fd9;&#x610f;&#x5473;&#x7740;&#x8fc7;&#x5b54;&#x7f51;&#x7edc;&#x53ef;&#x4ee5;&#x81ea;&#x52a8;&#x66f4;&#x65  
PCB&#x65f6;&#x6539;&#x53d8;&#x4e86;&#x7ebf;&#x6bb5;&#x7684;&#x7f51;&#x7edc;&#x540d;&#x3002;&#x5728;&#x67d0  
&#x5bf9;&#x4e8e;&#x7279;&#x5b9a;&#x7684;&#x8fc7;&#x5b54;&#xff0c;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x5173;&#x95  
"&#x81ea;&#x52a8;&#x66f4;&#x65b0;&#x8fc7;&#x5b54;&#x7f51;&#x7edc;"&#x590d;&#x9009;&#x6846;&#x6765;&#x7981;&#  
&#x4f7f;&#x7528;"&#x6dfb;&#x52a0;&#x72ec;&#x7acb;&#x8fc7;&#x5b54:"&#x5de5;&#x5177;&#x653e;&#x7f6e;&#x7684;&#x

#### 4.9.5 &#x5dee;&#x5206;&#x5bf9;&#x5e03;&#x7ebf;

KiCad &#x4e2d;&#x7684;&#x5dee;&#x5206;&#x5bf9;&#x88ab;&#x5b9a;&#x4e49;&#x4e3a;&#x5177;&#x6709;&#x5171;&#x540  
&#x57fa;&#x6570;&#x540d;&#x79f0;&#x548c;&#x6b63;&#x8d1f;&#x540e;&#x7f00;&#x7684;&#x7f51;&#x8def;&#x3002;  
KiCad &#x652f;&#x6301;&#x4f7f;&#x7528; + &#x548c; -&#xff0c;&#x6216;&#x8005; P &#x548c; N &#x4f5c;&#x4e3a;&#x540e;  
&#x4f8b;&#x5982;&#xff0c;USB+ &#x548c; USB- &#x6784;&#x6210;&#x4e00;&#x5bf9;&#x5dee;&#x5206;&#xff0c;USB\_P  
&#x548c; USB\_N &#x4e5f;&#x662f;&#x5982;&#x6b64;&#x3002; &#x5728;&#x7b2c;&#x4e00;&#x4e2a;&#x4f8b;&#x5b50;&#x4e  
USB&#xff0c;&#x7b2c;&#x4e8c;&#x4e2a;&#x4f8b;&#x5b50;&#x4e2d;&#x662f; USB\_&#x3002;&#x540e;&#x7f00;&#x6837;&#x5  
USB+ &#x548c; USB\_N &#x4e0d;&#x6784;&#x6210;&#x5dee;&#x5206;&#x5bf9;&#x3002; &#x8bf7;&#x786e;&#x4fdd;&#x4f60;  
PCB &#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x4f7f;&#x7528;&#x5dee;&#x5206;&#x5bf9;&#x5e03;&#x7ebf;&#x5668;&#x3002;  
&#x8981;&#x5bf9;&#x5dee;&#x5206;&#x5bf9;&#x8fdb;&#x884c;&#x5e03;&#x7ebf;&#xff0c;&#x8bf7;&#x70b9;&#x51fb;&#x5e0



&#x56fe;&#x6807;&#xff08;&#x4ece;&#x7ed8;&#x56fe;&#x5de5;&#x5177;&#x680f;&#x6216;&#x4ece;&#x9876;&#x90e8;  
&#x5e03;&#x7ebf;&#x4e0b;&#xff09;&#x6216;&#x4f7f;&#x7528;&#x70ed;&#x952e; kbd:[6]&#x3002;&#x70b9;&#x51fb;&#x4e  
&#x4f60;&#x53ef;&#x4ee5.&#x4ece;&#x5dee;&#x5206;&#x5bf9;&#x7684;&#x6b63;&#x7f51.&#x6216;&#x8d1f;&#x7f51.&#x5f

## Note

&#x76ee:&#x524d:&#x4e0d:&#x53ef:&#x80fd:&#x5728:&#x73b0:&#x6709:&#x5dee:&#x5206:&#x5bf9:&#x5e03:&#x7ebf:&#x7684:&#

&#x5dee;&#x5206;&#x5bf9;&#x5e03;&#x7ebf;&#x5668;&#x5c06;&#x5c1d;&#x8bd5;&#x7528;&#x8bbe;&#x8ba1;&#x89c4;&#x5  
" &#x6247;&#x51fa;" &#x90e8;&#x5206;&#xff0c;&#x4ee5;&#x6700;&#x5927;&#x9650;&#x5ea6;&#x5730;&#x7f29;&#x77ed;&#x  
&#x5f53;&#x4ea4;&#x6362;&#x5c42;&#x6216;&#x4f7f;&#x7528;&#x653e;&#x7f6e;&#x8fc7;&#x5b54;(kbd:[V])&#x64cd;&#x4







4.9.8 调整单轨布线长度

Note

调整单轨布线长度时，应确保目标长度大于等于最小波幅（Amin）。

4.9.8 调整单轨布线长度

调整单轨布线长度时，应确保目标长度大于等于最小波幅（Amin）。



|  |   |
|--|---|
| &#x8bbe;&#x7f6e;&#x6a21;&#x5f0f;                         | &#x8bf4;&#x660e;&#x8bbe;&#x7f6e;&#x7528;&#x4e8e;&#x521b;&#x5efa;&#x65b0;&#x5e03;&#x7ebf;&#x548c;&#x6709;&#x5173;&#x8be6;&#x7ec6;&#x4fe1;&#x606f;&#xff0c;&#x8bf7;&#x53c2;&#x9605;&#x81ea;&#x7531;&#x89d2;&#x5f0f;&#x41;&#x6a21;&#x7f0f;&#x4ee5;&#x4efb;&#x4f55;&#x89d2;&#x5ea6;&#x5e03;&#x7ebf;&#xff0c;&#x45&#x5ea6;&#x589e;&#x91cf;&#x5e03;&#x7ebf;&#x3002;&#x4ec5;&#x5f53;&#x5e03;&#x7ebf;&#x6a21;&#x5f0f;&#x8bbe;&#x7f6e;&#x4e3a;&#x7a81;&#x6b64;&#x9009;&#x9879;&#x624d;&#x53ef;&#x7528;&#x3002; |
| &#x7ed5;&#x8fc7;&#x969c;&#x7857;&#x7269;                 | &#x8572;&#x7269;&#x6324;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x5141;&#x8bb8;&#x5e03;&#x5982;&#x710a;&#x76d8;)&#x540e;&#x9762;&#x79fb;&#x52a8;&#x78b0;&#x649e;&#x5e03;&#x7ebf;&#x3002;   |
| &#x79fb;&#x9664;&#x591a;&#x4f59;&#x7652;&#x7b05;&#x76b6; | &#x5278;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x4e2d;&#x4ec5;&#x4fdd;&#x7559;&#x56de;&#x8def;&#x4e2d;&#x6700;&#x8fd1;&#x5e03;&#x7ebf;&#x4f18;&#x5316;&#x710a;&#x76d8;&#x7b05;&#x6b64;&#x8bbe;&#x7f6e;&#x65f6;&#xff0c;&#x4ea4;&#x4e92;&#x5f0f;&#x548c;&#x5176;&#x4ed6;&#x4e0d;&#x9700;&#x8981;&#x7684;&#x5e03;&#x7ebf;&#x3002;  |
| &#x5e73;&#x6ed1;&#x62d6;&#x7269;&#x7b05;&#x7b05;         | &#x5e03;&#x7ebf;&#x65f6;&#xff0c;&#x4f1a;&#x5c1d;&#x8bd5;&#x5c06;&#x4ee5;&#x6700;&#x5927;&#x9650;&#x5ea6;&#x5730;&#x51cf;&#x5c11;&#x65b9;&#x5411;  |

|  |   |
|--|---|
| &#x8bbe;&#x7f6e;<br>&#x5141;&#x8bb8;&#x8fdd;&#x5728;&#x9ad8;&#x4eae;&#x78b0;&#x649e;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x5141;&#x5219;<br>DRC &#x89c4;&#x5219; | &#x8bf4;&#x660e;<br>&#x5728;&#x9ad8;&#x4eae;&#x78b0;&#x649e;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x5141;&#x5219;<br>DRC<br>&#x89c4;&#x5219;&#x7684;&#x5e03;&#x7ebf;&#x548c;&#x8fc7;&#x5b54;&#x3002;<br>&#x5728;&#x5176;&#x4ed6;&#x6a21;&#x5f0f;&#x4e0b;&#x4e0d;&#x8d77;&#x4f5c;&#x7528;   |
| &#x4f18;&#x5316;&#x6b63;&#x5728;&#x52a8;&#x5e03;&#x7ebf;&#x6bb5;&#x5c06;   | KiCad<br>&#x4f18;&#x5316;&#x5c4f;&#x5e55;&#x4e0a;&#x53ef;&#x89c1;&#x7684;&#x5176;&#x4f59;<br>&#x4f18;&#x5316;&#x8fc7;&#x7a0b;&#x53bb;&#x9664;&#x4e86;&#x4e0d;&#x5fc5;&#x8981;<br>&#x7981;&#x7528;&#x65f6;&#xff0c;&#x4e0d;&#x4f1a;&#x5bf9;&#x6b63;&#x5728;&#x62d6;<br>&#x5728;&#x62d6;&#x52a8;&#x5e03;&#x7ebf;&#x65f6;&#x5c1d;&#x8bd5;&#x4f18;&#x5316;   |
| &#x4f7f;&#x7528;&#x9f20;&#x6807;&#x4f4d;&#x7f6e;&#x5f00;&#x59cb;&#x4f4d;&#x7f6e;&#x5f00;   | &#x6807;&#x4f4d;&#x7f6e;&#x5f00;&#x59cb;&#x4f4d;&#x7f6e;&#x5f00;&#x59cb;&#x4f4d;&#x7f6e;&#x5f00;<br>&#x5982;&#x679c;&#x9f20;&#x6807;&#x4ece;&#x5f00;&#x59cb;&#x4f4d;&#x7f6e;&#x5f00;<br>&#x5982;&#x679c;&#x9f20;&#x6807;&#x4e3b;&#x8981;&#x6c34;&#x5e73;&#x6216;&#x5782;<br>&#x5f53;&#x9f20;&#x6807;&#x79bb;&#x5f00;&#x5e03;&#x7ebf;&#x8d77;&#x59cb;&#x4f4d;<br>&#x5e76;&#x4e14;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x79fb;&#x56de;&#x8d77;&#x59cb;  |
| &#x70b9;&#x51fb;&#x65f6;&#x56f4;&#x5b9a;&#x62a8;&#x9009;&#x5728;&#x6b55;   | &#x56f4;&#x5b9a;&#x62a8;&#x9009;&#x5728;&#x6b55;&#x7ebf;&#x65f6;&#x5355;&#x51fb;&#x5305;&#x62ec;&#x5728;&#x9f20;&#x6807;&#x5149;&#x6807;&#x7ed3;&#x675f;&#x7684;<br>&#x7981;&#x7528;&#x65f6;&#xff0c;&#x6700;&#x540e;&#x4e00;&#x4e2a;&#x7ebf;&#x6bb5;<br>(&#x5728;&#x9f20;&#x6807;&#x5149;&#x6807;&#x5904;&#x7ed3;&#x675f;&#x7684;&#x7ebf;&#x5c06;&#x4e0d;&#x4f1a;&#x56fa;&#x5b9a;&#x5728;&#x9002;&#x5f53;&#x4f4d;&#x7f6e;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x8fdb;&#x4e00;&#x6b65;&#x7684;&#x9f20;&#x6807; |

## 4.10 &#x5411;&#x524d;&#x548c;&#x5411;&#x540e;&#x6279;&#x6ce8;

### Note

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;

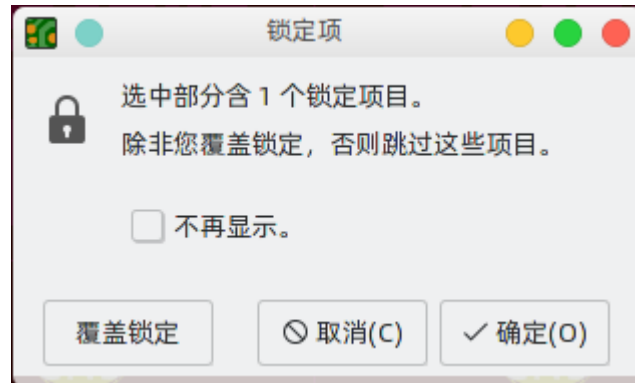
### 4.10.1 &#x4f4d;&#x7f6e;&#x91cd;&#x65b0;&#x6279;&#x6ce8;

### Note

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;

## 4.11 &#x9501;&#x5b9a;

&#x5927;&#x591a;&#x6570;&#x5bf9;&#x8c61;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x5176;&#x5c5e;&#x6027;&#x5bf9;&#x8104;  
" &#x5207;&#x6362;&#x9501;&#x5b9a;" &#x70ed;&#x952e;&#xff08[&#x5b9a]&#x5b9a;&#x3002;  
&#x88ab;&#x9501;&#x5b9a;&#x7684;&#x5bf9;&#x8c61;&#x4e0d;&#x80fd;&#x88ab;&#x9009;&#x62e9;&#xff0c;&#x9664;&#x9009;  
" &#x88ab;&#x9501;&#x5b9a;&#x7684;&#x9879;&#x76ee;" &#x590d;&#x9009;&#x6846;&#x88ab;&#x542f;&#x7528;&#x3002;  
&#x8bd5;&#x56fe;&#x79fb;&#x52a8;&#x9501;&#x5b9a;&#x7684;&#x9879;&#x76ee;&#x5c06;&#x5bfc;&#x81f4;&#x4e00;&#x4e00;



&#x5728;&#x8fd9;&#x4e2a;&#x5bfb9;&#x8bdd;&#x6846;&#x4e2d;&#x9009;&#x62e9;"&#x8986;&#x76d6;&#x9501;&#x5b9a:"&#x5c06;&#x5141;&#x8bb8;&#x79fb;&#x52a8;&#x9501;&#x5b9a;&#x7684;&#x9879;&#x76ee;&#x3002;&#x9009;&#x62e9;"&#x786e;&#x5b9a;"&#x5c06;&#x5141;&#x8bb8;&#x4f60;&#x5728;&#x9009;&#x62e9;&#x4e2d;&#x79fb;&#x52a8;&#x4efb;&#x9009;&#x62e9;"&#x4e0d;&#x518d;&#x663e;&#x793a;"&#x5c06;&#x4f7f;&#x4f60;&#x5728;&#x5269;&#x4e0b;&#x7684;&#x9879;

#### 4.12 **批量编辑工具**

### Note

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;

#### 4.13 $\times 10^5$ ; $\times 10^6$ ; $\times 10^5$ ; $\times 10^7$ ;

### Note

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;

4.14 &#x6b63;&#x5728;&#x5bfc;&#x5165;&#x56fe;&#x5f62;

### Note

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;

#### 4.14.1 **丽 DXF &#x548c; SVG &#x6587;&#x4ef6;&#x5bfc;&#x165;&#x77e2;&#x91cf;&#x56fe;**

### Note

TODO&#xff1a;&#x5199;&#x4e0b;&#x8fd9;&#x4e00;&#x8282;

#### 4.14.2 **¶**

### Note

TODO&#xff1a;&#x5199:&#x4e0b:&#x8fd9:&#x4e00:&#x8282:

## Chapter 5

&#x68c0;&#x67e5;&#x7535;&#x8def;&#x677f;

## 5.1

&#x6d4b;&#x91cf;&#x5de5;&#x5177;&#x5141;&#x8bb8;&#x4f60;&#x5728; PCB &#x4e0a;&#x7684;&#x5404;&#x70b9;&#x4e4b  
&#x8981;&#x6fc0;&#x6d3b;&#x8be5;&#x5de5;&#x5177;&#xff0c;&#x8bf7;&#x70b9;&#x51fb;&#x53f3;&#x4fa7;&#x5de5;&#x51



&#x56fe;&#x6807;&#xff0c;&#x6216;&#x4f7f;&#x7528;&#x70ed;&#x952e; kbd:[Ctrl + Shift + M]&#x3002; &#x4e00;&#x65

### Note

&#x64b;&#x91cf;&#x5de5;&#x5177;&#x7528;&#x4e8e;&#x4e0d;&#x9700;&#x8981;&#x6c38;&#x4e45;&#x663e;&#x793a;&#x7684;&#x60a8;&#x6240;&#x505a;&#x7684;&#x4efb;&#x4f55;&#x6d4b;&#x91cf;&#x90fd;&#x5c06;&#x4ec5;&#x5728;&#x8be5;&#x5de5;&#x8981;&#x521b;&#x5efa;&#x5c06;&#x663e;&#x793a;&#x5728;&#x6253;&#x5370;&#x8f93;&#x51fa;&#x548c;&#x6253;&#x5370;&

## 5.2 $\times$ ba1;89c4;5219;68c0;67e5;

&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x68c0;&#x67e5;&#x5668;&#x7528;&#x4e8e;&#x9a8c;&#x8bc1; PCB &#x662f;&#x5426  
KiCad &#x53ef;&#x4ee5;&#x5728;&#x5e03;&#x7ebf;&#x7ebf;&#x6bb5;&#x65f6;&#x81ea;&#x52a8;&#x9632;&#x6b62;&#x4e00  
&#x8fd9;&#x610f;&#x5473;&#x7740;&#x5728;&#x4e3a; PCB &#x521b;&#x5efa;&#x5236;&#x9020;&#x6587;&#x4ef6;&#x4e4b;

&#x8981;&#x4f7f;&#x7528;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x68c0;&#x67e5;&#x5668;&#xff0c;&#x8bf7;&#x70b9;&#x51



&#x56fe;&#x6807;&#xff0c;&#x6216;&#x8005;&#x4ece;&#x68c0;&#x67e5;&#x83dc;&#x5355;&#x4e2d;&#x9009;&#x62e9



DRC 规则检查报告摘要：发现 3 个错误，1 个警告。错误详情如下：

- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)
- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)
- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)

PCB 规则检查详细报告：DRC 检查 PCB 和原理图是否一致。发现 3 个错误，1 个警告。错误详情如下：

- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)
- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)
- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)

警告：丝印与开窗区域相交。警告详情如下：

- 警告：丝印与开窗区域相交  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)

PCB 规则检查详细报告：DRC 检查 PCB 和原理图是否一致。发现 3 个错误，1 个警告。错误详情如下：

- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)
- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)
- 错误：与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)

警告：丝印与开窗区域相交。警告详情如下：

- 警告：丝印与开窗区域相交  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)



错误: 与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)  
错误: 与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)  
错误: 与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)

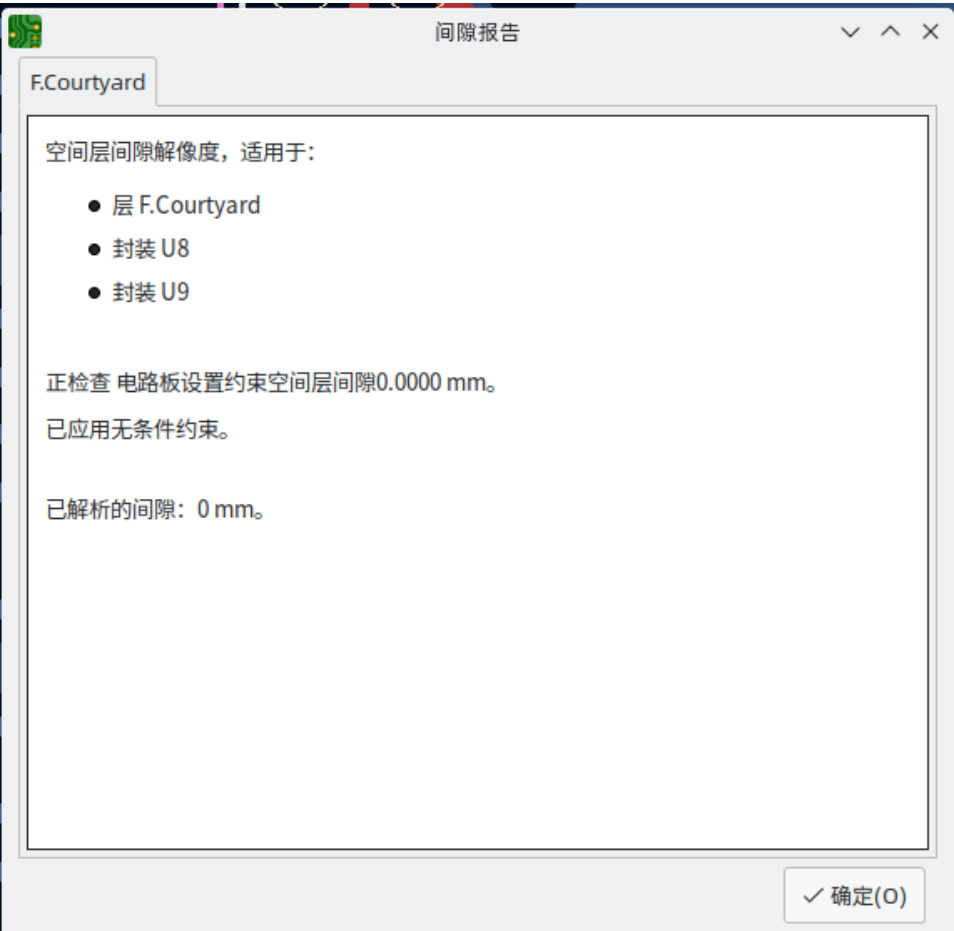
显示: ☐ 所有 ☒ 错误 3 ☐ 警告 0 ☐ 排除

5.2.1 错误: 与开窗区域相交的丝印

错误: 与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)  
错误: 与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)  
错误: 与开窗区域相交的丝印  
线 (在 Edge.Cuts 上)  
线 (在 F.Silkscreen 上)

显示: ☐ 所有 ☒ 错误 3 ☐ 警告 0 ☐ 排除



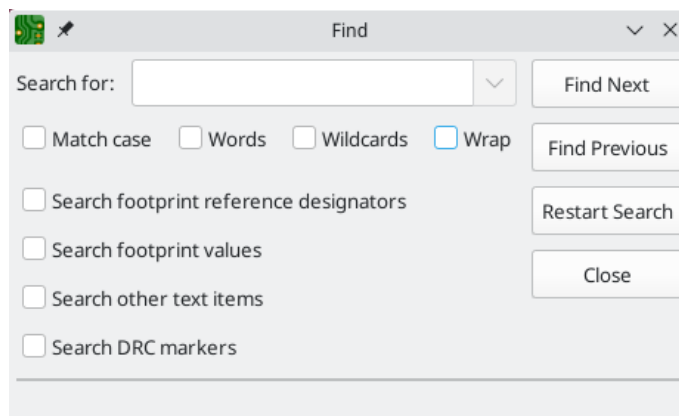


To inspect the design constraints that apply to an object, select it and choose Constraints Resolution from the Inspect menu. The Constraints Report dialog will show any constraints that apply to the object.



### 5.3 Find tool

The Find tool searches for text in the PCB, including reference designators, footprint fields, and graphic text. When the tool finds a match, the canvas is zoomed and centered on the match and the text is highlighted. Launch the tool using the (A) button in the top toolbar.



The Find tool has several options:

**Match case:** Selects whether the search is case-sensitive.

**Words:** When selected, the search will only match the search term with complete words in the PCB. When unselected, the search will match if the search term is part of a larger word in the PCB.

**Wildcards:** When selected, wildcards can be used in the search terms. ? matches any single character, and \\* matches any number of characters. Note that when this option is selected, partial matches are not returned: searching for abc\* will match the string abcd, but searching for abc will not.

**Wrap:** When selected, search results will return to the first hit after reaching the last hit.

**Search footprint reference designators:** Selects whether the search should apply to footprint reference designators.

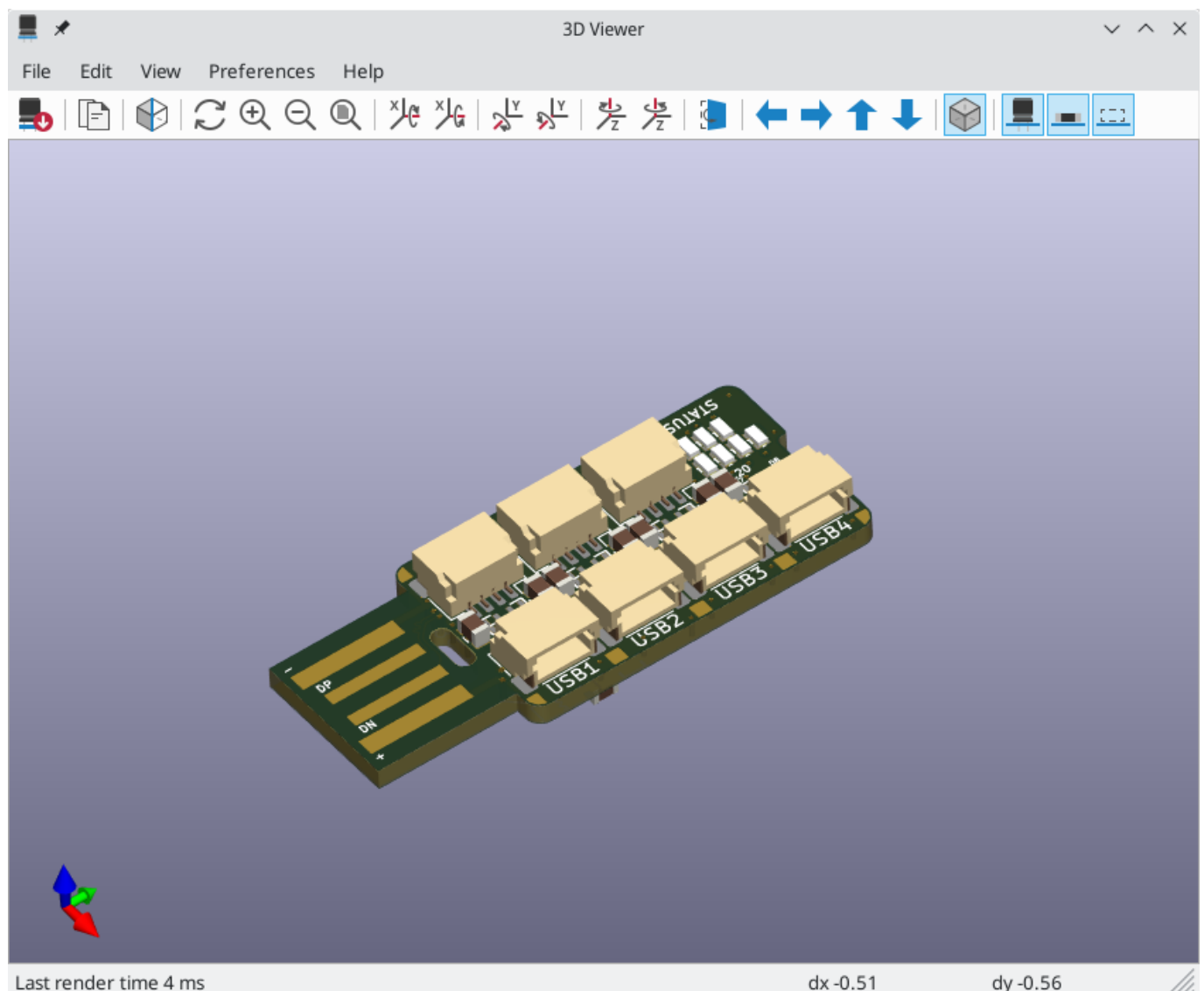
**Search footprint values:** Selects whether the search should apply to footprint value fields.

