PCB 编辑器

The KiCad Team

REVISION HISTORY

NUMBER	DATE	DESCRIPTION	NAME

Contents

1 Pcbnew 简介 1

1.1 初始配置 . . . 1

1.2 Pcbnew 用户界面 . . . 2

1.3 导航编辑画布 . . . 2

1.4 快捷键 . . . 3

2 显示和选择控件 4

2.1 板层 . . . 4

2.1.1 电路板层的显示顺序 . . . 4

2.2 外观面板 . . . 4

2.2.1 图层控件 . . . 4

2.2.2 对象控件 . . . 5

2.2.3 图层预设 . . . 5

2.2.4 网络和网络类控件 . . . 5

2.3 选择和选择筛选器 . . . 6

2.4 网络高亮 . . . 6

2.5 从原理图交叉探测 . . . 7

2.6 左侧工具栏显示控件 . . . 7

3 创建 PCB 9

3.1 基本 PCB 概念 . . . 9

3.2 性能 . . . 9

3.3 从原理图开始 . . . 9

3.3.1 选项 . . . 10

3.4 从头开始 . . . 11

3.5 电路板设置 . . . 11

3.5.1 配置电路板压层和物理参&#x657

3.5.2 配置默认文本和图形设置 12

3.5.3 配置设计规则 . . . 13

3.5.3.1 约束 . . . 13

3.5.3.2 预定义大小 . . . 14

3.5.3.3 网络类 . . . 15
3.5.3.4 自定义规则 . . . 15
3.5.3.5 违规严重程度 . . . 16
3.5.3.6 正在导入设置 . . . 16

4 编辑电路板 18

4.1 放置和绘制操作 . . . 18
4.2 捕捉 . . . 19
4.3 编辑对象属性 . . . 19
4.4 使用封装 . . . 20
4.5 使用焊盘 . . . 20
4.6 使用区域 . . . 22
4.7 图形对象 . . . 24
4.7.1 正在创建图形形状 . . . 24
4.7.2 正在创建文本对象 . . . 25
4.7.3 电路板轮廓 (边缘切割) . . . 26
4.8 标注 . . . 26
4.8.1 标注格式选项 . . . 28
4.8.2 标注文本选项 . . . 28
4.8.3 标注线选项 . . . 29
4.8.4 引线选项 . . . 29
4.9 布线 . . . 29
4.9.1 布线形态 . . . 30
4.9.2 布线转角模式 . . . 31
4.9.3 布线宽度 . . . 31
4.9.4 放置过孔 . . . 32
4.9.5 差分对布线 . . . 32
4.9.6 修改布线 . . . 33
4.9.7 长度调整 . . . 33
4.9.8 交互式布线设置 . . . 34
4.10 向前和向后批注 . . . 36
4.10.1 位置重新批注 . . . 36
4.11 锁定 . . . 36
4.12 批量编辑工具 . . . 37
4.13 清理工具 . . . 37
4.14 正在导入图形 . . . 37
4.14.1 从 DXF 和 SVG 文件导入矢量图 . . . 37
4.14.2 正在导入位图图像 . . . 37

5 检查电路板 38

5.1 测量工具 . . . 38

5.2 设计规则检查 . . . 38

5.2.1 间隙和约束解析 . . . 40

5.3 Find tool . . . 42

5.4 3D 查看器 . . . 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 网络检查 . . . 45

6 生成输出 47

6.1 制造输出和绘制 . . . 47

6.1.1 绘制选项 . . . 48

6.1.2 Gerber 选项 . . . 49

6.1.3 Postscript 选项 . . . 49

6.1.4 SVG 选项 . . . 49

6.1.5 DXF 选项 . . . 50

6.1.6 HPGL 选项 . . . 50

6.2 钻孔文件 . . . 50

6.3 元件放置文件 . . . 51

6.4 额外的制造产出 . . . 52

6.5 打印 . . . 52

6.6 正在导出文件 . . . 53

6.6.1 IDF Exporter . . . 53

7 封装和封装库 55

7.1 管理封装库 . . . 55

7.1.1 Initial Configuration . . . 56

7.1.2 Managing Table Entries . . . 56

7.1.3 Environment Variable Substitution . . . 57

7.1.4 Using the GitHub Plugin . . . 57

7.2 创建和编辑封装 . . . 57

7.2.1 自定义焊盘形状 . . . 57

7.2.2 封装属性 . . . 57

7.2.3 封装向导 . . . 58

8 高级主题 59

8.1 配置和自定义 . . . 59

8.1.1 显示选项 . . . 59

8.1.2 编辑选项 . . . 60

8.1.3 颜色 . . . 61

8.1.4 操作插件 . . . 62

8.1.5 原点和轴 . . . 62

8.2 自定义设计规则 . . . 63

8.2.1 自定义规则编辑器 . . . 63

8.2.2 自定义规则语法 . . . 63

8.2.2.1 图层子句 . . . 64

8.2.2.2 条件子句 . . . 64

8.2.2.3 约束 . . . 65

8.2.3 对象属性和函数参考 . . . 67

8.2.3.1 