**Search other text items:** Selects whether the search should apply to other text items, including graphical text and footprint fields other than value and reference.

**Search DRC markers:** Selects whether the search should apply to the violation descriptions of DRC markers shown on the board.

## 5.4 3D &#x67e5;&#x770b;&#x5668;

The 3D Viewer shows a 3-dimensional view of the board and the components on the board. You can view the board from different perspectives, show or hide different types of components, cross-probe from the PCB Editor to the 3D viewer, and generate raytraced renders of the board.



### Note

The 3D model for a component will only appear if the 3D model file exists and has been [assigned to the footprint](#).

### Note

Many footprints in KiCad's standard library do not yet have model files created for them. However, these footprints may contain a path to a 3D model that does not yet exist, in anticipation of the 3D model being created in the future.


## 5.4.1 Navigating the 3D view

Dragging with the left mouse button will orbit the 3D view around the centroid of the board. Scrolling the mouse wheel will zoom the view in or out. Scrolling while holding kbd:[Ctrl] pans the view left and right, and scrolling while holding kbd:[Shift] pans up and down. Dragging with the middle mouse button also pans the view.


Different sized 3D grids can be set using the **Preferences** → **3D Grid** menu. Bounding boxes for each component can be enabled with **Preferences** → **Show Model Bounding Boxes**.

When the PCB Editor and the 3D Viewer are both open, selecting a footprint in the PCB Editor will also highlight the component in the 3D Viewer. The highlight color is adjustable in **Preferences** → **Preferences...** → **3D Viewer** → **Realtime Renderer** → **Selection Color**.

## 5.4.2 Generating images with the 3D Viewer

The current 3D view can be saved to an image with **File** → **Export Current View as PNG...** or **Export Current View as JPG...**, depending on the desired image format. The current view can also be copied to the clipboard using the  button, or **Edit** → **Copy 3D Image**.







The 3D Viewer has a raytracing rendering mode which displays the board using a more physically accurate rendering model than the default rendering mode. Raytracing is slower than the default rendering mode, but it can be used when the most visually

















attractive results are desired. Raytracing mode is enabled with the  button, or with **Preferences** → **Raytracing**. The 3D grid and selection highlights are not shown in raytracing mode.

Colors and other rendering options, for both raytraced and non-raytraced modes, can be adjusted in **Preferences** → **Preferences...** → **3D Viewer**.

## 5.4.3 3D viewer controls

Many viewing options are controlled with the top toolbar.

|   |                                      |
|---|--------------------------------------|
|  | Reload the 3D model                  |
|  | Copy 3D image to clipboard           |
|  | Render current view using raytracing |
|  | Zoom in                              |
|  | Zoom out                             |
|  | Redraw                               |

|   |   |
|---|---|
|    | Fit drawing in display area                             |
|    | Rotate X clockwise                                      |
|    | Rotate X counterclockwise                               |
|    | Rotate Y clockwise                                      |
|    | Rotate Y counterclockwise                               |
|    | Rotate Z clockwise                                      |
|    | Rotate Z counterclockwise                               |
|    | Flip board view   |
|    | Pan board left  |
|    | Pan board right   |
|   | Pan board up  |
|  | Pan board down  |
|  | Enable/disable orthographic projection                  |
|  | Show/hide 3D models for through-hole components         |
|  | Show/hide 3D models for surface mount components        |
|  | Show/hide 3D models for components of type <i>other</i> |

5.5



网络检查

网络名称筛选:

☒ 显示 0 焊盘网络

☐ 分组依据:

通配符

| 网络  | 名称       | 焊盘计数 | 过孔计数 | 过孔长度      | 布线长度       |
|-----|----------|------|------|-----------|------------|
| 001 | /8MH-OUT | 2    | 0    | 0.0000 mm | 15.7661 mm |
| 002 | /ACK     | 2    | 0    | 0.0000 mm | 11.9103 mm |
| 003 | /AUTOFD- | 2    | 0    | 0.0000 mm | 15.6008 mm |
| 004 | /BIT0    | 2    | 0    | 0.0000 mm | 12.7361 mm |
| 005 | /BIT1    | 2    | 0    | 0.0000 mm | 9.2710 mm  |
| 006 | /BIT2    | 2    | 0    | 0.0000 mm | 12.1665 mm |
| 007 | /BIT3    | 2    | 0    | 0.0000 mm | 9.2099 mm  |
| 008 | /BIT4    | 2    | 0    | 0.0000 mm | 9.4378 mm  |
| 009 | /BIT5    | 2    | 0    | 0.0000 mm | 13.0082 mm |
| 010 | /BIT6    | 2    | 0    | 0.0000 mm | 10.7222 mm |
| 011 | /BIT7    | 2    | 1    | 1.5450 mm | 24.1561 mm |

创建报告...

确定

001: 8MH-OUT, 2 pads, 0 vias, 0.0000 mm via length, 15.7661 mm track length  
002: ACK, 2 pads, 0 vias, 0.0000 mm via length, 11.9103 mm track length  
003: AUTOFD-, 2 pads, 0 vias, 0.0000 mm via length, 15.6008 mm track length  
004: BIT0, 2 pads, 0 vias, 0.0000 mm via length, 12.7361 mm track length  
005: BIT1, 2 pads, 0 vias, 0.0000 mm via length, 9.2710 mm track length  
006: BIT2, 2 pads, 0 vias, 0.0000 mm via length, 12.1665 mm track length  
007: BIT3, 2 pads, 0 vias, 0.0000 mm via length, 9.2099 mm track length  
008: BIT4, 2 pads, 0 vias, 0.0000 mm via length, 9.4378 mm track length  
009: BIT5, 2 pads, 0 vias, 0.0000 mm via length, 13.0082 mm track length  
010: BIT6, 2 pads, 0 vias, 0.0000 mm via length, 10.7222 mm track length  
011: BIT7, 2 pads, 1 via, 1.5450 mm via length, 24.1561 mm track length

Pad Count and Via Count show the number of pads (surface mount and through hole) and vias on a net. Via Length shows the total height of each via (not accounting for which copper layers the via connects to). In other words, Via Length is equal to Via Count multiplied by the stackup height of the board. Track Length shows the total length of all track segments in a net, not accounting for topology. Die length shows the total of all Pad to Die Length values set for pads on the net.

Note

001: 8MH-OUT, 2 pads, 0 vias, 0.0000 mm via length, 15.7661 mm track length  
002: ACK, 2 pads, 0 vias, 0.0000 mm via length, 11.9103 mm track length  
003: AUTOFD-, 2 pads, 0 vias, 0.0000 mm via length, 15.6008 mm track length  
004: BIT0, 2 pads, 0 vias, 0.0000 mm via length, 12.7361 mm track length  
005: BIT1, 2 pads, 0 vias, 0.0000 mm via length, 9.2710 mm track length  
006: BIT2, 2 pads, 0 vias, 0.0000 mm via length, 12.1665 mm track length  
007: BIT3, 2 pads, 0 vias, 0.0000 mm via length, 9.2099 mm track length  
008: BIT4, 2 pads, 0 vias, 0.0000 mm via length, 9.4378 mm track length  
009: BIT5, 2 pads, 0 vias, 0.0000 mm via length, 13.0082 mm track length  
010: BIT6, 2 pads, 0 vias, 0.0000 mm via length, 10.7222 mm track length  
011: BIT7, 2 pads, 1 via, 1.5450 mm via length, 24.1561 mm track length

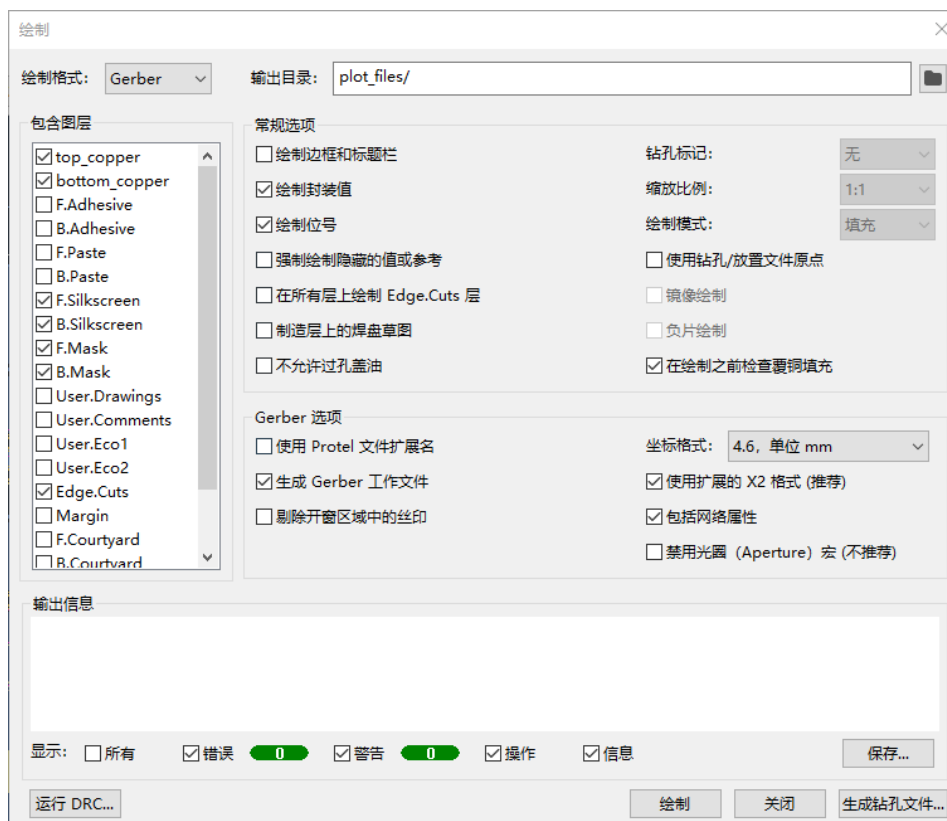
## Chapter 6

# &#x751f;&#x6210;&#x8f93;&#x51fa;

KiCad &#x53ef;&#x4ee5;&#x751f;&#x6210;&#x548c;&#x5bfc;&#x51fa;&#x591a;&#x79cd;&#x4e0d;&#x540c;&#x683c;&#x5f0f;  
 PCB &#x548c;&#x4e0e;&#x5916;&#x90e8;&#x8f6f;&#x4ef6;&#x7684;&#x63a5;&#x53e3;&#x5f88;&#x6709;&#x7528;&#x3002;  
 &#x8be5;&#x529f;&#x80fd;&#x53ef;&#x5728;&#x6587;&#x4ef6;&#x83dc;&#x5355;&#x7684;&#x51e0;&#x4e2a;&#x4e0d;&#x5  
 &#x5236;&#x9020;&#x8f93;&#x51fa;&#x90e8;&#x5206;&#x5305;&#x542b;&#x51c6;&#x5907;&#x5236;&#x9020; PCB &#x6240  
 &#x8f93;&#x51fa;&#x90e8;&#x5206;&#x5305;&#x542b;&#x751f;&#x6210;&#x53ef;&#x7531;&#x5916;&#x90e8;&#x8f6f;&#x4e  
 &#x7ed8;&#x56fe;&#x529f;&#x80fd;&#x5141;&#x8bb8;&#x4f60;&#x4ee5;&#x5404;&#x79cd;&#x683c;&#x5f0f;&#x5bfc;&#x51  
 PCB &#x7684; 2D &#x7ebf;&#x56fe;&#x3002; &#x6253;&#x5370;&#x529f;&#x80fd;&#x5141;&#x8bb8;&#x4f60;&#x5c06;  
 PCB &#x7684;&#x89c6;&#x56fe;&#x53d1;&#x9001;&#x5230; 2D &#x6253;&#x5370;&#x673a;&#x4e0a;&#x3002;

### 6.1 &#x5236;&#x9020;&#x8f93;&#x51fa;&#x548c;&#x7ed8;&#x5236;

KiCad &#x4f7f;&#x7528; Gerber &#x6587;&#x4ef6;&#x4f5c;&#x4e3a;&#x5176; PCB &#x5236;&#x9020;&#x7684;&#x4e3b;&#x8  
 &#x8981;&#x521b;&#x5efa; Gerber &#x6587;&#x4ef6;&#xff0c;&#x8bf7;&#x4ece;&#x6587;&#x4ef6;&#x83dc;&#x5355;&#x4e2d  
 Gerbers&#x3002; &#x7ed8;&#x56fe;&#x5bf9;&#x8bdd;&#x6846;&#x5c06;&#x6253;&#x5f00;&#xff0c;&#x5141;&#x8bb8;&#x4f6  
 Gerber &#x6587;&#x4ef6;&#x3002;



### 6.1.1 输出 Gerber 文件

在 PCB 编辑器中，选择“文件”菜单下的“输出 Gerber 文件”选项，将弹出“输出 Gerber 文件”对话框。

在“输出 Gerber 文件”对话框中，选择“Gerber”格式，并指定输出目录。在“包含图层”列表中，选择要输出的图层。在“常规选项”中，选择“绘制钻孔标记”、“缩放比例”和“绘制模式”。在“Gerber 选项”中，选择“使用 Protel 文件扩展名”、“生成 Gerber 工作文件”和“包括网络属性”。

在“输出信息”部分，显示错误和警告的数量。在“显示”部分，选择要显示的信息。单击“保存”按钮。

单击“运行 DRC”按钮，将弹出“运行 DRC”对话框。在“运行 DRC”对话框中，选择“运行 DRC”选项，并指定输出目录。

在“运行 DRC”对话框中，选择“运行 DRC”格式，并指定输出目录。在“包含图层”列表中，选择要输出的图层。在“常规选项”中，选择“绘制钻孔标记”、“缩放比例”和“绘制模式”。在“Gerber 选项”中，选择“使用 Protel 文件扩展名”、“生成 Gerber 工作文件”和“包括网络属性”。

在“输出信息”部分，显示错误和警告的数量。在“显示”部分，选择要显示的信息。单击“保存”按钮。

单击“运行 DRC”按钮，将弹出“运行 DRC”对话框。在“运行 DRC”对话框中，选择“运行 DRC”选项，并指定输出目录。

在“运行 DRC”对话框中，选择“运行 DRC”格式，并指定输出目录。在“包含图层”列表中，选择要输出的图层。在“常规选项”中，选择“绘制钻孔标记”、“缩放比例”和“绘制模式”。在“Gerber 选项”中，选择“使用 Protel 文件扩展名”、“生成 Gerber 工作文件”和“包括网络属性”。

在“输出信息”部分，显示错误和警告的数量。在“显示”部分，选择要显示的信息。单击“保存”按钮。

#### Note

KiCad 输出 Gerber 文件时，将输出 Gerber 文件保存到指定目录。在“输出 Gerber 文件”对话框中，选择“Gerber”格式，并指定输出目录。



&#x94bb;&#x5b54;&#x6807;&#x8bb0;&#xff1a; &#x5bf9;&#x4e8e; Gerber &#x4ee5;&#x5916;&#x7684;&#x7ed8;&#x56fe;&#x6807;&#x5b54;&#x6807;&#x8bb0;&#x53ef;&#x4ee5;&#x6309;&#x6210;&#x54c1;&#x5b54;&#x7684;&#x5b9e;&#x9645;&#x76f4;&#x5f84;) &#x4e0a;&#x521b;&#x5efa;&#xff0c;&#x4e5f;&#x53ef;&#x4ee5;&#x5728;&#x66f4;&#x5c0f;&#x7684;&#x5c0f;&#x7f29;&#x653e;&#xff1a; &#x5bf9;&#x4e8e;&#x652f;&#x6301;&#x975e; 1:1 &#x7f29;&#x653e;&#x7684;&#x6253;&#x5370;&#x81ea;&#x52a8;&#x7f29;&#x653e;&#x8bbe;&#x7f6e;&#x5c06;&#x7f29;&#x653e;&#x7ed8;&#x56fe;&#x4ee5;&#x9002;&#x542d;&#x5236;&#x6a21;&#x5f0f;&#xff1a; &#x5bf9;&#x4e8e;&#x67d0;&#x4e9b;&#x7ed8;&#x5236;&#x683c;&#x5f0f;&#x4f7f;&#x7528;&#x94bb;&#x5b54;/&#x653e;&#x7f6e;&#x6587;&#x4ef6;&#x539f;&#x70b9;&#xff1a; &#x542f;&#x7528;&#x7981;&#x7528;&#x65f6;&#xff0c;&#x5750;&#x6807;&#x539f;&#x70b9;&#x5c06;&#x662f;&#x7edd;&#x5bf9;&#x539f;&#x70b9;&#x56fe;&#x6846;&#x7684;&#x5de6;&#x4e0a;&#x89d2;)&#x3002;

&#x955c;&#x50cf;&#x7ed8;&#x5236;&#xff1a; &#x5f53;&#x8bbe;&#x7f6e;&#x8be5;&#x9009;&#x9879;&#x65f6;&#xff0c;&#x58d1f;&#x7247;&#x7ed8;&#x5236;&#xff1a; &#x5bf9;&#x4e8e;&#x67d0;&#x4e9b;&#x7ed8;&#x5236;&#x683c;&#x5f0f;&#x5728;&#x6b64;&#x6a21;&#x5f0f;&#x4e0b;&#xff0c;&#x5c06;&#x4e3a;&#x7535;&#x8def;&#x677f;&#x8f6e;&#x5ed3;&#x518b; PCB &#x4e2d;&#x5b58;&#x5728;&#x5bf9;&#x8c61;&#x7684;&#x4f4d;&#x7f6e;&#x7559;&#x4e0b;&#x7a7a;&#x767d;&#x533a;&#x7ed8;&#x5236;&#x524d;&#x68c0;&#x67e5;&#x6577;&#x94dc;&#x586b;&#x5145;&#xff1a; &#x542f;&#x7528;&#x540e;&#x5982;&#x679c;&#x8fc7;&#x671f;&#x5219;&#x91cd;&#x65b0;&#x586b;&#x5145;)&#x3002; &#x5982;&#x679c;&#x7981;&#x7528;&#x540e;&#x56fe;&#x6846;&#x7684;&#x5de6;&#x4e0a;&#x89d2;)&#x3002;

### 6.1.2 Gerber &#x9009;&#x9879;

&#x4f7f;&#x7528; Protel &#x6587;&#x4ef6;&#x6269;&#x5c55;&#x540d;&#xff1a; &#x542f;&#x7528;&#x540e;&#xff0c;&#x7ed8;&#x5236;&#x683c;&#x5f0f;&#x4f7f;&#x7528;&#x57fa;&#x4e8e; Protel (. GBL&#x3001; . GTL &#x7b49;) &#x7684;&#x6587;&#x5f53;&#x7981;&#x7528;&#x65f6;&#xff0c;&#x6587;&#x4ef6;&#x5c06;&#x6709; . gbr &#x7684;&#x6269;&#x5c55;&#x540d;&#x751f;&#x6210; Gerber &#x4f5c;&#x4e1a;&#x6587;&#x4ef6;&#xff1a; &#x5f00;&#x542f;&#x540e;&#xff0c; Gerber &#x4f5c; (. gbr job) &#x5c06;&#x4e0e;&#x4efb;&#x4f55; Gerber &#x6587;&#x4ef6;&#x4e00;&#x8d77;&#x751f;&#x6210;&#x3002; Gerber &#x4f5c;&#x4e1a;&#x6587;&#x4ef6;&#x662f; Gerber &#x683c;&#x5f0f;&#x7684;&#x6269;&#x5c55;&#xff0c;&#x5305;&#x533b;&#x5c42;&#x3001;&#x6750;&#x6599;&#x548c;&#x8868;&#x9762;&#x5904;&#x7406;&#x7684;&#x4fe1;&#x606f;&#x6709;&#x5173; Gerber &#x5de5;&#x4f5c;&#x6587;&#x4ef6;&#x7684;&#x66f4;&#x591a;&#x4fe1;&#x606f;&#xff0c;&#x8bf7; &#x7f51;&#x7ad9;&#x3002;

&#x5750;&#x6807;&#x683c;&#x5f0f;&#xff1a; &#x914d;&#x7f6e;&#x5750;&#x6807;&#x5728;&#x7ed8;&#x5236;&#x7684; Gerber &#x6587;&#x4ef6;&#x4e2d;&#x7684;&#x5b58;&#x50a8;&#x65b9;&#x5f0f;&#x3002; &#x8bf7;&#x54a8;&#x8be2;&#x60a0;

&#x4f7f;&#x7528;&#x6269;&#x5c55; X2 &#x683c;&#x5f0f;&#xff1a; &#x542f;&#x7528;&#x540e;&#xff0c;&#x7ed8;&#x5236;&#x683c;&#x5f0f;&#xff0c;&#x5176;&#x4e2d;&#x5305;&#x62ec;&#x6b64;&#x683c;&#x5f0f;&#x53ef;&#x80fd;&#x4e0e;&#x67d0;&#x4e9b;&#x5236;&#x9020;&#x5546;&#x4f7f;&#x7528;&#x7684;&#x8f6f;&#x4ef6;&#x4e0d;&#x517c;&#x5bb9;&#x3002;

&#x5305;&#x542b;&#x7f51;&#x8868;&#x5c5e;&#x6027;&#xff1a; &#x542f;&#x7528;&#x540e;&#xff0c;&#x7ed8;&#x5236;&#x683c;&#x5f0f;&#xff0c;&#x5305;&#x542b;&#x53ef;&#x7528;&#x4e8e;&#x5728; CAM &#x8f6f;&#x4ef6;&#x4e2d;&#x7981;&#x7528; X2 &#x683c;&#x5f0f;&#x6a21;&#x5f0f;&#x65f6;&#xff0c;&#x6b64;&#x4fe1;&#x606f;&#x5c06;&#x4f5c;&#x6587;&#x4ef6;&#x4e2d;&#x3002;

&#x7981;&#x7528;&#x5149;&#x5708;&#x5b8f;&#xff1a; &#x542f;&#x7528;&#x540e;&#xff0c;&#x6240;&#x6709;&#x5f62;&#x6b64;&#x8bbe;&#x7f6e;&#x4ec5;&#x5728;&#x5236;&#x9020;&#x5546;&#x8981;&#x6c42;&#x65f6;&#x7528;&#x4e8e;&#x4f7f;&#x8f6f;&#x4ef6;&#x517c;&#x5bb9;&#x3002;

### 6.1.3 Postscript &#x9009;&#x9879;

**Scale factor:** Controls how coordinates in the board file will be scaled to coordinates in the PostScript file. Using a different value for X and Y scale factors will result in a stretched / distorted output. These factors may be used to correct for scaling in the PostScript output device to achieve an exact-scale output.

**Track width correction:** A global factor that is added (or subtracted, if negative) from the size of tracks, vias, and pads when plotting a PostScript file. This factor may be used to correct for errors in the PostScript output device to achieve an exact-scale output.

&#x5f3a;&#x5236; A4 &#x8f93;&#x51fa;&#xff1a; &#x542f;&#x7528;&#x540e;&#xff0c;&#x751f;&#x6210;&#x7684; PostScript &#x6587;&#x4ef6;&#x5c06;&#x4e3a; A4 &#x5927;&#x5c0f;&#xff0c;&#x5373;&#x4f7f; KiCad &#x7535;&#x8def;&#x677f;&#x6b64;&#x8bbe;&#x7f6e;&#x4ec5;&#x5728;&#x5236;&#x9020;&#x5546;&#x8981;&#x6c42;&#x65f6;&#x7528;&#x4e8e;&#x4f7f;&#x8f6f;&#x4ef6;&#x517c;&#x5bb9;&#x3002;

#### 6.1.4 SVG &#x9009;&#x9879;

&#x5355;&#x4f4d;&#xff1a; &#x63a7;&#x5236; SVG &#x6587;&#x4ef6;&#x4e2d;&#x4f7f;&#x7528;&#x7684;&#x5355;&#x4f4d  
&#x7531;&#x4e8e; SVG &#x683c;&#x5f0f;&#x6ca1;&#x6709;&#x6307;&#x5b9a;&#x7684;&#x5355;&#x4f4d;&#x7cfb;&#x7edf;  
&#x56e0;&#x6b64;&#x60a8;&#x5fc5;&#x987b;&#x4f7f;&#x7528;&#x4e0e;&#x5bfc;&#x5165;&#x5230;&#x5176;&#x4ed6;&#x8  
&#x7cbe;&#x5ea6;&#xff1a; &#x63a7;&#x5236;&#x5c06;&#x4f7f;&#x7528;&#x591a;&#x5c11;&#x4e2a;&#x6709;&#x6548;&#x

#### 6.1.5 DXF &#x9009;&#x9879;

&#x4f7f;&#x7528;&#x56fe;&#x5f62;&#x9879;&#x76ee;&#x7684;&#x8f6e;&#x5ed3;&#x7ed8;&#x5236;&#x56fe;&#x5f62;&#x  
DXF &#x6587;&#x4ef6;&#x4e2d;&#x7684;&#x56fe;&#x5f62;&#x5f62;&#x72b6;&#x6ca1;&#x6709;&#x5bbd;&#x5ea6;&#x3002;  
&#x6b64;&#x9009;&#x9879;&#x63a7;&#x5236;&#x5982;&#x4f55;&#x5c06; KiCad &#x7535;&#x8def;&#x677f;&#x4e2d;&#x517  
(&#x539a;&#x5ea6;) &#x7684;&#x56fe;&#x5f62;&#x56fe;&#x5f62;&#x7ed8;&#x5236;&#x5230; DXF &#x6587;&#x4ef6;&#x300  
&#x542f;&#x7528;&#x6b64;&#x9009;&#x9879;&#x540e;&#xff0c;&#x5c06;&#x7ed8;&#x5236;&#x5f62;&#x72b6;&#x7684;&#x5  
&#x7981;&#x7528;&#x6b64;&#x9009;&#x9879;&#x65f6;&#xff0c;&#x5c06;&#x6253;&#x5370;&#x5f62;&#x72b6;&#x7684;&#x4  
(&#x5e76;&#x4e14;&#x5f62;&#x72b6;&#x7684;&#x539a;&#x5ea6;&#x5728;&#x751f;&#x6210;&#x7684; DXF &#x6587;&#x4ef6;  
&#x4f7f;&#x7528; KiCad &#x5b57;&#x4f53;&#x7ed8;&#x5236;&#x6587;&#x672c;&#xff1a; &#x542f;&#x7528;&#x540e;&#xf  
&#x8bbe;&#x8ba1;&#x4e2d;&#x7684;&#x6587;&#x672c;&#x5c06;&#x4f7f;&#x7528; KiCad &#x5b57;&#x4f53;&#x7ed8;&#x523  
&#x7981;&#x7528;&#x65f6;&#xff0c;&#x6587;&#x672c;&#x5c06;&#x4f5c;&#x4e3a; DXF &#x6587;&#x672c;&#x5bf9;&#x8c61;  
KiCad &#x7535;&#x8def;&#x677f;&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x663e;&#x793a;&#x7684;&#x5b8c;&#x5168;&#x76f8  
&#x5bfc;&#x51fa;&#x5355;&#x4f4d;&#xff1a; &#x63a7;&#x5236;&#x5c06;&#x5728; DXF &#x6587;&#x4ef6;&#x4e2d;&#x4f7f  
&#x7531;&#x4e8e; DXF &#x683c;&#x5f0f;&#x6ca1;&#x6709;&#x6307;&#x5b9a;&#x7684;&#x5355;&#x4f4d;&#x7cfb;&#x7edf;

#### 6.1.6 HPGL &#x9009;&#x9879;

&#x9ed8;&#x8ba4;&#x7b14;&#x5c3a;&#x5bf8;&#xff1a; &#x63a7;&#x5236;&#x7528;&#x4e8e;&#x521b;&#x5efa;&#x56fe;&#x

### 6.2 &#x94bb;&#x5b54;&#x6587;&#x4ef6;

KiCad &#x53ef;&#x4ee5;&#x751f;&#x6210;&#x5927;&#x591a;&#x6570; PCB &#x5236;&#x9020;&#x5de5;&#x827a;&#x6240;&#x  
Excellon &#x6216; Gerber X2 &#x683c;&#x5f0f;&#x7684;&#x6570;&#x63a7;&#x94bb;&#x5b54;&#x6587;&#x4ef6;&#x3002;  
KiCad &#x8fd8;&#x53ef;&#x4ee5;&#x751f;&#x6210;&#x94bb;&#x5b54;&#x56fe;&#xff1a;&#x4e00;&#x4e2a;&#x663e;&#x793a;  
&#x4ece; "&#x5236;&#x9020;&#x8f93;&#x51fa;" &#x83dc;&#x5355;&#x4e2d;&#x9009;&#x62e9; "&#x751f;&#x6210;&#x94bb;&  
&#x9009;&#x9879;&#x6765;&#x6253;&#x5f00;&#x8be5;&#x5bf9;&#x8bdd;&#x6846;&#xff1a;



输出文件路径: plot\_files/

钻孔文件格式: Excellon  
☐ Y 轴镜像  
☐ 最少钻头种类  
☐ 金属化 (PTH) 和非金属化 (NPTH) 孔在单个文件中  
 椭圆孔钻孔模式  
☒ 使用钻孔命令 (推荐)  
☐ 使用备用钻孔模式  
 Gerber X2  
 映射文件格式  
☐ HPGL  
☒ PostScript  
☐ Gerber  
☐ DXF  
☐ SVG  
☐ PDF

钻孔文件原点  
☒ 绝对  
☐ 钻孔/放置文件原点  
 钻孔单位  
☐ mm  
☒ 英寸  
 零的格式  
☒ 十进制格式(推荐)  
☐ 去掉前导零  
☐ 去掉后导零  
☐ 保留零  
 精度: 2x4

孔数量  
 金属化焊盘: 317  
 非金属化焊盘: 0  
 通孔: 84  
 微孔: 0  
 埋孔: 0

PTH (PTH) 和非金属化 (NPTH) 孔在单个文件中  
 Excellon  
☒ 使用钻孔命令 (推荐)  
☐ 使用备用钻孔模式  
 Gerber X2  
 映射文件格式  
☐ HPGL  
☒ PostScript  
☐ Gerber  
☐ DXF  
☐ SVG  
☐ PDF

钻孔文件原点  
☒ 绝对  
☐ 钻孔/放置文件原点  
 钻孔单位  
☐ mm  
☒ 英寸  
 零的格式  
☒ 十进制格式(推荐)  
☐ 去掉前导零  
☐ 去掉后导零  
☐ 保留零  
 精度: 2x4

孔数量  
 金属化焊盘: 317  
 非金属化焊盘: 0  
 通孔: 84  
 微孔: 0  
 埋孔: 0

PTH (PTH) 和非金属化 (NPTH) 孔在单个文件中  
 Excellon  
☒ 使用钻孔命令 (推荐)  
☐ 使用备用钻孔模式  
 Gerber X2  
 映射文件格式  
☐ HPGL  
☒ PostScript  
☐ Gerber  
☐ DXF  
☐ SVG  
☐ PDF

钻孔文件原点  
☒ 绝对  
☐ 钻孔/放置文件原点  
 钻孔单位  
☐ mm  
☒ 英寸  
 零的格式  
☒ 十进制格式(推荐)  
☐ 去掉前导零  
☐ 去掉后导零  
☐ 保留零  
 精度: 2x4

孔数量  
 金属化焊盘: 317  
 非金属化焊盘: 0  
 通孔: 84  
 微孔: 0  
 埋孔: 0

PTH (PTH) 和非金属化 (NPTH) 孔在单个文件中  
 Excellon  
☒ 使用钻孔命令 (推荐)  
☐ 使用备用钻孔模式  
 Gerber X2  
 映射文件格式  
☐ HPGL  
☒ PostScript  
☐ Gerber  
☐ DXF  
☐ SVG  
☐ PDF

钻孔文件原点  
☒ 绝对  
☐ 钻孔/放置文件原点  
 钻孔单位  
☐ mm  
☒ 英寸  
 零的格式  
☒ 十进制格式(推荐)  
☐ 去掉前导零  
☐ 去掉后导零  
☐ 保留零  
 精度: 2x4

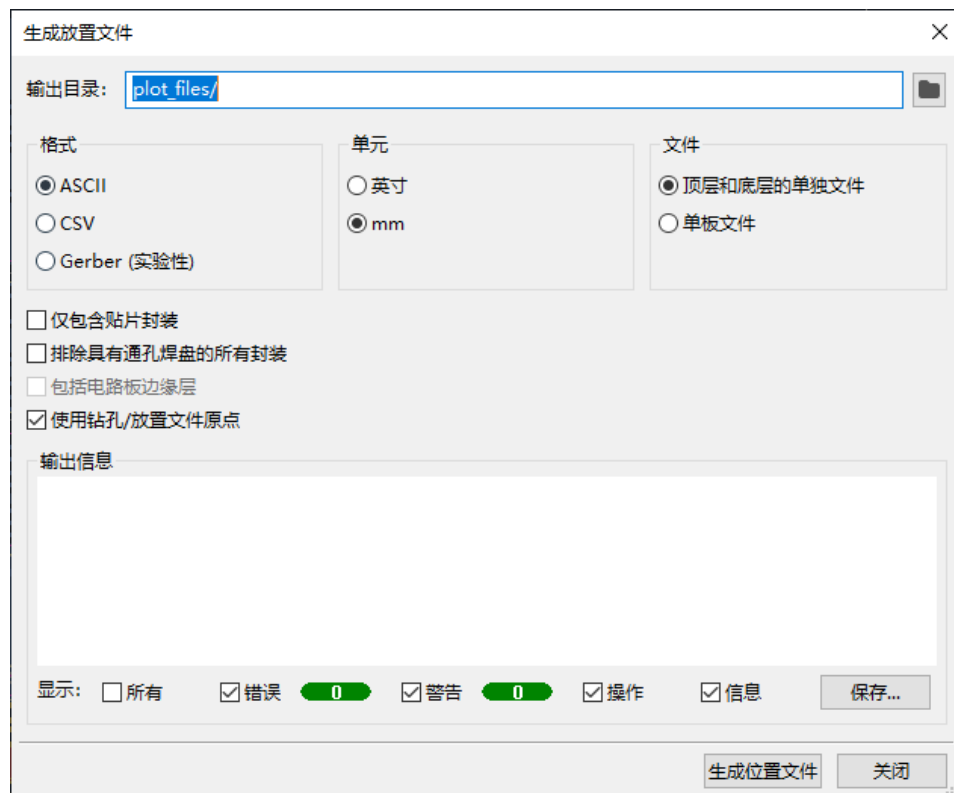
孔数量  
 金属化焊盘: 317  
 非金属化焊盘: 0  
 通孔: 84  
 微孔: 0  
 埋孔: 0

## 6.3 输出文件

PCB 设置

### Note

当启用“仅包含贴片封装”时，只有具有 SMD 制造属性的封装将包含在位置文件中。请与您的制造商检查，以确定是否应包含或排除非 SMD 封装。



当启用“仅包含贴片封装”时，只有具有 SMD 制造属性的封装将包含在位置文件中。请与您的制造商检查，以确定是否应包含或排除非 SMD 封装。

当启用“仅包含贴片封装”时，只有具有 SMD 制造属性的封装将包含在位置文件中。请与您的制造商检查，以确定是否应包含或排除非 SMD 封装。

**Include only SMD footprints:** When enabled, only footprints with the SMD fabrication attribute will be included. Check with your manufacturer to determine if non-SMD footprints should be included or excluded from the position file.

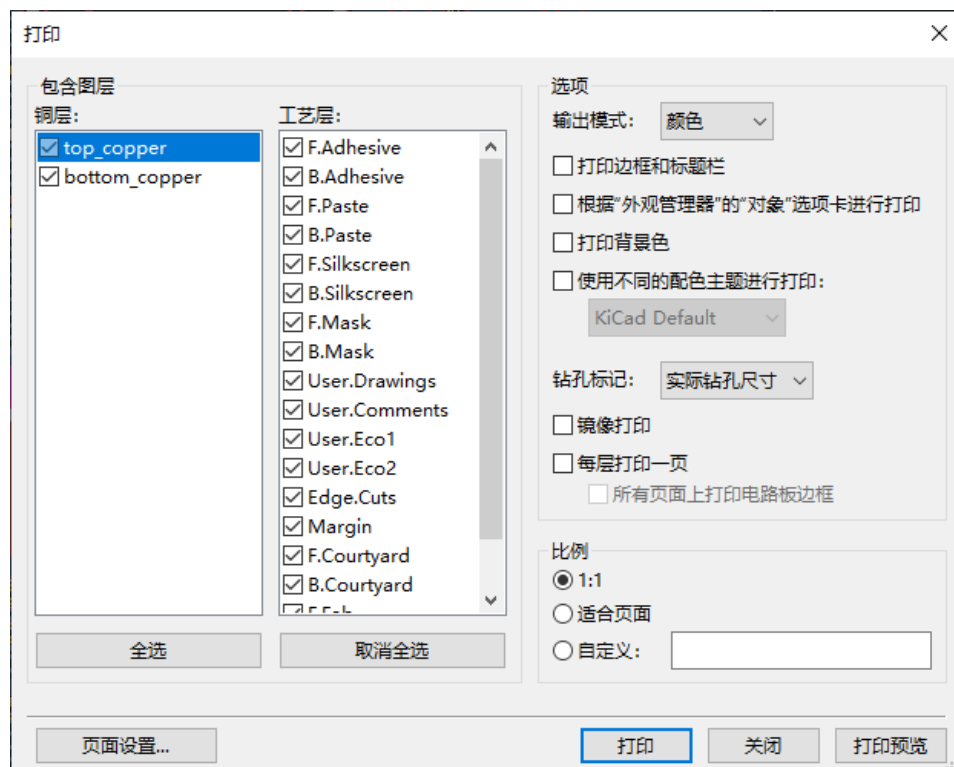
当启用“仅包含贴片封装”时，只有具有 SMD 制造属性的封装将包含在位置文件中。请与您的制造商检查，以确定是否应包含或排除非 SMD 封装。

## 6.4 生成位置文件

KiCad 生成位置文件 (ASCII, CSV, Gerber) 的对话框。该对话框允许用户指定输出目录、格式、单位、文件类型以及是否包含 SMD 封装等选项。

## 6.5 打印

KiCad 打印



Print

包含图层

铜层:

- ☒ top\_copper
- ☒ bottom\_copper

工艺层:

- ☒ F.Adhesive
- ☒ B.Adhesive
- ☒ F.Paste
- ☒ B.Paste
- ☒ F.Silkscreen
- ☒ B.Silkscreen
- ☒ F.Mask
- ☒ B.Mask
- ☒ User.Drawings
- ☒ User.Comments
- ☒ User.Eco1
- ☒ User.Eco2
- ☒ Edge.Cuts
- ☒ Margin
- ☒ F.Courtyard
- ☒ B.Courtyard

选项

输出模式: 颜色

☐ 打印边框和标题栏

☐ 根据“外观管理器”的“对象”选项卡进行打印

☐ 打印背景色

☐ 使用不同的配色主题进行打印:

KiCad Default

钻孔标记: 实际钻孔尺寸

☐ 镜像打印

☐ 每层打印一页

☐ 所有页面上打印电路板边框

比例

☒ 1:1

☐ 适合页面

☐ 自定义:

全选 取消全选

页面设置... 打印 关闭 打印预览

## 6.6 打印

KiCad 打印

Specctra .DSN

打印

**Note**

TODO&#xff1a;&#x6587;&#x6863; GenCAD &#x5bfc;&#x51fa;&#x5668;

**Note**

TODO: &#x6587;&#x6863; VRML &#x5bfc;&#x51fa;&#x5668;

### 6.6.1 IDF Exporter

The IDF exporter exports an **IDFv3** compliant board (.emn) and library (.emp) file for communicating mechanical dimensions to a mechanical CAD package. The exporter exports the board outline and cutouts, all pad and mounting through holes including slotted holes, and component outlines; this is the most basic set of mechanical data required for interaction with mechanical designers. All other entities described in the IDFv3 specification are currently not exported.

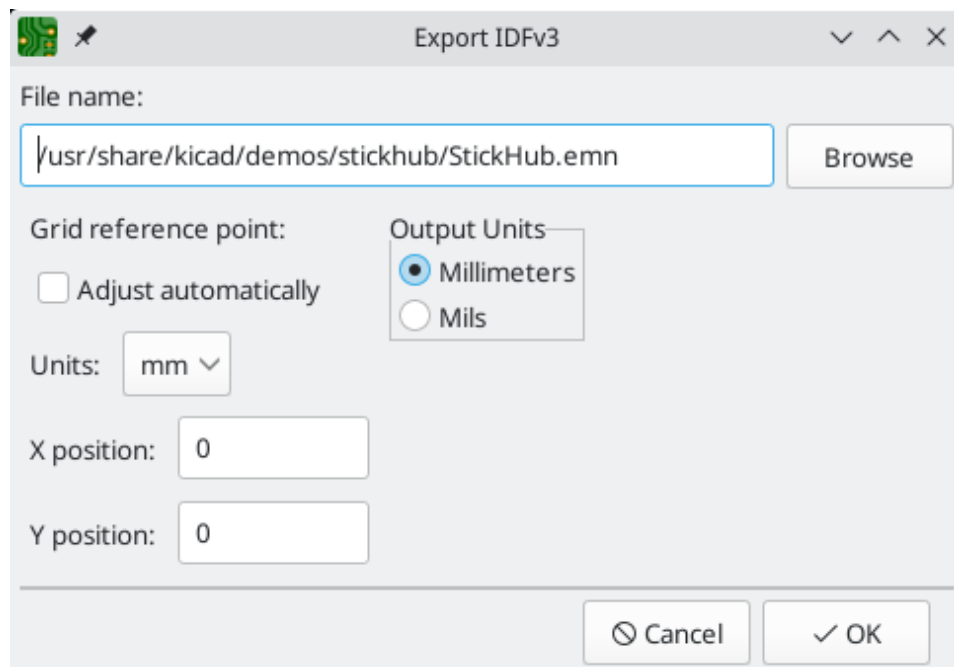
**Note**

You must attach IDF component models to your design's footprints before they will be included in the exported model. For more information on attaching models to footprints, see the [footprint documentation](#). Some IDF-specific guidance is included in the [Advanced Topics documentation](#).

**Note**

For more information on creating IDF component models, including descriptions of the IDF utility tools included with KiCad, see the [Advanced Topics documentation](#).

Once models have been specified for all desired components, the model of the board can be exported. In the PCB Editor, select **File** → **Export** → **IDFv3...**



**Grid reference point:** Choose where the exported model's reference point should be. If the **Adjust automatically** option is selected, KiCad will set the reference point to the centroid of the PCB. Otherwise, the reference point is set relative to the display origin.

**Output units:** Choose whether the exported model's units are millimeters or mils.

The outputs can be viewed directly in a mechanical CAD application or converted to VRML using the [idf2vrml tool](#).

**Note**

TODO: STEP

**Note**

TODO: SVG

**Note**

TODO: CMP

**Hyperlynx**; &#x521b;&#x5efa;&#x9002;&#x5408;&#x5bfc;&#x5165; Mentor Graphics(Siemens) HyperLynx &#x6a21;&#x

## Chapter 7

# Footprint Libraries

### 7.1 Footprint Libraries

KiCad's footprint library management system allows directly using several types of footprint libraries:

- KiCad `.pretty` footprint libraries (folders with `.pretty` extension, containing `.kicad_mod` files)
- KiCad Legacy footprint libraries (`.mod` files)
- GEDA libraries (folders containing `.fp` files)
- Eagle footprint libraries

---

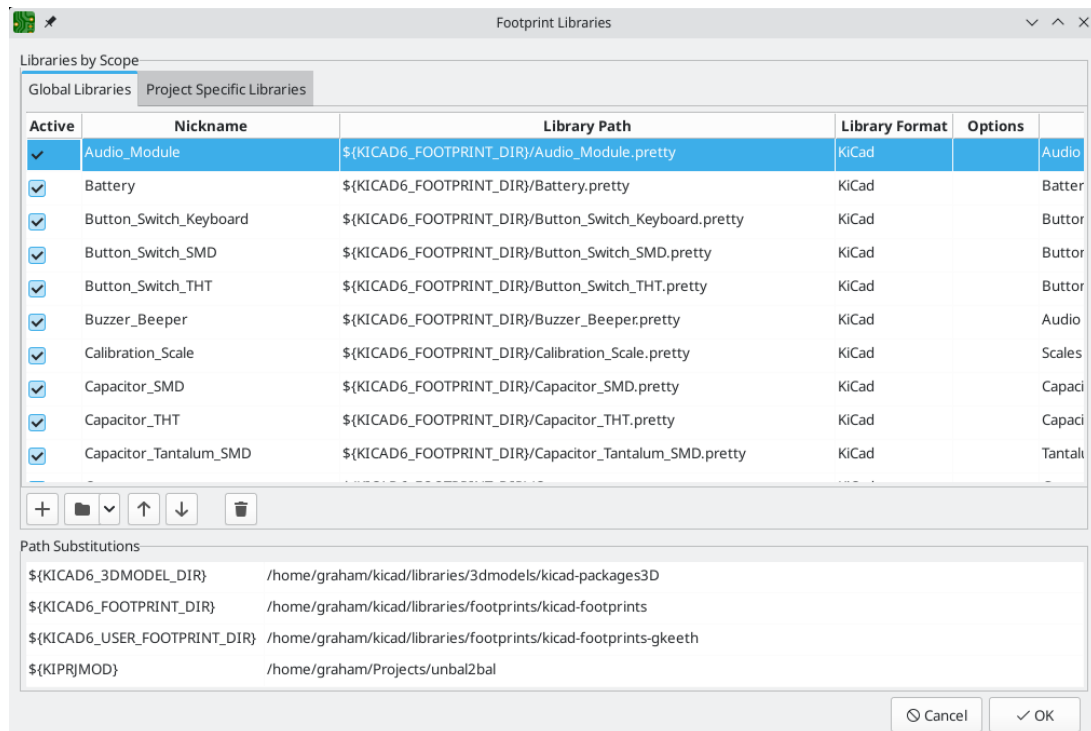
**Note**

KiCad only supports writing to KiCad's native `.pretty` format footprint libraries (and the `.kicad_mod` footprint files within them). All other footprint library formats are read-only.

---

KiCad uses a table of footprint libraries to map footprint libraries of any supported library type to a library nickname. KiCad uses a global footprint library table as well as a table specific to each project. To edit either footprint library table, use **Preferences** → **Manage Footprint Libraries...**





The global footprint library table contains the list of libraries that are always available regardless of the currently loaded project. The table is saved in the file `fp-lib-table` in the KiCad configuration folder. [The location of this folder](#) depends on the operating system being used.




The project specific footprint library table contains the list of libraries that are available specifically for the currently loaded project. If there are any project-specific footprint libraries, the table is saved in the file `fp-lib-table` in the project folder.



### 7.1.1 Initial Configuration

The first time the PCB Editor (or any other KiCad tool that uses footprints) runs and the global footprint table file `fp-lib-table` is not found, KiCad will guide the user through setting up a new footprint library table. This process is described [above](#).

### 7.1.2 Managing Table Entries

Footprint libraries can only be used if they have been added to either the global or project-specific footprint library table.

Add a library either by clicking the  button and selecting a library or clicking the  button and typing the path to a library file. The selected library will be added to the currently opened library table (Global or Project Specific). Libraries can be removed by selecting desired library entries and clicking the  button.

The  and  buttons move the selected library up and down in the library table. This does not affect the display order of libraries in the Footprint Library Browser, Footprint Editor, or Add Footprint tool.

Libraries can be made inactive by unchecking the **Active** checkbox in the first column. Inactive libraries are still in the library table but do not appear in any library browsers and are not loaded from disk, which can reduce loading times.

A range of libraries can be selected by clicking the first library in the range and then `kbd:[Shift]-clicking` the last library in the range.

Each library must have a unique nickname: duplicate library nicknames are not allowed in the same table. However, nicknames can be duplicated between the global and project library tables. Libraries in the project table take precedence over libraries with the same name in the global table.

Library nicknames do not have to be related to the library filename or path. The colon character (:) cannot be used in library nicknames or footprint names because it is used as a separator between nicknames and footprints.

Each library entry must have a valid path. Paths can be defined as absolute, relative, or by [environment variable substitution](#).

The appropriate library format must be selected in order for the library to be properly read. KiCad supports reading KiCad (.pretty), KiCad legacy (.mod), Eagle (.lbr), and GEDA (folder with .fp files) footprint libraries.

There is an optional description field to add a description of the library entry. The option field is not used at this time so adding options will have no effect when loading libraries.

### 7.1.3 Environment Variable Substitution

The footprint library tables support environment variable substitution, which allows you to define environment variables containing custom paths to where your libraries are stored. Environment variable substitution is supported by using the syntax `${ENV_VAR_NAME}` in the footprint library path.

By default, KiCad defines several environment variables:

- `${KIPROJMOD}` points to the current project directory and cannot be modified.
- `${KICAD6_FOOTPRINT_DIR}` points to the default location of KiCad's standard footprint libraries.
- `${KICAD6_SYMBOL_DIR}` points to the default location of KiCad's standard symbol libraries.
- `${KICAD6_3DMODEL_DIR}` points to the default location of KiCad's standard 3D model libraries.
- `${KICAD6_TEMPLATE_DIR}` points to the default location of KiCad's standard template library.

`${KIPROJMOD}` cannot be redefined, but the other environment variables can be redefined and new environment variables added in the **Preferences** → **Configure Paths...** dialog.

Using environment variables in the footprint library tables allows libraries to be relocated without breaking the footprint library tables, so long as the environment variables are updated when the library location changes.

`${KIPROJMOD}` allows libraries to be stored in the project folder without having to use an absolute path in the project library table. This makes it possible to relocate projects without breaking their project library tables.

### 7.1.4 Using the GitHub Plugin

---

#### Note

KiCad removed support for the GitHub library plugin in version 6.0.

---

## 7.2

---

#### Note

TODO

---

### 7.2.1

### 7.2.2

---

#### Note

---

**7.2.3 5c01;88c5;5411;5bfc;**

6709;5173;521b;5efa;65b0;7684;5c01;88c5;5411;5bfc;7684;66f4;591a;x4

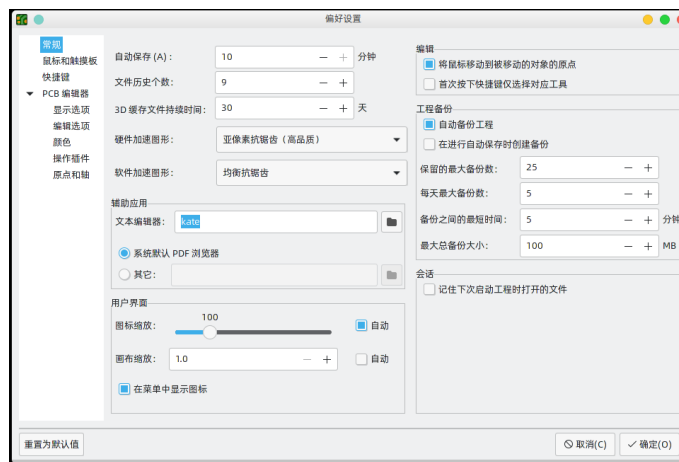
# Chapter 8

8;7;e3b;9898;

## 8.1 $\mathbb{R}^n$ and $\mathbb{C}^n$

Pcbnew &#x6709;&#x5404;&#x79cd;&#x504f;&#x597d;&#x8bbe;&#x7f6e;&#xff0c;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x504f  
KiCad &#x7684;&#x6240;&#x6709;&#x90e8;&#x5206;&#x4e00;&#x6837;&#xff0c;Pcbnew &#x7684;&#x504f;&#x597d;&#x8bbe;  
KiCad &#x6b21;&#x8981;&#x7248;&#x672c;&#x4e4b;&#x95f4;&#x76f8;&#x4e92;&#x72ec;&#x7acb;&#xff0c;&#x4ece;&#x800c;  
&#x504f;&#x597d;&#x8bbe;&#x7f6e;&#x5bf9;&#x8bdd;&#x6846;&#x7684;&#x7b2c;&#x4e00;&#x90e8;&#x5206; (&#x901a;&#x  
&#x5728;&#x6240;&#x6709; KiCad &#x7a0b;&#x5e8f;&#x4e4b;&#x95f4;&#x5171;&#x4eab;&#x3002;KiCad &#x624b;&#x518c;  
Pcbnew &#x7684;&#x5feb;&#x6377;&#x952e;&#x53ea;&#x6709;&#x5728; Pcbnew &#x8fd0;&#x884c;&#x65f6;&#x624d;&#x4f1a

### 8.1.1 $\text{e}^{\text{a}} \cdot 10^{\text{b}}$



&#x6e32;&#x67d3;&#x5f15;&#x64ce;&#xff1a; &#x63a7;&#x5236;&#x662f;&#x5426;&#x4f7f;&#x7528;&#x786c;&#x4ef6;&#x5  
&#x7f51;&#x683c;&#x6837;&#x5f0f;&#xff1a; &#x63a7;&#x5236;&#x5982;&#x4f55;&#x7ed8;&#x5236;&#x8def;&#x7ebf;&#x7  
&#x7f51;&#x683c;&#x539a;&#x5ea6;&#xff1a; &#x63a7;&#x5236;&#x7f51;&#x683c;&#x7ebf;&#x6216;&#x7f51;&#x683c;&#x  
&#x6700;&#x5c0f;&#x7f51;&#x683c;&#x95f4;&#x8ddd;&#xff1a; &#x63a7;&#x5236;&#x4e24;&#x6761;&#x7f51;&#x683c;&#x  
(&#x4ee5;&#x50cf;&#x7d20;&#x4e3a;&#x5355;&#x4f4d;)&#x3002;&#x65e0;&#x8bba;&#x5f53;&#x524d;&#x7684;&#x6805;&#x  
&#x6355;&#x6349;&#x5230;&#x7f51;&#x683c;&#xff1a; &#x63a7;&#x5236;&#x4f55;&#x65f6;&#x5c06;&#x7ed8;&#x5236;&#x  
“&#x59cb;&#x7ec8;”&#x5c06;&#x542f;&#x7528;&#x5bf9;&#x9f50;&#xff0c;&#x5373;&#x4f7f;&#x7f51;&#x683c;&#x5904;&#x4  
“&#x5f53;&#x7f51;&#x683c;&#x663e;&#x793a;&#x65f6;”&#x5c06;&#x4ec5;&#x5728;&#x7f51;&#x683c;&#x53ef;&#x89c1;&#x

### Note

&#x6309;&#x4f4f; kbd:[Ctrl] &#x53ef;&#x4ee5;&#x6682;&#x65f6;&#x7981;&#x7528;&#x7f51;&#x683c;&#x6355;&#x6349;&#x3002;

&#x5149;&#x6807;&#x5f62;&#x72b6;&#xff1a;&#x63a7;&#x5236;&#x7f16;&#x8f91;&#x5149;&#x6807;&#x662f;&#x7ed8;&#x8fd8;&#x8986;&#x76d6;&#x6574;&#x4e2a;&#x7ed8;&#x56fe;&#x753b;&#x5e03;&#x7684;&#x4e00;&#x7ec4;&#x7ebf;)&#x3002;&#x7f16;&#x8f91;&#x5149;&#x6807;&#x663e;&#x793a;&#x4e0b;&#x4e00;&#x4e2a;&#x7ed8;&#x56fe;&#x6216;&#x7f16;&#x8fd8;&#x59cb;&#x7ec8;&#x663e;&#x793a;&#x5341;&#x5b57;&#x51c6;&#x7ebf;&#xff1a;&#x63a7;&#x5236;&#x662f;&#x59cb;&#x8fd8;&#x662f;&#x4ec5;&#x5728;&#x7f16;&#x8f91;&#x6216;&#x7ed8;&#x56fe;&#x5de5;&#x5177;&#x5904;&#x4e8e;&#x60c5;&#x7f51;&#x7edc;&#x540d;&#x79f0;&#xff1a;&#x63a7;&#x5236;&#x662f;&#x5426;&#x5728;&#x94dc;&#x5bf9;&#x8c61;&#x8fd9;&#x4e9b;&#x6807;&#x7b7e;&#x4ec5;&#x4f5c;&#x4e3a;&#x7f16;&#x8f91;&#x6307;&#x5357;&#xff0c;&#x4e0d;&#x4f10;&#x663e;&#x793a;&#x710a;&#x76d8;&#x7f16;&#x53f7;&#xff1a;&#x63a7;&#x5236;&#x662f;&#x5426;&#x5728;&#x5c01;&#x76d8;&#x663e;&#x793a;&#x710a;&#x76d8;<&#x65e0;&#x7f51;&#x7edc;>&#x6307;&#x793a;&#x5668;&#xff1a;&#x63a7;&#x5236;&#x7f51;&#x7edc;&#x7684;&#x710a;&#x76d8;&#x3002;

&#x5e03;&#x7ebf;&#x95f4;&#x9699;&#xff1a;&#x63a7;&#x5236;&#x662f;&#x5426;&#x663e;&#x793a;&#x5e03;&#x7ebf;&#x95f4;&#x9699;&#x8f6e;&#x5ed3;&#x663e;&#x793a;&#x4e3a;&#x5bf9;&#x8c61;&#x5468;&#x56f4;&#x7684;&#x7ec6;&#x95f4;&#x5982;&#x7ea6;&#x675f;&#x548c;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x6240;&#x5b9a;&#x4e49;&#x3002;

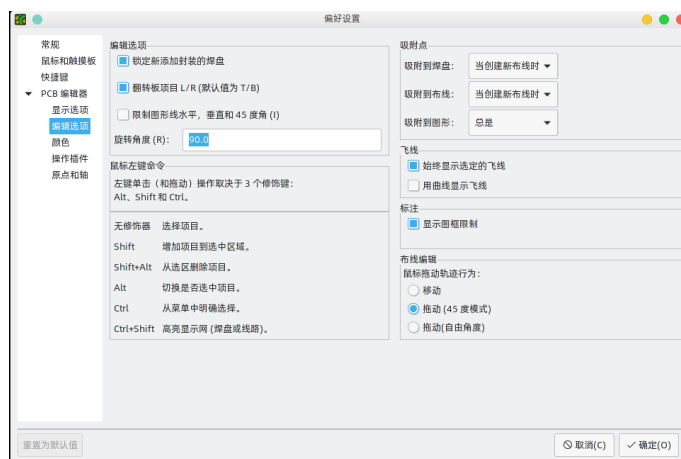
&#x663e;&#x793a;&#x710a;&#x76d8;&#x95f4;&#x9699;&#xff1a; &#x63a7;&#x5236;&#x662f;&#x5426;&#x663e;&#x793a;&#x

&#x4ea4;&#x53c9;&#x63a2;&#x6d4b;&#x9879;&#x7684;&#x5c45;&#x4e2d;&#x89c6;&#x56fe;&#xff1a;&#x5f53; Eess-  
chema &#x548c; Pcbnew &#x90fd;&#x5728;&#x8fd0;&#x884c;&#x65f6;&#xff0c;&#x63a7;&#x5236;&#x70b9;&#x51fb; Eess-  
chema &#x4e2d;&#x7684;&#x5143;&#x4ef6;&#x6216;&#x5f15;&#x811a;&#x662f;&#x5426;&#x4f1a;&#x4f7f; Pcbnew &#x89c6;&

&#x7f29;&#x653e;&#x4ee5;&#x9002;&#x5408;&#x4ea4;&#x53c9;&#x63a2;&#x6d4b;&#x9879;&#xff1a;&#x63a7;&#x5236;&#x5c01;&#x88c5;&#x6216;&#x710a;&#x76d8;&#x3002;

&#x9ad8;&#x4eae;&#x663e;&#x793a;&#x4ea4;&#x53c9;&#x63a2;&#x6d4b;&#x7f51;&#x7edc;&#xff1a; &#x63a7;&#x5236;&#x662f;&#x5426;&#x5728; Pcbnew &#x4e2d;&#x9ad8;&#x4eae;&#x663e;&#x793a; Eesschema &#x4e2d;&#x9ad8;&#x4eae;&#x

### 8.1.2 8.1.2 8.1.2 8.1.2

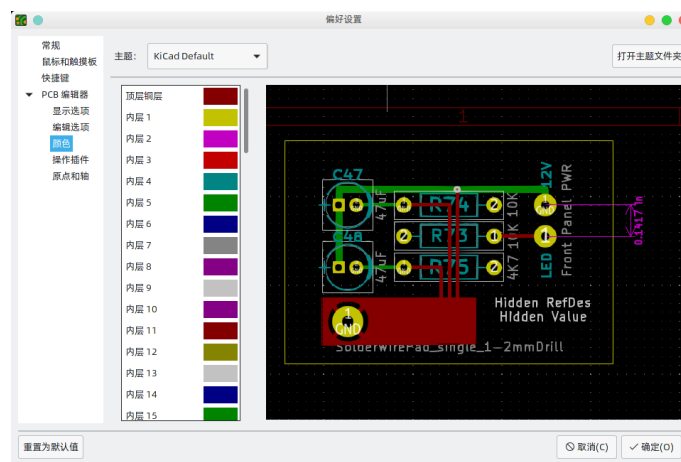


&#x7ffb;&#x8f6c;&#x7535;&#x8def;&#x677f;&#x9879;&#x76ee; L/R&#xff1a; &#x63a7;&#x5236;&#x5728;&#x9876;&#x5c42  
&#x9009;&#x4e2d;&#x65f6;&#xff0c;&#x9879;&#x76ee;&#x4ece;&#x5de6;&#x5411;&#x53f3;&#x7ffb;&#x8f6c;(&#x56f4;&#x7e  
&#x53d6;&#x6d88;&#x9009;&#x4e2d;&#x65f6;&#xff0c;&#x9879;&#x76ee;&#x4ece;&#x4e0a;&#x5411;&#x4e0b;&#x7ffb;&#x8f6c;  
&#x65cb;&#x8f6c;&#x547d;&#x4ee4;&#x7684;&#x6b65;&#x957f;&#xff1a; &#x63a7;&#x5236;&#x6bcf;&#x6b21;&#x47f7;&#x  
&#x5141;&#x8bb8;&#x81ea;&#x7531;&#x710a;&#x76d8;&#xff1a; &#x63a7;&#x5236;&#x5c01;&#x88c5;&#x710a;&#x76d8;&

**Magnetic points:** This section controls object snapping, also called magnetic points. Object snapping takes precedence over grid snapping when it is enabled. Object snapping only works to objects on the active layer. Hold kbd:[Shift] to temporarily disable object snapping.

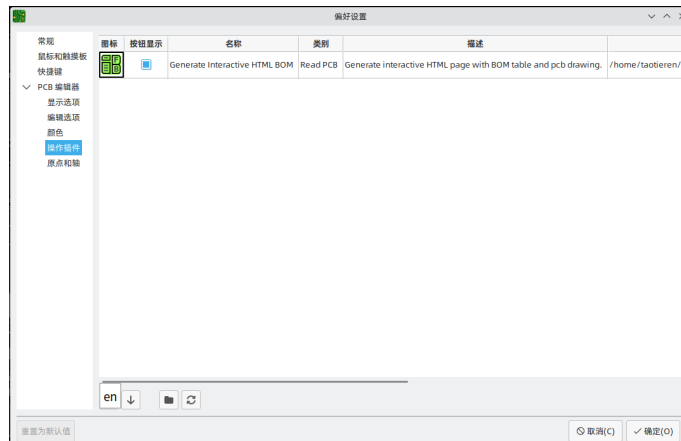
&#x6355;&#x6349;&#x710a;&#x76d8;&#xff1a; &#x63a7;&#x5236;&#x7f16;&#x8f91;&#x5149;&#x6807;&#x4f55;&#x65f6;&#x  
&#x6355;&#x6349;&#x5230;&#x5e03;&#x7ebf;&#xff1a; &#x63a7;&#x5236;&#x7f16;&#x8f91;&#x5149;&#x6807;&#x4f55;&#x  
&#x5bf9;&#x9f50;&#x56fe;&#x5f62;&#xff1a; &#x63a7;&#x5236;&#x7f16;&#x8f91;&#x5149;&#x6807;&#x4f55;&#x65f6;&#x5  
&#x59cb;&#x7ec8;&#x663e;&#x793a;&#x9009;&#x5b9a;&#x7684;&#x98de;&#x7ebf;&#xff1a; &#x542f;&#x7528;&#x540e;&#  
&#x9009;&#x5b9a;&#x5c01;&#x88c5;&#x5916;&#x5f62;&#x7684;&#x98de;&#x7ebf;&#x4e5f;&#x5c06;&#x59cb;&#x7ec8;&#x6  
&#x7528;&#x66f2;&#x7ebf;&#x663e;&#x793a;&#x98de;&#x7ebf;&#xff1a; &#x63a7;&#x5236;&#x98de;&#x7ebf;&#x662f;&#x  
&#x9f20;&#x6807;&#x62d6;&#x52a8;&#x5e03;&#x7ebf;&#x884c;&#x4e3a;&#xff1a; &#x63a7;&#x5236;&#x4f7f;&#x7528;&#x  
&#x5c06;&#x72ec;&#x7acb;&#x4e8e;&#x4efb;&#x4f55;&#x5176;&#x4ed6;&#x5e03;&#x7ebf;&#x6bb5;&#x79fb;&#x52a8;&#x5e  
"&#x62d6;&#x52a8;(45 &#x5ea6;&#x6a21;&#x5f0f;)" &#x5c06;&#x8c03;&#x7528;&#x63a8;&#x6324;&#x5f0f;&#x5e03;&#x7ebf  
"&#x62d6;&#x52a8;(&#x81ea;&#x7531;&#x89d2;&#x5ea6;)" &#x5c06;&#x79fb;&#x52a8;&#x5e03;&#x7ebf;&#x6bb5;&#x6700;&  
&#x5c06;&#x56fe;&#x5f62;&#x7ebf;&#x6761;&#x9650;&#x5236;&#x4e3a; H&#x3001;V &#x548c;45&#x5ea6;&#xff1a;  
&#x63a7;&#x5236;&#x4f7f;&#x7528;&#x56fe;&#x5f62;&#x7ed8;&#x5236;&#x5de5;&#x5177;&#x7ed8;&#x5236;&#x7684;&#xs  
&#x8bf7;&#x6ce8;&#x610f;&#xff0c;&#x8fd9;&#x4ec5;&#x5f71;&#x54cd;&#x7ed8;&#x5236;&#x65b0;&#x7684;&#x5e03;&#xe  
&#x663e;&#x793a;&#x9875;&#x9762;&#x9650;&#x5236;&#xff1a; &#x63a7;&#x5236;&#x9875;&#x9762;&#x8fb9;&#x754c;&#  
&#x8986;&#x94dc;&#x5c5e;&#x6027;&#x540e;&#x91cd;&#x65b0;&#x586b;&#x5145;&#x8986;&#x94dc;&#x5bf9;&#x8bdd;  
&#x63a7;&#x5236;&#x7f16;&#x8f91;&#x4efb;&#x4f55;&#x8986;&#x94dc;&#x7684;&#x5c5e;&#x6027;&#x540e;&#x662f;&#x54  
&#x53ef;&#x4ee5;&#x5728;&#x590d;&#x6742;&#x7684;&#x8bbe;&#x8ba1;&#x6216;&#x901f;&#x5ea6;&#x8f83;&#x6162;&#x7

### 8.1.3 $\epsilon$ ;



Pcbnew &#x652f;&#x6301;&#x5728;&#x4e0d;&#x540c;&#x7684;&#x989c;&#x8272;&#x4e3b;&#x9898;&#x4e4b;&#x95f4;&#x526.0 &#x6709;&#x4e24;&#x4e2a;&#x5185;&#x7f6e;&#x7684;&#x989c;&#x8272;&#x4e3b;&#x9898;&#xff1a;"KiCad &#x9ed8;&#x&#x662f;&#x4e00;&#x4e2a;&#x65b0;&#x4e3b;&#x9898;&#xff0c;&#x8bbe;&#x8ba1;&#x7528;&#x4e8e;&#x5927;&#x591a;&#x6&#x7ecf;&#x5178;&#x7248;" &#x662f; KiCad 5.1 &#x53ca;&#x66f4;&#x65e9;&#x7248;&#x672c;&#x7684;&#x9ed8;&#x8ba4;&#xPcbnew &#x7684;&#x5916;&#x89c2;&#xff0c;&#x4e5f;&#x53ef;&#x4ee5;&#x5b89;&#x88c5;&#x5176;&#x4ed6;&#x7528;&#x623&#x989c;&#x8272;&#x4e3b;&#x9898;&#x5b58;&#x50a8;&#x5728;&#x4f4d;&#x4e8e; KiCad &#x914d;&#x7f6e;&#x76ee;&#x5f5Colors &#x5b50;&#x76ee;&#x5f55;&#x4e2d;&#x7684; JSON &#x6587;&#x4ef6;&#x4e2d;&#x3002;"&#x6253;&#x5f00;&#x4e3KiCad&#x3002;&#x5982;&#x679c;&#x6587;&#x4ef6;&#x662f;&#x6709;&#x6548;&#x7684;&#x989c;&#x8272;&#x4e3b;&#x989&#x8981;&#x521b;&#x5efa;&#x4e00;&#x4e2a;&#x65b0;&#x7684;&#x989c;&#x8272;&#x4e3b;&#x9898;&#xff0c;&#x4ece;&#x9&#x4e3a;&#x4f60;&#x7684;&#x4e3b;&#x9898;&#x8f93;&#x5165;&#x4e00;&#x4e2a;&#x540d;&#x79f0;&#xff0c;&#x7136;&#x54&#x65b0;&#x4e3b;&#x9898;&#x4e2d;&#x7684;&#x989c;&#x8272;&#x5c06;&#x4ece;&#x4f60;&#x521b;&#x5efa;&#x65b0;&#x4&#x8981;&#x66f4;&#x6539;&#x989c;&#x8272;&#xff0c;&#x8bf7;&#x53cc;&#x51fb;&#x6216;&#x4e2d;&#x952e;&#x5355;&#x51&#x9ed8;&#x8ba4;"&#x989c;&#x8272;&#x4e3b;&#x9898;&#x4e2d;&#x7684;&#x76f8;&#x5e94;&#x6761;&#x76ee;&#x3002;&#x989c;&#x8272;&#x4e3b;&#x9898;&#x4f1a;&#x81ea;&#x52a8;&#x4fdd;&#x5b58;&#xff1b;&#x5f53;&#x60a8;&#x5173;&#x95

### 8.1.4 64cd;4f5c;63d2;4ef6;



KiCad PCB 8f16;8f91;5668;652f;6301;7528; Python 8f16;8f5199;7684;63d2;4ef6;8fd9;4e9b;63d2;4ef6;53ef;4ee5;4f7f;7528;5185;7f6e;7684;63d2;4ef6;54; KiCad 7ae0;8282;ff09;ff0c;6216;8005;5c06;63d2;4ef6;6587;4ef6;653e;5728; 8be6;89c1;4e0b;9762;7684;811a;672c;90e8;5206;3002;

6bcf;4e2a;88ab;68c0;6d4b;5230;7684;63d2;4ef6;90fd;4f1a;5728;8fd9;4e 63d2;4ef6;53ef;4ee5;5728; PCB 8f16;8f91;5668;7684;9876;90e8;5de5;5177; 5982;679c;4e00;4e2a;63d2;4ef6;7684; "663e;793a;6309;94ae;" 63a7;5236; 5de5;5177;" > "5916;90e8;63d2;4ef6;" 83dc;5355;4e2d;8bbf;95ee;3002;

5217;8868;5e95;90e8;7684;7bad;5934;63a7;5236;5141;8bb8;6539;53d8; 6587;4ef6;5939;6309;94ae;5c06;542f;52a8;4e00;4e2a;6587;4ef6;8d44;6 5237;65b0;6309;94ae;5c06;626b;63cf;63d2;4ef6;6587;4ef6;5939;4e2d;6

### 8.1.5 539f;70b9;548c;8f74;



663e;793a;539f;70b9;ff1a; 51b3;5b9a;5728;8f16;8f91;753b;5e03;4e2d;8 9875;9762;539f;70b9;56fa;5b9a;5728;9875;9762;7684;89d2;843d;3002; 7528;6237;53ef;4ee5;79fb;52a8;94bb;53d6;/653e;7f6e;6587;4ef6;539f;7

X 8f74;ff1a; 63a7;5236; X 5750;6807;5411;53f3;8fd8;662f;5411;5de6;589e;

Y 8f74;ff1a; 63a7;5236; Y 5750;6807;662f;5411;4e0a;8fd8;662f;5411;4e0b;

## 8.2

KiCad's custom design rule system allows creating design rules that are more specific than the generic rules available in the Constraints page of the Board Setup dialog. Custom design rules have many applications, but in general they are used to apply certain rules to a portion of the board, such as a specific net or netclass, a specific area, or a specific footprint.

```
&#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x5b58;&#x50a8;&#x5728;&#x4e00;&#x4e2a;&#x6269;&#x  
kicad_dra &#x7684;&#x5355;&#x72ec;&#x6587;&#x4ef6;&#x4e2d;&#x3002;&#x5f53;&#x60a8;&#x5f00;&#x59cb;&#x5411;&#x  
kicad_dra &#x6587;&#x4ef6;&#x4e0e; kicad_pcb &#x548c; kicad_pro &#x6587;&#x4ef6;&#x4e00;&#x8d77;&#x4fdd;&#x
```

### Note

```
kicad_dra&#x6587;&#x4ef6;&#x7531; KiCad&#x81ea;&#x52a8;&#x7ba1;&#x7406;&#xff0c;&#x4e0d;&#x5e94;&#x47f7;&#x7528;&#x59cb;&#x7ec8;&#x47f7;&#x7528;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#x5bf9;&#x8bdd;&#x6846;&#x7684;&#x81ea;&#x
```

### 8.2.1 **8.2.1**

&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x7f16;&#x8f91;&#x5668;&#x4f4d;&#x4e8e;&#x7535;&#x8def;&#x677f;&#x8b  
&#x6700;&#x597d;&#x5728;&#x7f16;&#x8f91;&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x540e;&#x4f7f;&#x7528;  
&#x68c0;&#x67e5;&#x89c4;&#x5219;&#x8bed;&#x6cd5;&#x6309;&#x94ae;&#xff0c;&#x4ee5;&#x786e;&#x4fdd;&#x6ca1;&#x

## 8.2.2 `&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x8bed;&#x6cd5;`

&#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x8bed;&#x8a00;&#x57fa;&#x4e8e; s &#x8868;&#x8fbe;&#x  
&#x6761;&#x4ef6;&#xff0c;&#x4ee5;&#x53ca;&#x4e00;&#x4e2a;&#x5b9a;&#x4e49;&#x8981;&#x5e94;&#x7528;&#x4e8e;&#x5  
&#x7ea6;&#x675f;&#x3002;

&#x8be5;&#x8bed;&#x8a00;&#x4f7f;&#x7528;&#x5706;&#x62ec;&#x53f7; ( ( &#x548c; ) ) &#x6765;&#x5b9a;&#x4e49;&#x76f8  
( &#x5fc5;&#x987b;&#x6709;&#x5339;&#x914d;&#x7684; ) &#x3002;&#x5728;&#x5b50;&#x53e5;&#x4e2d;&#xff0c;&#x6807;&#x6216;  
" &#x6216; ' &#x5f15;&#x8d77;&#x6765;&#xff0c;&#x4e5f;&#x53ef;&#x4ee5;&#x4e0d;&#x52a0;&#x5f15;&#x53f7;&#x3002;&#x4e00;  
" &#x4f5c;&#x4e3a;&#x5916;&#x5f15;&#x53f7;&#x5b57;&#x7b26;&#xff0c;&#x4f7f;&#x7528; ' &#x4f5c;&#x4e3a;&#x5185;&#x4e00;  
( &#x53cd;&#x4e4b;&#x4ea6;&#x7136; ) &#xff0c;&#x4ece;&#x800c;&#x5b9e;&#x73b0;&#x5355;&#x5c42;&#x5d4c;&#x5957;&#x4e00;

```
&#x5728;&#x4e0b;&#x9762;&#x7684;&#x8bed;&#x6cd5;&#x63cf;&#x8ff0;&#x4e2d;&#xff0c;< &#x5c16;&#x62ec;&#x53f7
> &#x4e2d;&#x7684;&#x9879;&#x8868;&#x793a;&#x5fc5;&#x987b;&#x5b58;&#x5728;&#x7684;&#x6807;&#x8bb0;&#xff0c;[
&#x65b9;&#x62ec;&#x53f7; ]&#x4e2d;&#x7684;&#x9879;&#x8868;&#x793a;&#x53ef;&#x9009;&#x6216;&#x4ec5;&#x6
```

&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x6587;&#x4ef6;&#x5fc5;&#x987b;&#x4ee5;&#x5b9a;&#x4e49;&#x89c4;&#x5  
&#x4ece; KiCad 6.0 &#x5f00;&#x59cb;&#xff0c;&#x7248;&#x672c;&#x662f; 1&#x3002; &#x7248;&#x672c;&#x5934;&#x7684;&#x  
(version<number>) &#x3002;&#x56e0;&#x6b64;&#xff0c;&#x5728; KiCad 6.0 &#x4e2d;&#xff0c;&#x6807;&#x9898;&#x5e94;

```
(version 1)
```

&#x5728;&#x7248;&#x672c;&#x6807;&#x9898;&#x4e4b;&#x540e;&#xff0c;&#x60a8;&#x53ef;&#x4ee5;&#x8f93;&#x5165;&#x4c  
&#x4f8b;&#x5982;&#xff0c;&#x5982;&#x679c;&#x60a8;&#x521b;&#x5efa;&#x4e00;&#x6761;&#x89c4;&#x5219;&#x6765;&#x9  
HV &#x4e2d;&#x7684;&#x5e03;&#x7ebf;&#x4e0e;&#x4efb;&#x4f55;&#x5176;&#x4ed6;&#x7f51;&#x7edc;&#x4e2d;&#x7684;&#  
HV &#x7f51;&#x7edc;&#x4e2d;&#x7684;&#x5e03;&#x7ebf;&#x843d;&#x5728;&#x89c4;&#x5219;&#x533a;&#x57df;&#x5185;&

```
&#x6b6f;&#x6761;&#x89c4;&#x5219;&#x5fc5;&#x987b;&#x6709;&#x4e00;&#x4e2a;&#x540d;&#x79f0;&#x548c;&#x4e00;&#x4e2a;&#x7ea6;&#x675f; (constraint) &#x5b50;&#x53e5;&#x3002;&#x8be5;&#x540d;&#x79f0;&#x53ef;&#x4ee5;&#x662f;&#x5b9a;&#x62a5;&#x544a;&#x4e2d;&#x5f15;&#x7528;&#x8be5;&#x89c4;&#x5219;&#x3002;&#x7ea6;&#x675f; (constraint) &#x5b9a;&#x4e49;&#x4e86;&#x89c4;&#x5219;&#x7684;&#x884c;&#x4e3a;&#x3002;&#x89c4;&#x5219;&#x8fd8;&#x53ef;&#x4e2a;&#x6761;&#x4ef6; (condition) &#x5b50;&#x53e5;&#xff0c;&#x51b3;&#x5b9a;&#x54ea;&#x4e9b;&#x5bf9;&#x8c61;&#x5c42; (layer) &#x5b50;&#x53e5;&#xff0c;&#x6307;&#x5b9a;&#x8be5;&#x89c4;&#x5219;&#x9002;&#x7528;&#x4e8e;&
```





&#x4f8b;&#x5982;&#xff0c;&#x5f53;&#x68c0;&#x67e5;&#x94dc;&#x5bf9;&#x8c61;&#x4e4b;&#x95f4;&#x7684;&#x95f4;&#x96f4;&#x5982;&#x679c;&#x5b58;&#x5728;&#x4e00;&#x4e2a;&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#xff0c;&#x5176;&#x88ab;&#x6d4b;&#x5bf9;&#x8c61;&#x5728;&#x8868;&#x8fbc;&#x5f0f;&#x8bed;&#x8a00;&#x4e2d;&#x79f0;&#x4e3a;  
A &#x548c; B&#x3002;&#x8fd9;&#x4e24;&#x4e2a;&#x5bf9;&#x8c61;&#x7684;&#x987a;&#x5e8f;&#x5e76;&#x4e0d;&#x91cd;&#x4e3a;&#x5e03;&#x7ebf;&#xff0c;B &#x4e3a;&#x8fc7;&#x5b54;&#x3002; &#x6709;&#x4e00;&#x4e9b;&#x8868;&#x8fbc;&#x4f5c;&#x4e3a;&#x5bf9;&#x8c61;&#x540d;&#x3002;  
&#x6761;&#x4ef6;&#x4e2d;&#x7684;&#x8868;&#x8fbc;&#x5f0f;&#x5fc5;&#x987b;&#x89e3;&#x6790;&#x4e3a;&#x5e03;&#x5c77f; (true &#x6216; false )&#x3002;&#x5982;&#x679c;&#x8868;&#x8fbc;&#x5f0f;&#x89e3;&#x6790;&#x4e3a; true&#xff0c;&#x5219;&#x89c4;&#x5219;&#x5e94;&#x7528;&#x4e8e;&#x7ed9;&#x5b9a;&#x7684;&#x5bf9;&#x8c61;&#x3002;  
&#x6bcf;&#x4e2a;&#x88ab;&#x6d4b;&#x5bf9;&#x8c61;&#x90fd;&#x6709;&#x53ef;&#x4ee5;&#x6bd4;&#x8f83;&#x7684;  
&#x5c5e;&#x6027;&#xff0c;&#x4ee5;&#x53ca;&#x53ef;&#x4ee5;&#x6267;&#x884c;&#x7279;&#x5b9a;&#x6d4b;&#x8bd5;&#x7684;  
&#x51fd;&#x6570;&#x3002;&#x5c5e;&#x6027;&#x548c;&#x51fd;&#x6570;&#x7684;&#x4f7f;&#x7528;&#x8bed;&#x6cd5;&#x5c77f; <object> .<property> &#x548c; <object> .<function> ( [arguments] ) &#x3002;&#x8bd1;&#x8005;&#x6ce8;&#xff1a; <&#x5bf9; &#x8c61; > .<&#x5c5e; &#x6027; > &#x548c; <&#x5bf9; &#x8c61; > .<&#x51fd; &#x6570; > ( [ &#x53c2

**Note**

&#x5f53;&#x60a8;&#x5728;&#x6587;&#x672c;&#x7f16;&#x8f91;&#x5668; ( A.&#x3001;B.&#x6216; AB. )  
&#x4e2d;&#x95e2;&#x5165; <&#x5bf9; &#x8c61; (object) > . &#x65f6;&#xff0c;&#x4f1a;&#x51fa;&#x73b0;&#x4e00;&#x4e2d;

&#x4f7f;&#x7528; &#x5e03;&#x5c14;&#x8fd0;&#x7b97;&#x7b26; &#x6bd4;&#x8f83;&#x5bf9;&#x8c61;&#x5c5e;&#x6027;&#x5c77f; C/C++ &#x8bed;&#x6cd5;&#xff0c;&#x5e76;&#x652f;&#x6301;&#x4ee5;&#x4e0b;&#x8fd0;&#x7b97;&#x7b26;&#xff1a;

|        |  |
|--------|--|
| ==     | &#x7b49;&#x4e8e;   |
| !=     | &#x4e0d;&#x7b49;&#x4e8e;   |
| >, >=  | &#x5927;&#x4e8e;&#x3001;&#x5927;&#x4e8e;&#x6216;&#x7b49;&#x4e8e; |
| <, \<= | &#x5c0f;&#x4e8e;&#x3001;&#x5c0f;&#x4e8e;&#x6216;&#x7b49;&#x4e8e; |
| &&     | &#x548c;   |
|        | &#x6216;   |

&#x4f8b;&#x5982;&#xff0c;A.NetClass == 'HV' &#x5c06;&#x9002;&#x7528;&#x4e8e;&#x4efb;&#x4f55;&#x5c5e;&#x4e8e; "HV" &#x7f51;&#x7c7b;&#x7684;&#x5bf9;&#x8c61;&#xff0c;A.NetClass != B.NetClass &#x5c06;&#x9002;&#x7528;&#x5c77f; &#x6709;&#x4e9b;&#x5c5e;&#x6027;&#x8868;&#x793a;&#x7269;&#x7406;&#x6d4b;&#x91cf;&#xff0c;&#x6bd4;&#x5982;&#x5728;&#x8fd9;&#x4e9b;&#x5c5e;&#x6027;&#x4e0a;&#xff0c;&#x5355;&#x4f4d;&#x540e;&#x7f00; &#x53ef;&#x4ee5;&#x5982;&#x679c;&#x6ca1;&#x6709;&#x4f7f;&#x7528;&#x5355;&#x4f4d;&#x540e;&#x7f00;&#xff0c;&#x5c5e;&#x6027;&#x7684;&#x652f;&#x6301;&#x4ee5;&#x4e0b;&#x540e;&#x7f00;&#xff1a;

|         |   |
|---------|---|
| mm      | &#x6beb;&#x7c73;  |
| mil, th | &#x5343;&#x5206;&#x4e4b;&#x4e00;&#x82f1;&#x5bf8; (mils) |
| in, "   | &#x82f1;&#x5bf8;  |
| deg     | &#x5ea6;  |
| rad     | &#x5f27;&#x5ea6;  |

**Note**

&#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x4e2d;&#x4f7f;&#x7528;&#x7684;&#x5355;&#x4f4d;&#x72ec;&#x7f16;&#x7f16;&#x8f91;&#x5668;&#x4e2d;&#x7684;&#x663e;&#x793a;&#x5355;&#x4f4d;&#x3002;

**8.2.2.3 &#x7ea6;&#x675f;**

&#x89c4;&#x5219;&#x7684; &#x7ea6;&#x675f; &#x5b50;&#x53e5;&#x5b9a;&#x4e49;&#x4e86;&#x89c4;&#x5219;&#x5728;&#x7ea6;&#x675f;&#x7c7b;&#x578b; &#x548c;&#x4e00;&#x4e2a;&#x6216;&#x591a;&#x4e2a;&#x8bbe;&#x7f6e;&#x7ea6;&#x5c77f;

```
(clearance) (trace_width)
)&
```

```
&
&
"min/opt/max" &
DRC &
KiCad &
diff_pair_gap &
&
&
(constraint track_width (min 0.5mm) (opt 0.5mm) (max 1.0mm)) &
`&
```

| Constraint type     | Argument type   | Description   |
|---------------------|---|---|
| annular_width       | min/opt/max   | Checks the width of annular rings on vias.  |
| clearance           | min   | Checks the clearance between copper objects of different nets. KiCad's design rule system does not permit constraining clearance between objects on the same net at this time. To allow copper objects to overlap (collide), create a <code>clearance</code> constraint with the <code>min</code> value less than zero (for example, <code>-1</code> ).   |
| courtyard_clearance | min   | Checks the clearance between footprint courtyards and generates an error if any two courtyards are closer than the <code>min</code> distance. If a footprint does not have a courtyard shape, no errors will be generated from this constraint.   |
| diff_pair_gap       | min/opt/max   | Checks the gap between coupled tracks in a differential pair. Coupled tracks are segments that are parallel to each other. Differential pair gap is not tested on uncoupled portions of a differential pair (for example, the fanout from a component).   |
| diff_pair_uncoupled | max   | Checks the distance that a differential pair track is routed uncoupled from the other polarity track in the pair (for example, where the pair fans out from a component, or becomes uncoupled to pass around another object such as a via).   |
| disallow            | track + via + micro_via + buried_via + pad + zone + text + graphic + hole + footprint | Specify one or more object types to disallow, separated by spaces. For example, <code>(constraint disallow track)</code> or <code>(constraint disallow track via pad)</code> . If an object of this type matches the rule condition, a DRC error will be created. This constraint is essentially the same as a keepout rule area, but can be used to create more specific keepout restrictions. |
| edge_clearance      | min/opt/max   | Checks the clearance between objects and graphical items on the <code>Edge.Cuts</code> layer (the board outline, as well as any board cutouts or slots defined on that layer).  |
| length              | min/max   | Checks the total routed length for the nets that match the rule condition and generates an error for each net that is below the <code>min</code> value (if specified) or above the <code>max</code> value (if specified) of the constraint.   |
| hole                | min/max   | Checks the size (diameter) of a drilled hole in a pad or via. For oval holes, the smaller (minor) diameter will be tested against the <code>min</code> value (if specified) and the larger (major) diameter will be tested against the <code>max</code> value (if specified).   |
| hole_clearance      | min   | Checks the clearance between a drilled hole in a pad or via and copper objects on a different net. The clearance is measured from the diameter of the hole, not its center.   |

| Constraint type | Argument type | Description  |
|-----------------|---------------|--|
| hole_to_hole    | min           | Checks the clearance between mechanically-drilled holes in pads and vias. The clearance is measured between the diameters of the holes, not between their centers. HDI vias (microvias, blind vias, and buried vias) are not tested by this constraint.  |
| silk_clearance  | min/opt/max   | Checks the clearance between objects on silkscreen layers and other objects.   |
| skew            | max           | Checks the total skew for the nets that match the rule condition, that is, the difference between the length of each net and the average of all the lengths of each net that is matched by the rule. If the absolute value of the difference between that average and the length of any one net is above the constraint max value, an error will be generated. |
| track_width     | min/opt/max   | Checks the width of track and arc segments. An error will be generated for each segment that has a width below the min value (if specified) or above the max value (if specified).   |
| via_count       | max           | Counts the number of vias on every net matched by the rule condition. If that number exceeds the constraint max value on any matched net, an error will be generated for that net.   |

### 8.2.3

PCB

#### 8.2.3.1

PCB

| Layer      | string    |  |
|------------|-----------|--|
| Locked     | boolean   |  |
| Parent     | string    |  |
| Position_X | dimension |  |
| Position_Y | dimension |  |
| Type       | string    |  |

**8.2.3.2 &#x8fde;&#x63a5;&#x7684;&#x5bf9;&#x8c61;&#x5c5e;&#x6027;**

&#x8fd9;&#x4e9b;&#x5c5e;&#x6027;&#x9002;&#x7528;&#x4e8e;&#x53ef;&#x4ee5;&#x5206;&#x914d;&#x7f51;&#x7684;&#x9

| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x627f;&#x8f7b; |         |  |
|--|---------|--|
| Net  | integer | &#x94dc;&#x5bf9;&#x8c61;&#x7684;&#x7f51;&#x7edc;&#x7801;&#x3002;&#x8bf7;&#x6ce8;&#x610f;&#xff0c;&#x4e0d;&#x80fd;&#x4f9d;&#x8d56;&#x7f51;&#x7684;&#x53ef;&#x4ee5;&#x7528;&#x6765;&#x6bd4;&#x8f83;&#x4e24;&#x4e2a;&#x6027;&#x4f8b;&#x5982; A.Net == B.Net &#x6bd4; A.NetName == B.NetName &#x5feb;&#x3002; |
| NetClass   | string  | &#x94dc;&#x5bf9;&#x8c61;&#x7684;&#x7f51;&#x7edc;&#x7c7b;&#x7684;&#x540d;&#x3002;   |
| NetName  | string  | &#x94dc;&#x5bf9;&#x8c61;&#x7684;&#x7f51;&#x7edc;&#x540d;&#x79f0;&#x3002;   |

**8.2.3.3 &#x5c01;&#x88c5;&#x5c5e;&#x6027;**

These properties apply to footprints.

| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x627f;&#x8f7b; |           |  |
|--|-----------|--|
| Clearance_Overdimension                          | dimension | &#x4e3a;&#x5c01;&#x88c5;&#x8bbe;&#x7f6e;&#x7684;&#x94dc;&#x95f4;&#x9699;&#x3002;                       |
| Orientation                                      | double    | &#x5c01;&#x88c5;&#x7684;&#x65b9;&#x5411; (&#x65cb;&#x8f6c;) (&#x5355;&#x4f4d;&#xff1a;&#x5ea6;)&#x3002; |
| Reference  | string    | &#x5c01;&#x88c5;&#x7684;&#x4f4d;&#x53f7;&#x3002;   |
| Solderpaste_MaterialOverride                     | dimension | &#x4e3a;&#x5c01;&#x88c5;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x8fb9;&#x3002;                       |
| Solderpaste_MaterialRatioOverride                | dimension | &#x4e3a;&#x5c01;&#x88c5;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x4f59;&#x3002;                       |
| Thermal_ReliefDimension                          | dimension | &#x4e3a;&#x5c01;&#x8f6c;&#x8bbe;&#x7f6e;&#x7684;&#x6563;&#x70ed;&#x95f4;&#x3002;                       |
| Thermal_ReliefDimension                          | dimension | &#x4e3a;&#x5c01;&#x88c5;&#x8bbe;&#x7f6e;&#x7684;&#x6563;&#x70ed;&#x8fde;&#x3002;                       |
| Value  | string    | &#x5c01;&#x88c5;&#x7684; "&#x503c;" &#x5b57;&#x6bb5;&#x7684;&#x5185;&#x5bb9;&#x3002;                   |

**8.2.3.4 &#x710a;&#x76d8;&#x5c5e;&#x6027;**

These properties apply to footprint pads.

| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x627f;&#x8f7b; |           |  |
|--|-----------|--|
| Clearance_Overdimension                          | dimension | &#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x94dc;&#x95f4;&#x9699;&#x3002;   |
| Fabrication_Property                             | string    | "&#x65e0;"&#x3001;"BGA &#x710a;&#x76d8;"&#x3001;"&#x57fa;&#x51c6;,&#x5168;&#x5c40;&#x5230;&#x7535;&#x8def;&#x677f;"&#x3001;"&#x57fa;&#x51c6;,&#x672c;&#x5730;&#x5230;&#x5c01;&#x88c5;"&#x3001;"&#x6d4b;&#x8bd5;&#x70b5;"&#x6563;&#x70ed;&#x7247;&#x710a;&#x76d8;"&#x3001;"&#x8702;&#x7a9d;&#x72b5;&#x4e4b;&#x4e00;&#x3002; |
| Hole_Size_X                                      | dimension | &#x710a;&#x76d8;&#x5728; X &#x8f74;&#x4e0a;&#x7684;&#x901a;&#x5b54;/&#x69fd;&#x7684;&#x5927;&#x5c0f;&#x3002;   |
| Hole_Size_Y                                      | dimension | &#x710a;&#x76d8;&#x5728; Y &#x8f74;&#x4e0a;&#x7684;&#x901a;&#x5b54;/&#x69fd;&#x7684;&#x5927;&#x5c0f;&#x3002;   |
| Orientation                                      | double    | &#x710a;&#x76d8;&#x7684;&#x65b9;&#x5411; (&#x65cb;&#x8f6c;) (&#x5355;&#x4f4d;&#xff1a;&#x5ea6;)&#x3002;   |
| Pad_Number                                       | string    | &#x710a;&#x76d8;&#x7684; "&#x7f16;&#x53f7;"&#xff0c;&#x53ef;&#x4ee5;&#x662f;&#x5b57;&#x7b26;&#x4e32; (&#x4f8b;&#x5982;&#xff0c;BGA &#x4e2d;&#x7684; "A1")&#x3002;  |
| Pad_To_Die_Lengdimension                         | dimension | &#x710a;&#x76d8;&#x7684; "&#x710a;&#x76d8;&#x5230;&#x82af;&#x7247;&#x957f;&#x5ea6;" &#x5c5e;&#x6027;&#x7684;&#x503c;&#xff0c;&#x5b83;&#x662f;&#x5728;&#x8ba1;&#x3002;   |

|                       |                    |  |
|-----------------------|--------------------|--|
| Pad_Type              | string             | "&#x901a;&#x5b54;"&#x3001;"&#x8d34;&#x7247;"&#x3001;"&#x677f;&#x8fb9;&#x8f7b;"&#x6216;<br>"&#x975e;&#x5bfc;&#x901a;&#x5b54;&#xff0c;&#x673a;&#x68b0;"&#x4e4b;&#x4e00;&#x3002;   |
| Pin_Name              | string             | &#x710a;&#x76d8;&#x7684;&#x540d;&#x79f0;<br>(&#x901a;&#x5e38;&#x662f;&#x539f;&#x7406;&#x56fe;&#x4e2d;&#x76f8;&#x5e94;&#x6216;  |
| Pin_Type              | string             | &#x710a;&#x76d8;&#x7684;&#x7535;&#x6c14;&#x7c7b;&#x578b;<br>(&#x901a;&#x5e38;&#x53d6;&#x81ea;&#x539f;&#x7406;&#x56fe;&#x4e2d;&#x76f8;&#x8f93;&#x5165;"&#x3001;"&#x8f93;&#x51fa;"&#x3001;"&#x53cc;&#x5411;"&#x3001;"&#x7535;&#x6e90;&#x8f93;&#x5165;"&#x3001;"&#x7535;&#x6e90;&#x8f93;&#x51fa;"&#x6216;"&#x672a;&#x8fde;&#x63a5;"&#x4e4b;&#x4e00;&#x3002; |
| Round_Radius          | double             | &#x5bf9;&#x4e8e;&#x5706;&#x5f62;&#x77e9;&#x5f62;&#x710a;&#x76d8;&#xff0c;&#x4e00;&#x3002;   |
| Shape                 | string             | "&#x5706;&#x5f62;"&#x3001;"&#x77e9;&#x5f62;"&#x3001;"&#x692d;&#x5706;&#x5f62;"&#x81ea;&#x5b9a;&#x4e49;"&#x4e4b;&#x4e00;&#x3002;  |
| Size_X                | dimension          | &#x710a;&#x76d8;&#x5728; X<br>&#x8f74;&#x4e0a;&#x7684;&#x5927;&#x5c0f;&#x3002;   |
| Size_Y                | dimension          | &#x710a;&#x76d8;&#x5728; Y<br>&#x8f74;&#x4e0a;&#x7684;&#x5927;&#x5c0f;&#x3002;   |
| Soldermask_Margin     | dimension override | &#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x963b;&#x710a;&#x8fb9;&#x8f7b;"&#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x8fb9;&#x8f7b;"&#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x8fb9;&#x8f7b;"  |
| Solderpaste_Margin    | dimension override | &#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x8fb9;&#x8f7b;"&#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x8fb9;&#x8f7b;"   |
| Solderpaste_MaskRatio | dimension override | &#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x710a;&#x818f;&#x8fb9;&#x8f7b;"&#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x6563;&#x70ed;&#x95f4;&#x8fde;"   |
| Thermal_Relief        | dimension          | &#x4e3a;&#x710a;&#x76d8;&#x8bbe;&#x7f6e;&#x7684;&#x6563;&#x70ed;&#x8fde;&#x8f7b;"  |

### 8.2.3.5 `&#x5e03; &#x7ebf; &#x548c; &#x5706; &#x5f27; &#x5c5e; &#x6027;`

These properties apply to tracks and arc tracks.

|  |           |  |
|--|-----------|--|
| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x627b;&#x687b; |           |  |
| Origin_X   | dimension | &#x8d77;&#x70b9;&#x7684; X &#x5750;&#x6807;&#x3002;                      |
| Origin_Y   | dimension | &#x8d77;&#x70b9;&#x7684; Y &#x5750;&#x6807;&#x3002;                      |
| End_X  | dimension | &#x7ec8;&#x70b9;&#x7684; X &#x5750;&#x6807;&#x3002;                      |
| End_Y  | dimension | &#x7ec8;&#x70b9;&#x7684; Y &#x5750;&#x6807;&#x3002;                      |
| Width  | dimension | &#x5e03;&#x7ebf;&#x6216;&#x5706;&#x5f27;&#x7684;&#x5bbd;&#x5ea6;&#x3002; |

#### 8.2.3.6 $\frac{1}{2} \times 10^5$

These properties apply to vias.

|  |   |
|--|---|
| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x627b;&#x6870; |   |
| Diameter   | dimension<br>&#x8fc7;&#x5b54;&#x710a;&#x76d8;&#x7684;&#x76f4;&#x5f84;&#x3002;   |
| Drill  | dimension<br>&#x8fc7;&#x5b54;&#x6210;&#x54c1;&#x901a;&#x5b54;&#x7684;&#x76f4;&#x5f84;&  |
| Layer_Bottom                                     | string<br>&#x8fc7;&#x5b54;&#x538b;&#x5c42;&#x4e2d;&#x7684;&#x6700;&#x540e;&#x4e00;&   |
| Layer_Top  | string<br>&#x8fc7;&#x5b54;&#x538b;&#x5c42;&#x4e2d;&#x7684;&#x7b2c;&#x4e00;&#x5c42;&   |
| Via_Type   | string<br>"&#x901a;&#x5b54;"&#x3001;"&#x76f2;&#x5b54;/&#x57cb;&#x5b54;"<br>&#x6216;"&#x5fae;&#x5b54;"&#x4e4b;&#x4e00;&#x3002; |

8.2.3.7 &#x8986;&#x94dc;&#x548c;&#x89c4;&#x5219;&#x533a;&#x57df;&#x5c5e;&#x6027;

&#x8fd9;&#x4e9b;&#x5c5e;&#x6027;&#x9002;&#x7528;&#x4e8e;&#x94dc;&#x533a;&#x548c;&#x975e;&#x94dc;&#x533a;&#xf

|  |   |
|--|---|
| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x527b;&#x6873; |   |
| Clearance Overdimension                          | &#x4e3a;&#x8986;&#x94dc;&#x8bbe;&#x7f6e;&#x7684;&#x94dc;&#x95f4;&#x9699;& |

| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x6271;&#x670b; |           |  |
|--|-----------|--|
| Min_Width  | dimension | &#x8986;&#x94dc;&#x4e2d;&#x5141;&#x8bb8;&#x7684;&#x586b;&#x5145;&#x533a;   |
| Name   | string    | &#x7528;&#x6237;&#x6307;&#x5b9a;&#x7684;&#x540d;&#x79f0;<br>(&#x9ed8;&#x8ba4;&#x60c5;&#x51b5;&#x4e0b;&#x4e3a;&#x7a7a;)&#x3002; |
| Pad_Connections                                  | string    | "&#x7ee7;&#x627f;"&#x3001;"&#x65e0;"&#x3001;"&#x710a;&#x76d8;&#x6563;&#x70ed;&#x4e4b;&#x4e00;                                  |
| Priority   | int       | &#x8986;&#x94dc;&#x7684;&#x4f18;&#x5148;&#x7ea7;&#x522b;&#x3002;   |
| Thermal_Relief                                   | dimension | &#x4e3a;&#x8986;&#x94dc;&#x8bbe;&#x7f6e;&#x7684;&#x6563;&#x70ed;&#x95f4;&#x3002;   |
| Thermal_Relief                                   | dimension | &#x4e3a;&#x8986;&#x94dc;&#x8bbe;&#x7f6e;&#x7684;&#x6563;&#x70ed;&#x8fde;&#x3002;   |

### 8.2.3.8 &#x56fe;&#x5f62;&#x5f62;&#x72b6;&#x5c5e;&#x6027;

&#x8fd9;&#x4e9b;&#x5c5e;&#x6027;&#x9002;&#x7528;&#x4e8e;&#x56fe;&#x5f62;&#x7ebf;&#x3001;&#x5706;&#x5f27;&#x3002;

| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x6271;&#x670b; |           |  |
|--|-----------|--|
| End_X  | dimension | &#x7ec8;&#x70b9;&#x7684; X &#x5750;&#x6807;&#x3002;              |
| End_Y  | dimension | &#x7ec8;&#x70b9;&#x7684; Y &#x5750;&#x6807;&#x3002;              |
| Thickness  | dimension | &#x5f62;&#x72b6;&#x753b;&#x7b14;&#x7684;&#x7c97;&#x7ec6;&#x3002; |

### 8.2.3.9 &#x6587;&#x672c;&#x5c5e;&#x6027;

&#x8fd9;&#x4e9b;&#x5c5e;&#x6027;&#x9002;&#x7528;&#x4e8e;&#x6587;&#x672c;&#x5bf9;&#x8c61;&#xff08;&#x5c01;&#x8fde;&#x3002;

| &#x5c5e;&#x6027;&#x6570;&#x636e;&#x6271;&#x670b; |           |   |
|--|-----------|---|
| Bold   | boolean   | &#x5982;&#x679c;&#x6587;&#x672c;&#x4e3a;&#x7c97;&#x4f53;&#xff0c;&#x5219;&#x3002;  |
| Height   | dimension | &#x5b57;&#x4f53;&#x4e2d;&#x5b57;&#x7b26;&#x7684;&#x9ad8;&#x5ea6;&#x3002;  |
| Horizontal_Justification                         | string    | &#x6c34;&#x5e73;&#x6587;&#x672c;&#x5bf9;&#x9f50;<br>(&#x5bf9;&#x9f50;)&#xff1a;"&#x5411;&#x5de6;&#x5bf9;&#x9f50;"&#x3001;"&#x5c41;&#x6216;"&#x5411;&#x53f3;&#x5bf9;&#x9f50;"&#x3002; |
| Italic   | boolean   | &#x5982;&#x679c;&#x6587;&#x672c;&#x4e3a;&#x659c;&#x4f53;&#xff0c;&#x5219;&#x3002;  |
| Mirrored   | boolean   | &#x5982;&#x679c;&#x6587;&#x672c;&#x4e3a;&#x955c;&#x50cf;&#xff0c;&#x5219;&#x3002;  |
| Text   | string    | &#x6587;&#x672c;&#x5bf9;&#x8c61;&#x7684;&#x5185;&#x5bb9;&#x3002;  |
| Thickness  | dimension | &#x5b57;&#x4f53;&#x7b14;&#x5212;&#x7684;&#x7c97;&#x7ec6;&#x3002;  |
| Width  | dimension | &#x5b57;&#x4f53;&#x4e2d;&#x5b57;&#x7b26;&#x7684;&#x5bbd;&#x5ea6;&#x3002;  |
| Vertical_Justification                           | string    | &#x5782;&#x76f4;&#x6587;&#x672c;&#x5bf9;&#x9f50;&#x65b9;&#x5f0f;&#xff1a;"&#x5c41;&#x6216;"&#x5411;&#x4e0b;&#x5bf9;&#x9f50;"&#x3002;   |
| Visible  | boolean   | &#x5982;&#x679c;&#x6587;&#x672c;&#x5bf9;&#x8c61;&#x53ef;&#x89c1;<br>(&#x663e;&#x793a;)&#xff0c;&#x5219;&#x4e3a; true&#x3002;   |

### 8.2.3.10 &#x8868;&#x8fbe;&#x5f0f;&#x51fd;&#x6570;

&#x53ef;&#x4ee5;&#x5bf9;&#x81ea;&#x5b9a;&#x4e49;&#x89c4;&#x5219;&#x8868;&#x8fbe;&#x5f0f;&#x4e2d;&#x7684;&#x5b57;&#x3002;

| Function                  | Objects | Description   |
|---------------------------|---------|---|
| existsOnLayer('layer_id') | A or B  | Returns true if the object exists on the given board layer. layer_id is a string containing the name of a board layer.                            |
| fromTo('x', 'y')          | A or B  | Returns true if the object exists on the copper path between the given pads. x and y are the full names of pads in the design, such as 'R1-Pad1'. |

| Function   | Objects | Description  |
|--|---------|--|
| <code>inDiffPair('x')</code>   | A or B  | Returns true if the object is part of a differential pair and the base name of the pair matches the given argument x. For example, <code>inDiffPair('/USB_')</code> or <code>inDiffPair('/USB')</code> return <code>true</code> for objects in the nets <code>/USB_P</code> and <code>/USB_N</code> . The <code>\*</code> can be used as a wildcard, so <code>inDiffPair('/USB*')</code> matches <code>/USB1_P</code> and <code>/USB1_N</code> . Note this will always return false if the given net is not a diff pair, meaning that there isn't a matching net of the opposite polarity. So, on a board with a net named <code>/USB_P</code> but no net named <code>/USB_N</code> , this function returns false. |
| <code>insideArea('x')</code>   | A or B  | Returns true if any part of the object is inside the named rule area or zone. Rule area and zone names can be set in their respective properties dialogs. If the given area is a filled copper zone, the function tests if the given object is inside any of the filled copper regions of the zone, not if the object is inside the zone's outline.  |
| <code>insideCourtyard('x')</code><br><code>insideFrontCourtyard('x')</code><br><code>insideBackCourtyard('x')</code> | A or B  | Returns true if the any part of the object is inside the courtyard of the given footprint reference. The first variant checks both the front or back courtyard and returns true if the object is inside either one; the second and third variants check a specific courtyard. The <code>\*</code> wildcard can be used in the reference: <code>insideCourtyard('R*')</code> would check all footprints with references that start with R.  |
| <code>isBlindBuriedVia()</code>  | A or B  | Returns true if the object is a blind/buried via.  |
| <code>isCoupledDiffPair()</code>   | A/B     | Returns true if the two objects being tested are part of the same differential pair but are opposite polarities. For example, returns true if A is in net <code>/USB+</code> and B is in net <code>/USB-</code> .  |
| <code>isMicroVia()</code>  | A or B  | Returns true if the object is a microvia.  |
| <code>isPlated()</code>  | A or B  | Returns true if the object is a plated hole (in a pad or via).   |
| <code>memberOf('x')</code>   | A or B  | Returns true if the object is a member of the named group x.   |

## 8.2.4 &#x81ea;&#x5b9a;&#x4e49;&#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x793a;&#x4f8b;

```
(rule RF_width
  (layer outer)
  (condition "A.NetClass == 'RF'")
  (constraint track_width (min 0.35mm) (max 0.35mm)))

(rule "BGA neckdown"
  (constraint track_width (min 0.2mm) (opt 0.25mm))
  (constraint clearance (min 0.05mm) (opt 0.08mm))
  (condition "A.insideCourtyard('U3')"))

(rule "Distance between Vias of Different Nets"
  (constraint hole_to_hole (min 0.25mm))
  (condition "A.Type == 'Via' && B.Type == 'Via' && A.Net != B.Net"))

(rule "Distance between test points"
  (constraint courtyard_clearance (min 1.5mm))
  (condition "A.Reference == 'TP*' && B.Reference == 'TP*'))

# This assumes that there is a cutout with 1mm thick lines
# &#x8fd9;&#x5047;&#x8bbe;&#x6709;&#x4e00;&#x4e2a;&#x5e26;&#x6709; 1mm ←
  &#x7c97;&#x7ec6;&#x7684;&#x6253;&#x65ad;

(rule "Clearance to cutout"
  (constraint clearance (min 0.8mm))
  (condition "A.Layer=='Edge.Cuts' && A.Thickness == 1.0mm"))

(rule "Max Drill Hole Size Mechanical"
  (constraint hole (max 6.3mm))
  (condition "A.Pad_Type == 'NPTH, mechanical'))
```



```
(rule "Max Drill Hole Size PTH"
  (constraint hole (max 6.35mm))
  (condition "A.Pad_Type == 'Through-hole'"))

# Specify an optimal gap for a particular differential pair
(rule "Clock gap"
  (condition "A.inDiffPair('/CLK')")
  (constraint diff_pair_gap (opt 0.8mm)))

# Specify a larger clearance between differential pairs and anything else
# ←
  &#x6307;&#x5b9a;&#x5dee;&#x5206;&#x5bf9;&#x4e4b;&#x95f4;&#x7684;&#x8f83;&#x5927;&#x95f4;&#x9699;
(rule "Differential pair clearance"
  (condition "A.inDiffPair('*') && !AB.isCoupledDiffPair()")
  (constraint clearance (min 1.5mm)))
```

### 8.3

&#x811a;&#x672c;&#x5141;&#x8bb8;&#x60a8;&#x4f7f;&#x7528; Python &#x8bed;&#x8a00;&#x81ea;&#x52a8;&#x6267;&#x88a8; KiCad &#x4e2d;&#x7684;&#x4efb;&#x52a1;&#x3002;&#x53ef;&#x4ee5;&#x901a;&#x8fc7; Python &#x64cd;&#x4f5c;&#x63d2;&#x5411; KiCad &#x6dfb;&#x52a0;&#x529f;&#x80fd;&#xff0c;&#x8fd9;&#x4e9b;&#x63d2;&#x4ef6;&#x53ef;&#x4ee5;&#x6dfb;&#x63d2; KiCad &#x6587;&#x4ef6;&#x4ea4;&#x4e92;&#x7684;&#x72ec;&#x7acb;&#x811a;&#x672c;&#xff0c;&#x4f8b;&#x5982;&#xff0c;&#x672c;&#x624b;&#x518c;&#x6db5;&#x76d6;&#x4e86;&#x4e00;&#x822c;&#x811a;&#x672c;&#x7f16;&#x5199;&#x6982;&#x88a8; <https://docs.kicad.org/doxygen-python/namespaces.html> &#x4e0a;&#x7684; Doxygen &#x6587;&#x6863;&#x3002; KiCad 6 &#x6216;&#x66f4;&#x65b0;&#x7248;&#x672c;&#x9700;&#x8981; Python 3 &#x6765;&#x652f;&#x6301;&#x811a;&#x672c; 2 &#x5df2;&#x4e0d;&#x518d;&#x88ab;&#x652f;&#x6301;&#x3002;

#### 8.3.1 Python

PCB &#x7f16;&#x8f91;&#x5668;&#x7684;&#x63d2;&#x4ef6;&#x811a;&#x672c;&#x53ef;&#x4ee5;&#x901a;&#x8fc7;&#x63d2;&#x6bcf;&#x4e2a;&#x63d2;&#x4ef6;&#x90fd;&#x5e94;&#x8be5;&#x5728; plugins &#x6587;&#x4ef6;&#x5939;&#x5185;&#x88a8; plugins &#x6587;&#x4ef6;&#x5939;&#x7684;&#x4f4d;&#x7f6e;&#x9ed8;&#x8ba4;&#x4e3a;&#xff1a;

| &#x5e73;&#x53f0; | &#x8def;&#x5f84;                             |
|------------------|--|
| Linux            | ~/.local/share/kicad/6.0/scripting/plugins   |
| macOS            | ~/Documents/KiCad/6.0/scripting/plugins      |
| Windows          | %HOME%\Documents\KiCad\6.0\scripting\plugins |

#### 8.3.2

##### Note


TODO&#xff1a;&#x7f16;&#x5199;&#x672c;&#x90e8;&#x5206; (&#x5982;&#x4f55;&#x5b89;&#x88c5;&#x65b0;&#x7684;&#x64cd;&#x4ef6;&#x5939;&#x5185;&#x88a8; &#x6587;&#x4ef6;&#x5939;&#x7684;&#x4f4d;&#x7f6e;&#x9ed8;&#x8ba4;&#x4e3a;&#xff1a;

#### 8.3.3

&#x5c01;&#x88c5;&#x5411;&#x5bfc;&#x662f;&#x53ef;&#x4ee5;&#x4ece;&#x5c01;&#x88c5;&#x7f16;&#x8f91;&#x5668;&#x8ba8; Python &#x811a;&#x672c;&#x7684;&#x96c6;&#x5408;&#x3002; &#x5982;&#x679c;&#x8c03;&#x7528;&#x5c01;&#x88c5;&#x5bfc;&#x5982;&#x679c;&#x63d2;&#x4ef6;&#x672a;&#x6b63;&#x786e;&#x5206;&#x53d1;&#x5230;&#x60a8;&#x7684;&#x7cfb;&#x7684; KiCad &#x6e90;&#x4ee3;&#x7801;&#x6811;&#x4e2d;&#x7684;&#x94fe;&#x63a5;&#x4e2d;&#x627e;&#x5230;&#x6700;&#x65b0;

### 8.3.4

Pcbnew comes with a built-in Python console that can be used to inspect and interact with the board. To launch the console, use

the  button in the top toolbar. The Pcbnew Python API is not automatically loaded, so to load it, type `import pcbnew` into the console. The command `pcbnew.GetBoard()` will then return a reference to the board currently loaded in Pcbnew, which can be inspected and modified through the console.

---

#### Note

TODO

---

### 8.3.5

---

#### Note

TODO

---

### 8.3.6

---

#### Note

TODO

---

## 8.4 Working With IDF Component Outlines

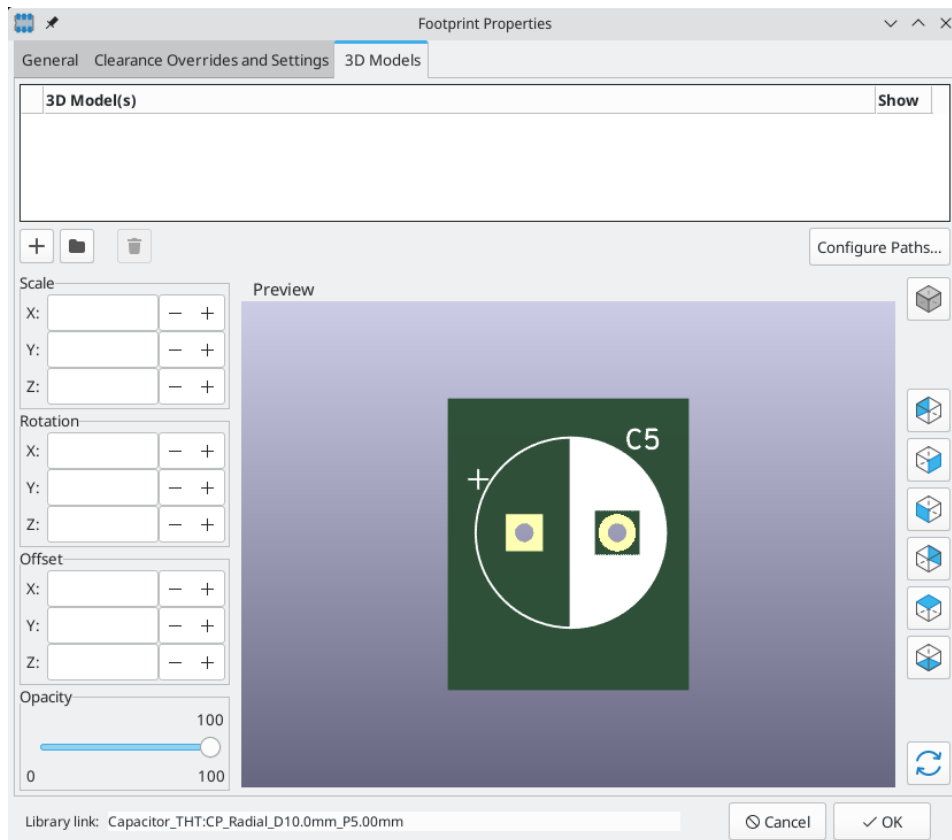
KiCad can [export an IDF representation of the board](#) for use in mechanical CAD software. Below is some guidance on attaching IDF component outlines to footprints, creating new IDF component outlines, and a description of the IDF utilities included with KiCad.

### 8.4.1 Specifying component models for use by the exporter

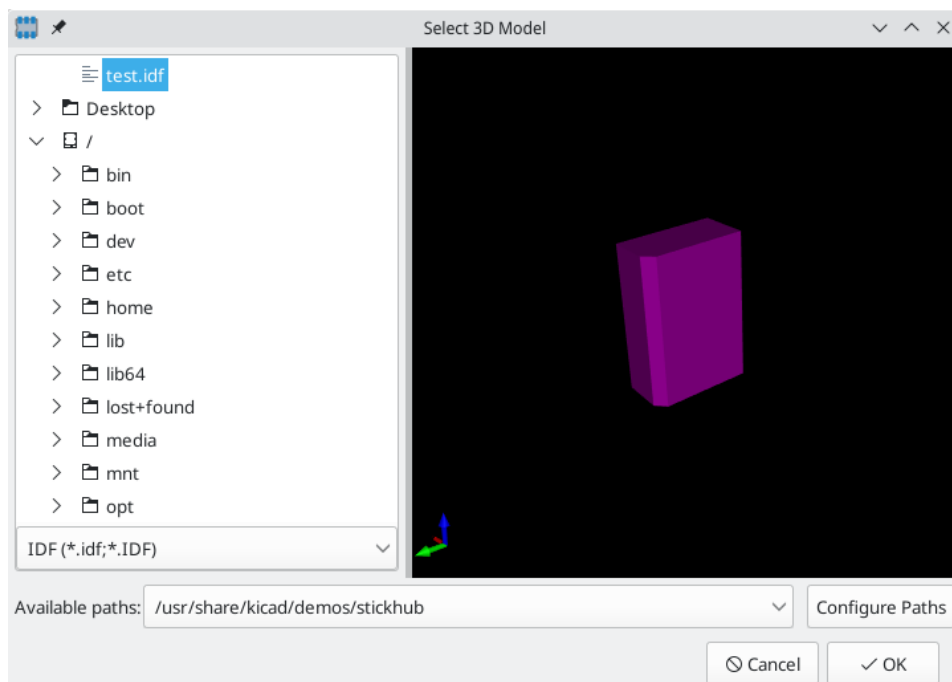
IDF component models are attached to footprints using the [footprint's 3D model properties](#). The IDF exporter uses different filetypes than the 3D viewer and other 3D model exporters, so adding 3D models for the IDF exporter does not conflict with 3D models added to a footprint for other purposes.

To add an IDF model to a footprint in the footprint or PCB editors, edit the footprint's properties and click on the 3D Models tab.

---



Click the  button and select the **IDF (\*.idf;\*.IDF)** filetype filter. Browse to the desired outline file.



Once the desired component outline file is selected, enter any necessary values for the offset and rotation. The offsets must be specified using the IDF board output units (mm or mils) and in the IDF coordinate system, which is a right-hand coordinate system with +Z pointing towards the viewer, +X to the viewer's right, and +Y towards the upper edge of the screen. The rotation must be in degrees; positive rotation is a counter-clockwise rotation as described in the IDFv3 specification.

Multiple outlines may be combined with appropriate offsets to represent simple assemblies such as a DIP package in a socket.

---

**Note**

Only the offset values and the Z rotation value are used by the IDF exporter; all other values are ignored.

---

### 8.4.2 Creating a component outline file

The component outline file (\* .idf) consists of a single .ELECTRICAL or .MECHANICAL section as described in the specification document. The section may be preceded by any number of comment lines; the comment lines are copied by the exporter into the library file and can be used to track metadata such as references to the documents used to determine the component's outline and dimensions.

The component outline section contains fields which are strings, integers, or floating point numbers. A string is a combination of characters which may include spaces; if a string contains spaces then it must be quoted. Quotation marks must not appear within a string. Floating point numbers may be represented using decimal or exponential notations but decimal notation is preferred for human readability. The decimal point must be a dot and not a comma. The IDF file must consist only of 7-bit ASCII characters; use of 8-bit characters will result in undefined behavior.

An IDF file consists of SECTIONS which consist of RECORDS which consist of FIELDS. For the IDF outline files only one type of section may exist and must be one of .ELECTRICAL or .MECHANICAL. A record is a single line of text and may contain one or more fields. Fields are sequences of characters separated by one or more spaces which do not appear between quotation marks. All fields of a record must appear on a single line; records may not span lines.

The section heading (.ELECTRICAL or .MECHANICAL) is considered the first record (Record 1) of the section. Record 1 must be followed by Record 2 which has four fields:

1. Geometry Name: a string which in combination with the Part Number must form a unique identifier for the component outline. For standardized packages, the package name is a good value for the geometry name, for example "SOT-23". For unique packages the manufacturer's part number is a good choice for the geometry name.
2. Part Number: although obviously intended for the part number, for example BS107, it is better to use this string to help describe the package. For example if the geometry name is "TO-92", the part number entry may be used to describe the layout of the pads or the orientation of this particular TO-92 outline file.
3. IDF Unit: this must be one of MM or THOU and it applies only to the units describing this single component outline.
4. Height: this is a floating point number representing the nominal height of the component using units specified in Field 3.

Record 2 must be followed by a number of Record 3 entries which specify the outline of the component. Record 3 consists of four fields:

1. Loop Index: 0 (outline points are specified in counter-clockwise order) or 1 (outline points are specified in clockwise order)
2. X coordinate: a floating point number
3. Y coordinate: a floating point number
4. Included Angle: a floating point number. If the value is 0 then a straight line segment is drawn from the previous point to this point. If the value is 360 then the previous point specifies the center of a circle and this point specifies a point on the circle; never specify a circle using a value of -360 as at least one major mechanical CAD package does not behave well in that situation. If the value is negative then a clockwise arc is drawn from the previous point to this point and if the value is positive then a counter-clockwise arc is drawn.

Only one closed loop is permitted and it is not possible to specify a cutout. The last point specified must be the same as the first point unless the outline is a circle.

Example IDF File 1:

---

```
# a simple cylinder - this could represent an electrolytic capacitor
.ELECTRICAL
    "cylinder" "5mm OD, 5mm height" MM 5
    0 0 0 0
    0 2.5 0 360
.END_ELECTRICAL
```

#### Example IDF File 2:

```
# an upside-down T
# a comment added for the sake of adding comments
.ELECTRICAL
    "Capital T" "5x8x10mm, upside down" MM 10
    0 -0.5 8 0
    0 -0.5 0.5 0
    0 -2.5 0.5 0
    0 -2.5 -0.5 180
    0 2.5 -0.5 0
    0 2.5 0.5 180
    0 0.5 0.5 0
    0 0.5 8 0
    0 -0.5 8 180
.END_ELECTRICAL
```

### 8.4.3 Guidelines for creating outlines

When creating outlines, and especially when sharing the work with others, consistency in the design and naming of files helps people locate files quicker and place the components with minimal hassles.

#### 8.4.3.1 Package naming

Try to make some information about the outline available in the filename to give the user a general idea of what the outline is. For example axial leaded cylindrical packages may represent some types of capacitors as well as some types of resistors, so it makes sense to identify an outline as a horizontal or vertical axial leaded device and to add some extra information on the relevant dimensions: diameter, length, and pitch are the most important. If a device has a unique outline, the manufacturer's part number and a prefix to indicate the class of device are adequate.

#### 8.4.3.2 Comments

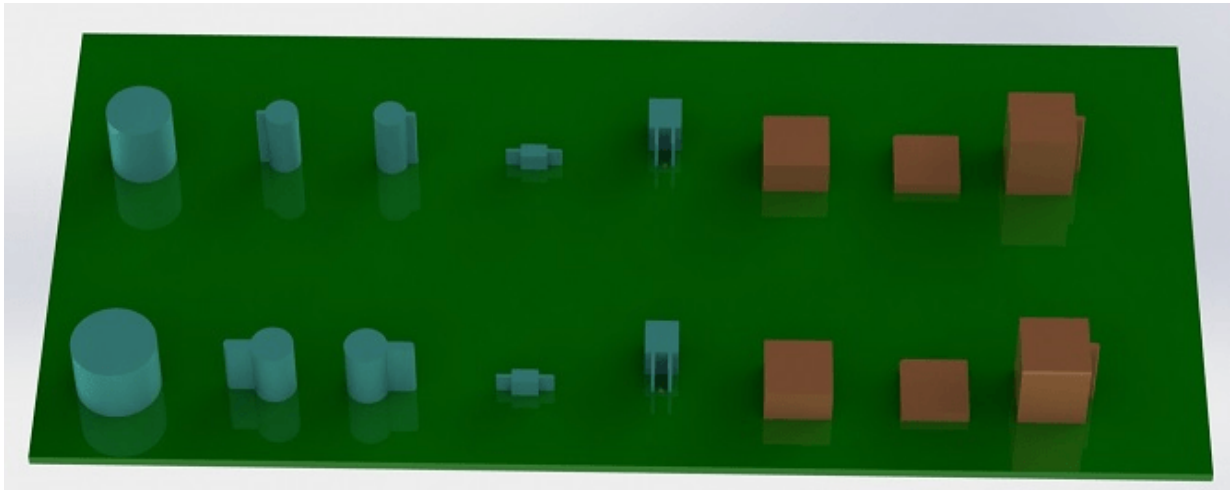
Use comments in the IDF file to give users more information about the outline, for example a reference to the source used for dimensional information.

#### 8.4.3.3 Geometry and Part Number entries

Think carefully about the values to give to the Geometry and Part Number entries. Taken together, these strings act as a unique identifier for the MCAD system. The values of the strings will ideally have some meaning to a user, but this is not necessary: the values are primarily intended for the MCAD system to use as a unique ID. Ideally the values chosen will be unique within any large collection of outlines; choosing values well will result in fewer clashes especially in complex boards.

#### 8.4.3.4 Pin orientation and positioning

Component outlines should be created to match the orientation and position of the corresponding footprints. This avoids the need to specify a non-zero rotation for the IDF component outline. Since the IDF exporter ignores the (X, Y) offset values, it is vital that you use the correct origin in the IDF component outline.



The image above shows sample outlines generated by the programs `idfcyl` and `idfrect` and rendered in a mechanical CAD program. From left to right are (a) vertical radial led cylinder, (b) vertical axial led cylinder with wire on left, (c) vertical axial led cylinder with wire on right, (d) horizontal axial led cylinder, (e) horizontal radial led cylinder, (f) square outline, plain, (g) square outline with chamfer, (h) square outline with axial lead on right. The top outlines were specified in units of millimeters while the bottom outlines were specified in units of inches.

#### 8.4.3.5 Tips on dimensions

The purpose served by the extruded outlines is to give the mechanical designer some idea of the location and physical space occupied by each component. In a typical scenario the mechanical designer will replace some of the crude outlines with more detailed mechanical models, for example when checking to ensure that a right-angle mounted LED will fit into a hole on a panel. In most situations the accuracy of an outline doesn't matter, but it is good practice to create outlines which convey the best mechanical information possible. In a few instances a user may wish to fit the component into a case with very little excess space, for example in a portable music player. In such a situation, if most extruded outlines are a good enough representation of components then the mechanical designer may only have to replace very few models while designing the case. If the outlines are not a reliable reflection of reality then the mechanical designer will waste a lot of time replacing models to ensure a good fit. After all, if you put garbage in you can expect garbage to come out. If you put in good information, you can be confident of good results.

### 8.4.4 IDF Component Outline Tools

A number of command-line tools are available to help generate IDF component outlines. The tools are:

1. `idfcyl`: creates an outline of a cylinder in vertical or horizontal orientation and with axial or radial leads
2. `idfrect`: creates an outline of a rectangle which may have either an axial lead or a chamfer in the top left corner
3. `dx2idf`: converts a drawing in DXF format into an IDF component outline

#### 8.4.4.1 idfcyl

`idfcyl` generates outlines for cylindrical components.

When `idfcyl` is invoked with no arguments it prints out a usage note and a summary of its inputs:

```
idfcyl: This program generates an outline for a cylindrical component.
The cylinder may be horizontal or vertical.
A horizontal cylinder may have wires at one or both ends.
A vertical cylinder may have at most one wire which may be
placed on the left or right side.
```

**Input:**

```

Unit: mm, in (millimeters or inches)
Orientation: V (vertical)
Lead type: X, R (axial, radial)
Diameter of body
Length of body
Board offset
* Wire diameter
* Pitch
** Wire side: L, R (left, right)
*** Lead length
File name (must end in *.idf)

```

**NOTES:**

```

* only required for horizontal orientation or
  vertical orientation with axial leads

** only required for vertical orientation with axial leads

*** only required for horizontal orientation with radial leads

```

The notes can be suppressed by entering any arbitrary argument on the command line. A user can manually enter information at the command line or create scripts to generate outlines. The following script creates a single cylinder axial leaded outline with the lead on the right hand side:

```

#!/bin/bash
# Generate a cylindrical IDF outline for test purposes
# vertical 5mm cylinder, nominal length 8mm + 3mm board offset,
# axial wire on right, 0.8mm wire dia., 3.5mm pitch
idfctl - 1 > /dev/null << _EOF
mm
v
x
5
8
3
0.8
3.5
r
cylvmm_1R_D5_L8_Z3_WD0.8_P3.5.idf
_EOF

```

**8.4.4.2 idfrect**

idfrect generates outlines for rectangular components.

When idfrect is invoked with no arguments it prints out a usage note and a summary of its inputs:

```

idfrect: This program generates an outline for a rectangular component.
The component may have a single lead (axial) or a chamfer on the
upper left corner.
Input:
Unit: mm, in (millimeters or inches)
Width:
Length:
Height:
Chamfer: length of the 45 deg. chamfer
* Leaded: Y,N (lead is always to the right)
** Wire diameter
** Pitch

```

File name (must end in \*.idf)

NOTES:

\* only required if chamfer = 0

\*\* only required for leaded components

The notes can be suppressed by entering any arbitrary argument on the command line. A user can manually enter information at the command line or create scripts to generate outlines. The following script creates a chamfered rectangle and an axial leaded outline:

```
#!/bin/bash
# Generate various rectangular IDF outlines for test purposes
# 10x10, 1mm chamfer, 2mm height
idfrect - 1 > /dev/null << _EOF
mm
10
10
2
1
rectMM_10x10x2_C0.5.idf
_EOF
# 10x10x12, 0.8mm lead on 6mm pitch
idfrect - 1 > /dev/null << _EOF
mm
10
10
12
0
Y
0.8
6
rectLMM_10x10x12_D0.8_P6.0.idf
_EOF
```

#### 8.4.4.3 dxf2idf

dxf2idf creates an IDF component file from a DXF outline.

The DXF file used to specify the component outline can be prepared with the free software [LibreCAD](#) for best compatibility.

When dxf2idf is invoked with no arguments it prints out a usage note and a summary of its inputs:

```
dxf2idf: this program takes line, arc, and circle segments
        from a DXF file and creates an IDF component outline file.

Input:
  DXF filename: the input file, must end in '.dxf'
  Units: mm, in (millimeters or inches)
  Geometry Name: string, as per IDF version 3.0 specification
  Part Name: as per IDF version 3.0 specification of Part Number
  Height: extruded height of the outline
  Comments: all non-empty lines are comments to be added to
            the IDF file. An empty line signifies the end of
            the comment block.
  File name: output filename, must end in '.idf'
```

The notes can be suppressed by entering any arbitrary argument on the command line. A user can manually enter information at the command line or create scripts to generate outlines. The following script creates a 5mm high outline from a DXF file test.dxf:



```
#!/bin/bash
# Generate an IDF outlines from a DXF file
dxf2idf - 1 > /dev/null << _EOF
test.dxf
mm
DXF TEST GEOMETRY
DXF TEST PART
5
This is an IDF test file produced from the outline 'test.dxf'
This is a second IDF comment to demonstrate multiple comments

test_dxf2idf.idf
_EOF
```

#### 8.4.4.4 idf2vrmI

The `idf2vrmI` tool reads a set of one IDF Board (`.emn`) and one IDF Component file (`.emp`) and produces a VRML file which can be viewed with a VRML viewer. This feature is useful for visualization of the board assembly in cases where the user does not have access to MCAD software. Invoking `idf2vrmI` without any arguments will result in the display of a usage message:

```
>./idf2vrmI
Usage: idf2vrmI -f input_file.emn -s scale_factor {-k} {-d} {-z} {-m}
flags:
  -k: produce KiCad-friendly VRML output; default is compact VRML
  -d: suppress substitution of default outlines
  -z: suppress rendering of zero-height outlines
  -m: print object mapping to stdout for debugging purposes
example to produce a model for use by KiCad: idf2vrmI -f input.emn -s 0.3937008 -k
```

---

#### Note

The `idf2vrmI` tool does not correctly render `OTHER_OUTLINE` entities in an `emn` file if that entity is specified on the back layer of the PCB; however you will not noticeable using files exported by KiCad because there is no mechanism to specify such an entity. This is only an issue if you render a third party `emn` file which does employ an entity on the back side of a board.

---

## Chapter 9

# &#x64cd;&#x4f5c;&#x53c2;&#x8003;

Below is a list of every available **action** in Pcbnew: a command that can be assigned to a hotkey.

### 9.1 PCB &#x7f16;&#x8f91;&#x5668;

The actions below are available in Pcbnew. Hotkeys can be assigned to any of these actions in the **Hotkeys** section of the preferences.

|  |  |
|--|--|
| &#x64cd;&#x4f5c;   | &#x9ed8;&#x8ba4;&#x56f4;&#x6007;&#x952e;   |
| &#x5411;&#x4e0b;&#x5bf9;&#x9f50;   | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x4e0e;&#x4e0b;&#x8fb9;   |
| &#x4e0e;&#x5782;&#x76f4;&#x4e2d;&#x5fc3;&#x5bf9;&#x9f50;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x5782;&#x76f4;&#x5c45; | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x5782;&#x76f4;&#x5c45;   |
| &#x4e0e;&#x6c34;&#x5e73;&#x4e2d;&#x5fc3;&#x5bf9;&#x9f50;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x4e0e;&#x6c34;&#x5e73; | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x4e0e;&#x6c34;&#x5e73;   |
| &#x5411;&#x5de6;&#x5bf9;&#x9f50;   | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x4e0e;&#x5de6;&#x8fb9;   |
| &#x5411;&#x53f3;&#x5bf9;&#x9f50;   | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x4e0e;&#x53f3;&#x8fb9;   |
| &#x5411;&#x4e0a;&#x5bf9;&#x9f50;   | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x4e0e;&#x4e0a;&#x8fb9;   |
| &#x6c34;&#x5e73;&#x5206;&#x5e03;   | &#x6cbf;&#x6c34;&#x5e73;&#x8f74;&#x5206;&#x5e03;&#x9009;&#x5b9a;   |
| &#x5782;&#x76f4;&#x5206;&#x5e03;   | &#x6cbf;&#x5782;&#x76f4;&#x8f74;&#x5206;&#x5e03;&#x9009;&#x5b9a;   |
| &#x653e;&#x7f6e;&#x7535;&#x8def;&#x677f;&#x5916;&#x7535;&#x8def;&#x677f;&#x533a;&#x57df;&#x5916;                 | &#x653e;&#x7f6e;&#x7535;&#x8def;&#x677f;&#x5916;&#x7535;&#x8def;&#x677f;&#x533a;&#x57df;&#x5916;                 |
| &#x653e;&#x7f6e;&#x9009;&#x62e9;&#x7684;&#x5143;&#x4ef6;&#x7684;   | &#x653e;&#x7f6e;&#x9009;&#x62e9;&#x7684;&#x5143;&#x4ef6;&#x7684;   |
| &#x7ffb;&#x8f6c;&#x7535;&#x8def;&#x677f;&#x9b64;&#x8f6c;&#x53cd;&#x9762;&#x770b;&#x677f;                         | &#x7ffb;&#x8f6c;&#x7535;&#x8def;&#x677f;&#x9b64;&#x8f6c;&#x53cd;&#x9762;&#x770b;&#x677f;                         |
| &#x8349;&#x56fe;&#x56fe;&#x5f62;&#x9879;&#x76ee;&#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a; | &#x8349;&#x56fe;&#x56fe;&#x5f62;&#x9879;&#x76ee;&#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a; |
| &#x964d;&#x4f4e;&#x56fe;&#x5f62;&#x4e0d;&#x900f;&#x8f6e;&#x5f53;&#x524d;&#x56fe;&#x5c42;&#x7684;&#x900f;         | &#x964d;&#x4f4e;&#x56fe;&#x5f62;&#x4e0d;&#x900f;&#x8f6e;&#x5f53;&#x524d;&#x56fe;&#x5c42;&#x7684;&#x900f;         |
| &#x589e;&#x52a0;&#x56fe;&#x5f62;&#x4e0d;&#x900f;&#x8f6e;&#x5f53;&#x524d;&#x56fe;&#x5c42;&#x7684;&#x900f;         | &#x589e;&#x52a0;&#x56fe;&#x5f62;&#x4e0d;&#x900f;&#x8f6e;&#x5f53;&#x524d;&#x56fe;&#x5c42;&#x7684;&#x900f;         |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;&#x4ef6;   | &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;&#x4ef6;   |
| (F.Cu) &#x5c42;  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 1  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 2  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 3  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 4  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 5  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 6  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;   |  |
| 7  |  |

|   |  |
|---|--|
| &#x64cd;&#x4f5c;  | &#x9ed8;&#x8ba4;&#x5ff4;&#x607e;&#x952e;                         |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>8                 |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>9                 |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>10                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>11                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>12                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>13                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>14                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>15                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>16                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>17                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>18                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>19                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>20                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>21                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>22                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>23                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>24                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>25                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>26                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>27                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>28                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>29                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>30                |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>(B.Cu) &#x5c42;   |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>&#x4e00;&#x5c42;  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>&#x4e00;&#x5c42;  |  |
| &#x5207;&#x6362;&#x5230;&#x5185;&#x5c42;<br>&#x56fe;&#xbdc[2] | &#x5728;&#x6d3b;&#x52a8;&#x5c42;&#x5bf9;&#x4e2d;&#x7684;&#x5c42  |
| &#x7f51;&#x7edc;&#x68c0;&#x67e5;&#x5668;                      | &#x663e;&#x793a;&#x7f51;&#x7edc;&#x68c0;&#x67e5;&#x5668;         |
| &#x9ad8;&#x4eae;&#x98de;&#x7ebf;                              | &#x663e;&#x793a;&#x6240;&#x9009;&#x9879;&#x76ee;&#x7684;&#x98de  |
| &#x8349;&#x56fe;&#x710a;&#x76d8;                              | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a; |
| &#x5f2f;&#x66f2;&#x7684;&#x98de;&#x7ebf;                      | &#x7528;&#x66f2;&#x7ebf;&#x663e;&#x793a;&#x98de;&#x7ebf;         |
| &#x4fee;&#x590d;&#x7535;&#x8def;&#x677f;                      | &#x8fd0;&#x884c;&#x5404;&#x79cd;&#x8bca;&#x65ad;&#x7a0b;&#x5e8f; |

|  |  |  |
|--|--|--|
| &#x64cd;&#x4f5c;   | &#x9ed8;&#x8ba4;&#x5f64;&#x6007;&#x952e;   | &#x5f64;&#x6007;&#x952e;   |
| &#x663e;&#x793a;&#x5916;&#x89c2;&#x7ba1;&#x7406;   | &#x7406;&#x5668;&#x952e;&#x85cf;&#x5916;&#x89c2;&#x7ba1;&#x7406;   | &#x7406;&#x5668;&#x952e;&#x85cf;&#x5916;&#x89c2;&#x7ba1;&#x7406;   |
| &#x663e;&#x793a;&#x710a;&#x76d8;&#x7f16;&#x53f7;   | &#x53f7;&#x793a;&#x710a;&#x76d8;&#x7f16;&#x53f7;   | &#x53f7;&#x793a;&#x710a;&#x76d8;&#x7f16;&#x53f7;   |
| &#x811a;&#x672c;&#x63a7;&#x5236;&#x53f0;   | &#x663e;&#x793a; Python<br>&#x811a;&#x672c;&#x63a7;&#x5236;&#x53f0;                                      | &#x663e;&#x793a; Python<br>&#x811a;&#x672c;&#x63a7;&#x5236;&#x53f0;                                      |
| &#x663e;&#x793a;&#x98de;&#x7ebf;   | &#x663e;&#x793a;&#x7535;&#x8def;&#x677f;&#x98de;&#x7ebf;   | &#x663e;&#x793a;&#x7535;&#x8def;&#x677f;&#x98de;&#x7ebf;   |
| &#x8349;&#x56fe;&#x6587;&#x672c;&#x9879;   | &#x5728;&#x884c;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;&#x5c01;   | &#x5728;&#x884c;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;&#x5c01;   |
| &#x8349;&#x56fe;&#x5e03;&#x7ebf;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   |
| &#x8349;&#x56fe;&#x8fc7;&#x5b54;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   |
| &#x7ebf;&#x6846;&#x533a;&#x57df;   | &#x4ec5;&#x663e;&#x793a;&#x533a;&#x57df;&#x8fb9;&#x754c;   | &#x4ec5;&#x663e;&#x793a;&#x533a;&#x57df;&#x8fb9;&#x754c;   |
| &#x586b;&#x5145;&#x8986;&#x94dc;   | &#x663e;&#x793a;&#x8986;&#x94dc;&#x7684;&#x586b;&#x5145;&#x533a;   | &#x663e;&#x793a;&#x8986;&#x94dc;&#x7684;&#x586b;&#x5145;&#x533a;   |
| &#x8349;&#x56fe;&#x533a;&#x57df;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   |
| &#x5207;&#x6362;&#x533a;&#x7ebf;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   | &#x5728;&#x8f6e;&#x5ed3;&#x6a21;&#x5f0f;&#x4e0b;&#x663e;&#x793a;   |
| &#x81ea;&#x52a8;&#x7f29;&#x653e;&#x4ee5;&#x9009;&#x5c01;&#x88c5;&#x65f6;&#x7f29;&#x653e;&#x4ee5;         | &#x5c06;&#x9009;&#x5b9a;&#x7684;&#x7ebf;&#x6bb5;&#x8f6c;&#x6362;   | &#x5c06;&#x9009;&#x5b9a;&#x7684;&#x7ebf;&#x6bb5;&#x8f6c;&#x6362;   |
| &#x8f6c;&#x6362;&#x4e3a;&#x5706;&#x5f27;   | &#x5c06;&#x9009;&#x5b9a;&#x7684;&#x7ebf;&#x6bb5;&#x8f6c;&#x6362;   | &#x5c06;&#x9009;&#x5b9a;&#x7684;&#x7ebf;&#x6bb5;&#x8f6c;&#x6362;   |
| &#x8f6c;&#x6362;&#x4e3a;&#x89c4;&#x5219;&#x533a;&#x76e4;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x89c4; | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x56fe;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x56fe;   |
| &#x8f6c;&#x6362;&#x4e3a;&#x7ebf;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x56fe;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x56fe;   |
| &#x8f6c;&#x6362;&#x4e3a;&#x591a;&#x8fb9;&#x56fe;   | &#x5c06;&#x9009;&#x5b9a;&#x7684;&#x56fe;&#x5f62;&#x7ebf;&#x8f6c;   | &#x5c06;&#x9009;&#x5b9a;&#x7684;&#x56fe;&#x5f62;&#x7ebf;&#x8f6c;   |
| &#x8f6c;&#x6362;&#x4e3a;&#x5e03;&#x7ebf;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x8986;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x8986;   |
| &#x8f6c;&#x6362;&#x4e3a;&#x8986;&#x94dc;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x8986;   | &#x4ece;&#x6240;&#x9009;&#x5185;&#x5bb9;&#x521b;&#x5efa;&#x8986;   |
| &#x8bbe;&#x8ba1;&#x89c4;&#x5219;&#x68c0;&#x67e5;   | &#x5728;&#x5c01;&#x88c5;&#x7f29;&#x653e;&#x4ee5;&#x9009;&#x5c01;&#x88c5;&#x65f6;&#x7f29;&#x653e;&#x4ee5; | &#x5728;&#x5c01;&#x88c5;&#x7f29;&#x653e;&#x4ee5;&#x9009;&#x5c01;&#x88c5;&#x65f6;&#x7f29;&#x653e;&#x4ee5; |
| &#x5728;&#x5c01;&#x88c5;&#x7f29;&#x653e;&#x4ee5;&#x9009;&#x5c01;&#x88c5;&#x65f6;&#x7f29;&#x653e;&#x4ee5; | &#x6253;&#x5f00;&#x53e6;&#x4e00;&#x4e2a;&#x7535;&#x8def;&#x677f;   | &#x6253;&#x5f00;&#x53e6;&#x4e00;&#x4e2a;&#x7535;&#x8def;&#x677f;   |
| &#x9644;&#x52a0;&#x7535;&#x8def;&#x677f;   | &#x7f16;&#x8f91;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#xff0c;   | &#x7f16;&#x8f91;&#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;&#xff0c;   |
| &#x7535;&#x8def;&#x677f;&#x8bbe;&#x7f6e;   | &#x6e05;&#x9664;&#x7f51;&#x4efb;&#x4f55;&#x73b0;&#x6709;&#x7f51;&#x7edc;                                 | &#x6e05;&#x9664;&#x7f51;&#x4efb;&#x4f55;&#x73b0;&#x6709;&#x7f51;&#x7edc;                                 |
| &#x6e05;&#x9664;&#x7f51;&#x4efb;&#x4f55;&#x73b0;&#x6709;&#x7f51;&#x7edc;                                 | &#x94bb;&#x5b54;&#x653e;&#x7f6e;&#x6587;&#x4ef6;&#x548c;&#x5143;   | &#x94bb;&#x5b54;&#x653e;&#x7f6e;&#x6587;&#x4ef6;&#x548c;&#x5143;   |
| &#x94bb;&#x5b54;&#x653e;&#x7f6e;&#x6587;&#x4ef6;&#x548c;&#x5143;   | &#x5bfc;&#x51fa; Spectra<br>DSN...   | &#x5bfc;&#x51fa; Spectra DSN<br>&#x5e03;&#x7ebf;&#x4fe1;&#x606f;   |
| &#x5bfc;&#x51fa; Spectra<br>DSN...   | BOM...   | &#x4ece;&#x7535;&#x8def;&#x677f;&#x521b;&#x5efa; BOM<br>&#x8868;   |
| BOM...   | IPC-D-356  | &#x7f51;&#x6210; IPC-D-356   |
| IPC-D-356  | &#x7f51;&#x8868;&#x6587;&#x4ef6;...  | &#x7f51;&#x8868;&#x6587;&#x4ef6;   |
| &#x7f51;&#x8868;&#x6587;&#x4ef6;...  | &#x94bb;&#x5b54;&#x6587;&#x4ef6;<br>(.drl)...  | &#x7f51;&#x6210; Excellon<br>&#x94bb;&#x5b54;&#x6587;&#x4ef6;  |
| &#x94bb;&#x5b54;&#x6587;&#x4ef6;<br>(.drl)...  | Gerbers (.gbr)...  | &#x4e3a;&#x5236;&#x9020;&#x7f51;&#x6210; Gerbers   |
| Gerbers (.gbr)...  | &#x5143;&#x4ef6;&#x653e;&#x7f6e;<br>(.pos)...  | &#x4e3a;&#x8d34;&#x7247;&#x548c;&#x63d2;&#x4ef6;&#x7f51;&#x6210;   |
| &#x5143;&#x4ef6;&#x653e;&#x7f6e;<br>(.pos)...  | &#x5c01;&#x88c5;&#x62a5;&#x544a;<br>(.rpt)...  | &#x4ece;&#x5f53;&#x524d;&#x7535;&#x8def;&#x677f;&#x521b;&#x5efa;   |
| &#x5c01;&#x88c5;&#x62a5;&#x544a;<br>(.rpt)...  | &#x7ec4;&#x5408;   | &#x5c06;&#x6240;&#x9009;&#x9879;&#x76ee;&#x7ec4;&#x5408;&#xff0c;   |
| &#x7ec4;&#x5408;   | &#x8fdb;&#x5165;&#x7ec4;&#x5408;   | &#x8fdb;&#x5165;&#x8981;&#x7f16;&#x8f91;&#x9879;&#x76ee;&#x7684;   |
| &#x8fdb;&#x5165;&#x7ec4;&#x5408;   | &#x79bb;&#x5f00;&#x7ec4;&#x5408;   | &#x79bb;&#x5f00;&#x5f53;&#x524d;&#x7ec4;&#x5408;   |
| &#x79bb;&#x5f00;&#x7ec4;&#x5408;   | &#x9690;&#x85cf;&#x7f51;&#x7edc;   | &#x9690;&#x85cf;&#x6240;&#x9009;&#x7f51;&#x7edc;&#x7684;&#x98de;   |
| &#x9690;&#x85cf;&#x7f51;&#x7edc;   | &#x9ad8;&#x4eae;&#x7f51;&#x7edc;   | &#x9ad8;&#x4eae;&#x7f51;&#x7edc;&#x4e2d;&#x7684;&#x6240;&#x6709;   |
| &#x9ad8;&#x4eae;&#x7f51;&#x7edc;   | &#x5bfc;&#x5165;&#x7f51;&#x8868;...  | &#x8bfb;&#x53d6;&#x7f51;&#x8868;&#x5e76;&#x66f4;&#x65b0;&#x7535;   |
| &#x5bfc;&#x5165;&#x7f51;&#x8868;...  | &#x5bfc;&#x5165;   | &#x5bfc;&#x5165;&#x5e03;&#x7ebf;&#x7684; Spectra session<br>(* .ses) &#x6587;&#x4ef6;                    |
| &#x5bfc;&#x5165;   | Spectra<br>&#x4f1a;&#x8bdd;...   |  |
| Spectra<br>&#x4f1a;&#x8bdd;...   | &#x9501;&#x5b9a;   | &#x9632;&#x6b62;&#x9879;&#x76ee;&#x5728;&#x753b;&#x5e03;&#x4e0a;   |
| &#x9501;&#x5b9a;   | &#x6dfb;&#x52a0;&#x5c01;&#x88c5;   | &#x6dfb;&#x52a0;&#x5c01;&#x88c5;   |
| &#x6dfb;&#x52a0;&#x5c01;&#x88c5;   | &#x6dfb;&#x52a0;&#x56fe;&#x5c42;&#x5bf9;&#x9f50;&#x76ee;&#x6807;   | &#x6dfb;&#x52a0;&#x56fe;&#x5c42;&#x5bf9;&#x9f50;&#x76ee;&#x6807;   |
| &#x6dfb;&#x52a0;&#x56fe;&#x5c42;&#x5bf9;&#x9f50;&#x76ee;&#x6807;   | &#x5220;&#x9664;&#x9879;&#x76ee;   | &#x4ece;&#x7ec4;&#x5408;&#x4e2d;&#x5220;&#x9664;&#x9879;&#x76ee;   |
| &#x5220;&#x9664;&#x9879;&#x76ee;   | &#x5207;&#x6362;&#x5230;&#x539f;&#x7406;&#x5668;   | &#x5668;&#x7f29;&#x653e;&#x4e2d;&#x6253;&#x5f00;&#x539f;&#x7406;&#x56fe;                                 |
| &#x5207;&#x6362;&#x5230;&#x539f;&#x7406;&#x5668;   | &#x663e;&#x793a;&#x7f51;&#x7edc;   | &#x663e;&#x793a;&#x6240;&#x9009;&#x7f51;&#x7edc;&#x7684;&#x98de;   |
| &#x663e;&#x793a;&#x7f51;&#x7edc;   | &#x5207;&#x6362;&#x4e0a;&#x6b21;&#x7f51;&#x76e4;&#x9ad8;&#x4eae;&#x7684;                                 | &#x5207;&#x6362;&#x4e0a;&#x6b21;&#x7f51;&#x76e4;&#x9ad8;&#x4eae;&#x7684;                                 |

|  |  |  |
|--|--|--|
| &#x64cd;&#x4f5c;   | &#x9ed8;&#x8ba4;&#x5b64;&#x607e;&#x952e;   | &#x5000;&#x5b9a;&#x6216;&#x89e3;&#x9501;&#x9009;&#x5b9a;&#x9879;   |
| &#x5207;&#x6362;&#x9501;&#x5b9a;   | &#x5207;&#x6362;&#x7f51;&#x7edc;&#x9ad8;&#x4eae;   | &#x5207;&#x6362;&#x7f51;&#x7edc;&#x9ad8;&#x4eae;   |
| &#x5c06;&#x5e03;&#x7ebf;&#x5ea6;&#x5207;&#x6362;&#x7f51;&#x4e0a;&#x66f4;&#x6539;&#x4e3a;                 | &#x5c06;&#x5e03;&#x7ebf;&#x5ea6;&#x5207;&#x6362;&#x7f51;&#x4e0a;&#x66f4;&#x6539;&#x4e3a;                 | &#x5c06;&#x5e03;&#x7ebf;&#x5ea6;&#x5207;&#x6362;&#x7f51;&#x4e0a;&#x66f4;&#x6539;&#x4e3a;                 |
| &#x89e3;&#x7ec4;   | &#x53d6;&#x6d88;&#x5bf9;&#x4efb;&#x4f55;&#x9009;&#x5b9a;&#x7ec4;   | &#x89e3;&#x7ec4;   |
| &#x89e3;&#x9501;   | &#x5141;&#x8bb8;&#x5728;&#x753b;&#x5e03;&#x4e0a;&#x79fb;&#x52a8  | &#x89e3;&#x9501;   |
| &#x51cf;&#x5c0f;&#x8fc7;&#x5b54;&#x5c3a;&#x5b5c;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x51cf;&#x5c0f;&#x8fc7;&#x5b54;&#x5c3a;&#x5b5c;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x51cf;&#x5c0f;&#x8fc7;&#x5b54;&#x5c3a;&#x5b5c;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; |
| &#x589e;&#x5927;&#x8fc7;&#x5b54;&#x5c3a;&#x5b5c;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x589e;&#x5927;&#x8fc7;&#x5b54;&#x5c3a;&#x5b5c;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x589e;&#x5927;&#x8fc7;&#x5b54;&#x5c3a;&#x5b5c;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; |
| &#x5c06;&#x533a;&#x57df;&#x590d;&#x5236;&#x5200;&#x5652;&#x5b53;&#x4b0f;&#x5ed3;&#x590d;&#x5236;&#x5230; | &#x5c06;&#x533a;&#x57df;&#x590d;&#x5236;&#x5200;&#x5652;&#x5b53;&#x4b0f;&#x5ed3;&#x590d;&#x5236;&#x5230; | &#x5c06;&#x533a;&#x57df;&#x590d;&#x5236;&#x5200;&#x5652;&#x5b53;&#x4b0f;&#x5ed3;&#x590d;&#x5236;&#x5230; |
| &#x5408;&#x5e76;&#x533a;&#x57df;   | &#x5408;&#x5e76;&#x533a;&#x57df;   | &#x5408;&#x5e76;&#x533a;&#x57df;   |
| &#x66f4;&#x6539;&#x5c01;&#x88c5;...  | &#x4ece;&#x5e93;&#x4e2d;&#x5206;&#x914d;&#x4e0d;&#x540c;&#x7684  | &#x66f4;&#x6539;&#x5c01;&#x88c5;...  |
| &#x66f4;&#x6539;&#x5c01;&#x88c5;...  | &#x4ece;&#x5e93;&#x4e2d;&#x5206;&#x914d;&#x4e0d;&#x540c;&#x7684  | &#x66f4;&#x6539;&#x5c01;&#x88c5;...  |
| &#x6e05;&#x9664;&#x56fe;&#x5f62;...  | &#x6e05;&#x9664;&#x591a;&#x4f59;&#x9879;&#x76ee;&#x7b49;&#x3002  | &#x6e05;&#x9664;&#x56fe;&#x5f62;...  |
| &#x6e05;&#x9664;&#x5e03;&#x7ebf;&#x548c;&#x5c06;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x6e05;&#x9664;&#x5e03;&#x7ebf;&#x548c;&#x5c06;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x6e05;&#x9664;&#x5e03;&#x7ebf;&#x548c;&#x5c06;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; |
| &#x7f16;&#x8f91;&#x6587;&#x672c;&#x548c;&#x5c06;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x7f16;&#x8f91;&#x6587;&#x672c;&#x548c;&#x5c06;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x7f16;&#x8f91;&#x6587;&#x672c;&#x548c;&#x5c06;&#x8fc7;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; |
| &#x7f16;&#x8f91;&#x5e03;&#x7ebf;&#x548c;&#x8fc7;&#x6b8c;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x7f16;&#x8f91;&#x5e03;&#x7ebf;&#x548c;&#x8fc7;&#x6b8c;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; | &#x7f16;&#x8f91;&#x5e03;&#x7ebf;&#x548c;&#x8fc7;&#x6b8c;&#x5b54;&#x5c3a;&#x5bf8;&#x66f4;&#x6539;&#x4e3a; |
| &#x5168;&#x5c40;&#x5220;&#x9664;...  | &#x4ece;&#x7535;&#x8def;&#x677f;&#x4e2d;&#x5220;&#x9664;&#x5e03;   | &#x5168;&#x5c40;&#x5220;&#x9664;...  |
| &#x79fb;&#x9664;&#x672a;&#x4f7f;&#x7528;&#x7684;&#x710a;&#x4b0f;&#x5ed3;&#x590d;&#x5236;&#x5230;         | &#x79fb;&#x9664;&#x672a;&#x4f7f;&#x7528;&#x7684;&#x710a;&#x4b0f;&#x5ed3;&#x590d;&#x5236;&#x5230;         | &#x79fb;&#x9664;&#x672a;&#x4f7f;&#x7528;&#x7684;&#x710a;&#x4b0f;&#                                       |

|  |  |  |
|--|--|--|
| &#x64cd;&#x4f5c;   | &#x9ed8;&#x8ba4;&#x5ff4;&#x6077;&#x952e; | &#x66f4;&#x6539;&#x4fa7;&#x97f2;/&#x7ffb;&#x8f6c06;&#x6240;&#x9009;&#x9879;&#x76ee;&#x7ffb;&#x8f6c;&#x5230;      |
| &#x955c;&#x50cf;   |  | &#x955c;&#x50cf;&#x9009;&#x62e9;&#x9879;   |
| &#x7cbe;&#x786e;&#x79fb;&#x52a8;[Ctrl+M]   |  | &#x6309;&#x7cbe;&#x786e;&#x7684;&#x6570;&#x91cf;&#x79fb;&#x52a8;   |
| &#x5c5e;&#x6027;... kbd:[E]  |  | &#x663e;&#x793a;&#x9879;&#x76ee;&#x5c5e;&#x6027;&#x5bf9;&#x8bdd;   |
| &#x9006;&#x65f6;&#x9488;&#x8f6c;&#x8f6c;   |  | &#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;&#x6240;&#x9009;&#x9879;   |
| &#x987a;&#x65f6;&#x9488;&#x8f6c;&#x8f6c;   |  | &#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;&#x6240;&#x9009;&#x9879;   |
| &#x5e26;&#x4f4d;&#x53f7;&#x590d;&#x5236;   |  | &#x5c06;&#x9009;&#x5b9a;&#x9879;&#x76ee;&#x590d;&#x5236;&#x5230;   |
| &#x79fb;&#x52a8; kbd:[M]   |  | &#x79fb;&#x52a8;&#x9009;&#x5b9a;&#x9879;&#x76ee;   |
| &#x968f;&#x4f4d;&#x53f7;&#x79fb;&#x52a8;   |  | &#x79fb;&#x52a8;&#x5177;&#x6709;&#x6307;&#x5b9a;&#x8d77;&#x70b9;   |
| &#x81ea;&#x52a8;&#x5b8c;&#x6210;&#x5e03;&#x7ebf;   |  | &#x81ea;&#x52a8;&#x5b8c;&#x6210;&#x5f53;&#x524d;&#x5e03;&#x7ebf;   |
| &#x5206;&#x5272;&#x5e03;&#x7ebf;   |  | &#x5c06;&#x5e03;&#x7ebf;&#x6bb5;&#x62c6;&#x5206;&#x4e3a;&#x5728;   |
| &#x81ea;&#x5b9a;&#x4e49;&#x7ebf;/&#x8f6c06;&#x5e03;&#x7ebf;  |  | &#x8f6c06;&#x5e03;&#x7ebf;&#x6bb5;&#x62c6;&#x5e03;&#x7ebf;   |
| &#x5dee;&#x5206;&#x5bf9;&#x4e92;&#x50f0;&#x5206;&#x5bf9;&#x4ea4;&#x4e92;&#x5e03;&#x7ebf;   |  | &#x5dee;&#x5206;&#x5bf9;&#x4e92;&#x50f0;&#x5206;&#x5bf9;&#x4ea4;&#x4e92;&#x5e03;&#x7ebf;                         |
| &#x5dee;&#x5206;&#x5bf9;&#x6807;&#x6ce8;...&#x6253;&#x5f00;&#x5dee;&#x5206;&#x5bf9;&#x6807;&#x6ce8;&#x8bbe;                              |  | &#x6253;&#x5f00;&#x5dee;&#x5206;&#x5bf9;&#x6807;&#x6ce8;&#x8bbe;   |
| &#x62d6;&#x52a8;(45 kbd:[D]  |  | &#x62d6;&#x52a8;&#x5e03;&#x7ebf;&#x6bb5;&#xff0c;&#x540c;&#x65f6;&#x45;&#x5ea6;&#x3002;                           |
| &#x62d6;&#x52a8; kbd:[G]   |  | &#x62d6;&#x52a8;&#x5e03;&#x7ebf;&#x4e2d;&#x6700;&#x8fd1;&#x7684;   |
| (&#x81ea;&#x7531;&#x89d2;&#x5ea6;)   |  |  |
| &#x5b8c;&#x6210;&#x5e03;&#x7ebf;   |  | &#x505c;&#x6b62;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x3002;   |
| &#x5e03;&#x7ebf;&#x9ad8;&#x4eae;&#x6a21;&#x8f6c06;&#x5e03;&#x7ebf;&#x5207;&#x6362;&#x5230;&#x9ad8;&#x4eae;                               |  | &#x8f6c06;&#x5e03;&#x7ebf;&#x5207;&#x6362;&#x5230;&#x9ad8;&#x4eae;   |
| &#x5206;&#x5272;&#x5e03;&#x7ebf;   |  | &#x5c06;&#x5e03;&#x7ebf;&#x6bb5;&#x5206;&#x5272;&#x4e3a;&#x5728;   |
| &#x653e;&#x7f6e;&#x76f2;&#x5b54;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef;   |  | &#x5b54;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef;   |
| &#x653e;&#x7f6e;&#x5fae;&#x5b54;[Ctrl+V]   |  | &#x5728;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef;   |
| &#x653e;&#x7f6e;&#x901a;&#x5b54;   |  | &#x5728;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef;   |
| &#x9009;&#x62e9;&#x56fe;&#x5e76;&#x5909;&#x7f6e;&#x5b54;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef;                         |  | &#x9009;&#x62e9;&#x56fe;&#x5e76;&#x5909;&#x7f6e;&#x5b54;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef; |
| &#x9009;&#x62e9;&#x56fe;&#x5e76;&#x5909;&#x7f6e;&#x5b54;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef;                         |  | &#x9009;&#x62e9;&#x56fe;&#x5e76;&#x5909;&#x7f6e;&#x5b54;&#x5f53;&#x524d;&#x5e03;&#x7ebf;&#x7684;&#x672b;&#x7aef; |
| &#x8bbe;&#x7f6e;&#x5c42;&#x5bf9;...&#x66f4;&#x6539;&#x5e03;&#x7ebf;&#x7684;&#x6d3b;&#x52a8;&#x5c42;                                      |  | &#x66f4;&#x6539;&#x5e03;&#x7ebf;&#x7684;&#x6d3b;&#x52a8;&#x5c42;   |
| &#x4ea4;&#x4e92;&#x5f0f;&#x5e03;&#x7ebf;&#x63a8;&#x6324;&#x6a21;&#x50f0;&#x5206;&#x5bf9;&#x4ea4;&#x4e92;&#x5f0f;&#x5e03;&#x7ebf;&#x8bbe; |  | &#x4ea4;&#x4e92;&#x5f0f;&#x5e03;&#x7ebf;&#x63a8;&#x6324;&#x6a21;&#x50f0;&#x5206;&#xbf9;&                         |

[illegible]

## 9.2 3D $\mathbb{R}^3$ Space

The actions below are available in the 3D Viewer. Hotkeys can be assigned to any of these actions in the **Hotkeys** section of the preferences.

|  |  |  |
|--|--|--|
| &#x64cd;&#x4f5c;                                 | &#x9ed8;&#x8ba4;&#x56f4;&#x6077;&#x952e;                         |  |
| &#x6dfb;&#x52a0;&#x57fa;&#x677f;                 | &#x5728;&#x7535;&#x8def;&#x677f;&#x4e0b;&#x9762;&#x6dfb;&#x52a0; |  |
| &#x6297;&#x952f;&#x9f7f;                         | &#x5728;&#x6700;&#x7ec8;&#x6e32;&#x67d3;&#x4e2d;&#x4ee5;&#x6700; |  |
| &#x5207;&#x6362;&#x8868;&#x8d34;                 | &#x5207;&#x6362;“&#x8868;&#x8d34;”&#x5c5e;&#x6027;&#x7684;       |  |
| 3D &#x6a21;&#x578b;                              | 3D &#x6a21;&#x578b;  |  |
| &#x5207;&#x6362;&#x76f4;&#x63d2;                 | &#x5207;&#x6362;“&#x76f4;&#x63d2;”&#x5c5e;&#x6027;&#x7684;       |  |
| 3D &#x6a21;&#x578b;                              | 3D &#x6a21;&#x578b;  |  |
| &#x5207;&#x6362;&#x865a;&#x62df;                 | &#x5207;&#x6362;“&#x865a;&#x62df;”&#x5c5e;&#x6027;&#x7684;       |  |
| 3D &#x6a21;&#x578b;                              | 3D &#x6a21;&#x578b;  |  |
| &#x7ffb;&#x8f6c;&#x7535;&#x8def;&#x677f;         | &#x7ffb;&#x8f6c;&#x7535;&#x8def;&#x677f;&#x89c6;&#x56fe;         |  |
| &#x4e3b;&#x89c6;&#x56fe;                         | &#x4e3b;&#x89c6;&#x56fe;   |  |
| CAD  | &#x6839;&#x636e;&#x6750;&#x8d28;&#x7684;&#x6f2b;&#x53cd;&#x5c04; |  |
| &#x989c;&#x8272;&#x6837;&#x5f0f;                 | CAD &#x989c;&#x8272;&#x6837;&#x5f0f;                             |  |
| &#x4ec5;&#x4f7f;&#x7528;&#x6f2b;&#x53cd;&#x5c04; | &#x4ec5;&#x4f7f;&#x7528; 3D                                      |  |
|  | &#x6a21;&#x578b;&#x6587;&#x4ef6;&#x4e2d;&#x7684;&#x6f2b;&#x53cd; |  |

|   |  |  |
|---|--|--|
| &#x64cd;&#x4f5c;  | &#x9ed8;&#x8ba4;&#x5ff4;&#x6007;&#x952e; |  |
| &#x4f7f;&#x7528;&#x6240;&#x6709;&#x5c5e;&#x602f7f;&#x7528;&#x6bcf;&#x4e2a; 3D |  | &#x6a21;&#x578b;&#x6587;&#x4ef6;&#x4e2d;&#x7684;&#x6240;&#x6709                                |
| &#x4e0b;&#x79fb;&#x7535;&#x8cdd;&#x677f;                                      | &#x4e0b;&#x79fb;&#x7535;&#x8def;&#x677f; |  |
| &#x5de6;&#x79fb;&#x7535;&#x8cdd;&#x677f;                                      | &#x5de6;&#x79fb;&#x7535;&#x8def;&#x677f; |  |
| &#x53f3;&#x79fb;&#x7535;&#x8cdd;&#x677f;                                      | &#x53f3;&#x79fb;&#x7535;&#x8def;&#x677f; |  |
| &#x4e0a;&#x79fb;&#x7535;&#x8cdd;&#x677f;                                      | &#x4e0a;&#x79fb;&#x7535;&#x8def;&#x677f; |  |
| &#x4e0d;&#x663e;&#x793a; 3D &#x7f51;&#x683c;                                  |  | &#x4e0d;&#x663e;&#x793a; 3D &#x7f51;&#x683c;   |
| &#x4e2d;&#x5fc3;&#x8f74;&#x65cb;&#x8f6c;                                      |  | &#x4e2d;&#x5fc3;&#x8f74;&#x65cb;&#x8f6c;<br>(&#x70b9;&#x51fb;&#x9f20;&#x6807;&#x4e2d;&#x952e;) |
| &#x540e;&#x671f;&#x5904;&#x7406;  |  | &#x5728;&#x6700;&#x7ec8;&#x6e32;&#x67d3;&#x4e2d;&#xff0c;&#x5e94;<br>(&#x6162;)                 |
| &#x8fc7;&#x7a0b;&#x7eb9;&#x7406;  |  | &#x5c06;&#x8fc7;&#x7a0b;&#x7eb9;&#x7406;&#x5e94;&#x7528;&#x4e8e;<br>(&#x6162;)                 |
| &#x6e32;&#x67d3;&#x9634;&#x5f71;  |  | &#x6e32;&#x67d3;&#x9634;&#x5f71;   |
| &#x91cd;&#x7f6e;&#x89c6;&#x56fe;  |  | &#x91cd;&#x7f6e;&#x89c6;&#x56fe;   |
| &#x6cbf; Z  | kbd:[Tab]                                |  |
| &#x8f74;&#x65cb;&#x8f6c; 45 &#x5ea6;  |  |  |
| &#x6cbf; X  |  | &#x6cbf; X   |
| &#x8f74;&#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;                              |  | &#x8f74;&#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;   |
| &#x6cbf; X  |  | &#x6cbf; X   |
| &#x8f74;&#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;                              |  | &#x8f74;&#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;   |
| &#x6cbf; Y  |  | &#x6cbf; Y   |
| &#x8f74;&#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;                              |  | &#x8f74;&#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;   |
| &#x6cbf; Y  |  | &#x6cbf; Y   |
| &#x8f74;&#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;                              |  | &#x8f74;&#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;   |
| &#x6cbf; Z  |  | &#x6cbf; Z   |
| &#x8f74;&#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;                              |  | &#x8f74;&#x987a;&#x65f6;&#x9488;&#x65cb;&#x8f6c;   |
| &#x6cbf; Z  |  | &#x6cbf; Z   |
| &#x8f74;&#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;                              |  | &#x8f74;&#x9006;&#x65f6;&#x9488;&#x65cb;&#x8f6c;   |
| 3D &#x7f51;&#x683c; 1 mm  |  | 3D &#x7f51;&#x683c; 1 mm   |
| 3D &#x7f51;&#x683c; 2.5 mm  |  | 3D &#x7f51;&#x683c; 2.5 mm   |
| 3D &#x7f51;&#x683c; 5 mm  |  | 3D &#x7f51;&#x683c; 5 mm   |
| 3D &#x7f51;&#x683c; 10mm  |  | 3D &#x7f51;&#x683c; 10 mm  |
| &#x663e;&#x793a; 3D &#x8f74;&#x7ebf;  |  | &#x663e;&#x793a; 3D &#x8f74;&#x7ebf;   |
| &#x663e;&#x793a;&#x6a21;&#x578b;&#x8fb9;&#x6846;                              |  | &#x663e;&#x793a;&#x6a21;&#x578b;&#x8fb9;&#x6846;   |
| &#x53cd;&#x5c04;  |  | &#x5728;&#x6700;&#x7ec8;&#x6e32;&#x67d3;&#x4e2d;&#x6e32;&#x67d3<br>(&#x6162;)                  |
| &#x6298;&#x5c04;  |  | &#x5728;&#x6700;&#x7ec8;&#x6e32;&#x67d3;&#x4e2d;&#x6e32;&#x67d3<br>(&#x6162;)                  |
| &#x5207;&#x6362;&#x80f6;&#x7c98;&#x5c42;&#x6632;&#x793a;&#x663e;&#x793a       |  | &#x5207;&#x6362;&#x80f6;&#x7c98;&#x5c42;&#x6632;&#x793a;&#x663e;&#x793a                        |
| &#x5207;&#x6362;&#x7535;&#x8def;&#x677f;&#x663e;&#x793a;                      |  | &#x5207;&#x6362;&#x7535;&#x8def;&#x677f;&#x663e;&#x793a;                                       |
| &#x5207;&#x6362;&#x6ce8;&#x91ca;&#x548c;&#x7ed8;&#x56fe;&#x5c42;              |  | &#x5207;&#x6362;&#x6ce8;&#x91ca;&#x548c;&#x7ed8;&#x56fe;&#x5c42;                               |
| &#x5207;&#x6362; ECO &#x5c42;&#x663e;&#x793a;                                 |  | &#x5207;&#x6362; ECO &#x5c42;&#x663e;&#x793a;  |
| &#x5207;&#x6362;&#x6b63;&#x4ea4;&#x6295;&#x5f71f;                             |  | &#x5207;&#x6362;&#x6b63;&#x4ea4;&#x6295;&#x5f71f;  |
| &#x5207;&#x6362;&#x771f;&#x5b9e;&#x6a21;&#x5f0f;                              |  | &#x5207;&#x6362;&#x771f;&#x5b9e;&#x6a21;&#x5f0f;   |
| &#x5207;&#x6362;&#x4e1d;&#x5370;&#x5c42;&#x6632;&#x4e1d;                      |  | &#x5207;&#x6362;&#x4e1d;&#x5370;&#x5c42;&#x6632;&#x4e1d;                                       |
| &#x5207;&#x6362;&#x963b;&#x710a;&#x5c42;&#x6632;&#x963b;&#x710a;              |  | &#x5207;&#x6362;&#x963b;&#x710a;&#x5c42;&#x6632;&#x963b;&#x710a;                               |
| &#x5207;&#x6362;&#x9521;&#x818f;&#x5c42;&#x6632;&#x9521;&#x818f;              |  | &#x5207;&#x6362;&#x9521;&#x818f;&#x5c42;&#x6632;&#x9521;&#x818f;                               |



|  |  |
|--|--|
| &#x64cd;&#x4f5c;   | &#x9ed8;&#x8ba4;&#x5668;&#x6007;&#x952e; |
| &#x5207;&#x6362;&#x533a;&#x57df;&#x663e;&#x793d;&#x5207;&#x6362;&#x533a;&#x57df;&#x663e;&#x793a; |  |
| &#x540e;&#x89c6;&#x56fe;   | kbd:[Shift+Y]                            |
| &#x5e95;&#x89c6;&#x56fe;   | kbd:[Shift+Z]                            |
| &#x6b63;&#x89c6;&#x56fe;   | kbd:[Y]                                  |
| &#x5de6;&#x89c6;&#x56fe;   | kbd:[Shift+X]                            |
| &#x53f3;&#x89c6;&#x56fe;   | kbd:[X]                                  |
| &#x9876;&#x89c6;&#x56fe;   | kbd:[Z]                                  |

## 9.3 Common

The actions below are available across KiCad, including in Pcbnew. Hotkeys can be assigned to any of these actions in the **Hotkeys** section of the preferences.

| Action                             | Default Hotkey    | Description  |
|------------------------------------|-------------------|--|
| Exclude Marker                     |                   | Mark current violation in Checker window as an exclusion |
| Next Marker                        |                   | Go to next marker in Checker window                      |
| Previous Marker                    |                   | Go to previous marker in Checker window                  |
| Add Library...                     |                   | Add an existing library folder                           |
| Click                              | kbd:[Return]      | Performs left mouse button click                         |
| Double-click                       | kbd:[End]         | Performs left mouse button double-click                  |
| Cursor Down                        | kbd:[Down]        |  |
| Cursor Down Fast                   | kbd:[Ctrl+Down]   |  |
| Cursor Left                        | kbd:[Left]        |  |
| Cursor Left Fast                   | kbd:[Ctrl+Left]   |  |
| Cursor Right                       | kbd:[Right]       |  |
| Cursor Right Fast                  | kbd:[Ctrl+Right]  |  |
| Cursor Up                          | kbd:[Up]          |  |
| Cursor Up Fast                     | kbd:[Ctrl+Up]     |  |
| Switch to Fast Grid 1              | kbd:[Alt+1]       |  |
| Switch to Fast Grid 2              | kbd:[Alt+2]       |  |
| Switch to Next Grid                | kbd:[N]           |  |
| Switch to Previous Grid            | kbd:[Shift+N]     |  |
| Grid Properties...                 |                   | Set grid dimensions                                      |
| Reset Grid Origin                  | kbd:[Z]           |  |
| Grid Origin                        | kbd:[S]           | Set the grid origin point                                |
| Inactive Layer View Mode           |                   | Toggle inactive layers between normal and dimmed         |
| Inactive Layer View Mode (3-state) | kbd:[H]           | Cycle inactive layers between normal, dimmed, and hidden |
| Inches                             |                   | Use inches   |
| Millimeters                        |                   | Use millimeters  |
| Mils                               |                   | Use mils   |
| New...                             | kbd:[Ctrl+N]      | Create a new document in the editor                      |
| New Library...                     |                   | Create a new library folder                              |
| Open...                            | kbd:[Ctrl+O]      | Open existing document                                   |
| Page Settings...                   |                   | Settings for paper size and title block info             |
| Pan Down                           | kbd:[Shift+Down]  |  |
| Pan Left                           | kbd:[Shift+Left]  |  |
| Pan Right                          | kbd:[Shift+Right] |  |
| Pan Up                             | kbd:[Shift+Up]    |  |
| Pin Library                        |                   | Keep the library at the top of the list                  |
| Plot...                            |                   | Plot   |
| Print...                           | kbd:[Ctrl+P]      | Print  |

| Action                       | Default Hotkey     | Description   |
|------------------------------|--------------------|---|
| Quit                         |                    | Close the current editor                                |
| Reset Local Coordinates      | kbd:[Space]        |   |
| Revert                       |                    | Throw away changes                                      |
| Save                         | kbd:[Ctrl+S]       | Save changes  |
| Save All                     |                    | Save all changes  |
| Save As...                   | kbd:[Ctrl+Shift+S] | Save current document to another location               |
| Save Copy As...              |                    | Save a copy of the current document to another location |
| 3D Viewer                    | kbd:[Alt+3]        | Show 3D viewer window                                   |
| Show Context Menu            |                    | Perform the right-mouse-button action                   |
| Footprint Library Browser    |                    | Browse footprint libraries                              |
| Footprint Editor             |                    | Create, delete and edit footprints                      |
| Symbol Library Browser       |                    | Browse symbol libraries                                 |
| Symbol Editor                |                    | Create, delete and edit symbols                         |
| Always Show Cursor           | kbd:[Ctrl+Shift+X] | Display crosshairs even in selection tool               |
| Full-Window Crosshairs       |                    | Switch display of full-window crosshairs                |
| Show Grid                    |                    | Display grid dots or lines in the edit window           |
| Polar Coordinates            |                    | Switch between polar and cartesian coordinate systems   |
| Switch units                 | kbd:[Ctrl+U]       | Switch between imperial and metric units                |
| Unpin Library                |                    | No longer keep the library at the top of the list       |
| Update PCB from Schematic... | kbd:[F8]           | Update PCB with changes made to schematic               |
| Update Schematic from PCB... |                    | Update schematic with changes made to PCB               |
| Center                       | kbd:[F4]           | Center  |
| Zoom to Objects              | kbd:[Ctrl+Home]    | Zoom to Objects   |
| Zoom to Fit                  | kbd:[Home]         | Zoom to Fit   |
| Zoom In at Cursor            | kbd:[F1]           | Zoom In at Cursor                                       |
| Zoom In                      |                    | Zoom In   |
| Zoom Out at Cursor           | kbd:[F2]           | Zoom Out at Cursor                                      |
| Zoom Out                     |                    | Zoom Out  |
| Refresh                      | kbd:[F5]           | Refresh   |
| Zoom to Selection            | kbd:[Ctrl+F5]      | Zoom to Selection                                       |
| Cancel                       |                    | Cancel current tool                                     |
| Change Edit Method           | kbd:[Ctrl+Space]   | Change edit method constraints                          |
| Copy                         | kbd:[Ctrl+C]       | Copy selected item(s) to clipboard                      |
| Cut                          | kbd:[Ctrl+X]       | Cut selected item(s) to clipboard                       |
| Delete                       | kbd:[Del]          | Deletes selected item(s)                                |
| Interactive Delete Tool      |                    | Delete clicked items                                    |
| Duplicate                    | kbd:[Ctrl+D]       | Duplicates the selected item(s)                         |
| Find                         | kbd:[Ctrl+F]       | Find text   |
| Find and Replace             | kbd:[Ctrl+Alt+F]   | Find and replace text                                   |
| Find Next                    | kbd:[F3]           | Find next match   |
| Find Next Marker             | kbd:[Shift+F3]     |   |
| Paste                        | kbd:[Ctrl+V]       | Paste item(s) from clipboard                            |
| Paste Special...             |                    | Paste item(s) from clipboard with options               |
| Redo                         | kbd:[Ctrl+Y]       | Redo last edit  |
| Replace All                  |                    | Replace all matches                                     |
| Replace and Find Next        |                    | Replace current match and find next                     |
| Select All                   | kbd:[Ctrl+A]       | Select all items on screen                              |
| Undo                         | kbd:[Ctrl+Z]       | Undo last edit  |
| Measure Tool                 | kbd:[Ctrl+Shift+M] | Interactively measure distance between points           |
| Select item(s)               |                    | Select item(s)  |

| Action                        | Default Hotkey | Description   |
|-------------------------------|----------------|---|
| Configure Paths...            |                | Edit path configuration environment variables             |
| Donate                        |                | Open "Donate to KiCad" in a web browser                   |
| Get Involved                  |                | Open "Contribute to KiCad" in a web browser               |
| Getting Started with KiCad    |                | Open "Getting Started in KiCad" guide for beginners       |
| Help                          |                | Open product documentation in a web browser               |
| List Hotkeys...               | kbd:[Ctrl+F1]  | Displays current hotkeys table and corresponding commands |
| Preferences...                | kbd:[Ctrl+,]   | Show preferences for all open tools                       |
| Report Bug                    |                | Report a problem with KiCad                               |
| Manage Footprint Libraries... |                | Edit the global and project footprint library lists       |
| Manage Symbol Libraries...    |                | Edit the global and project symbol library lists          |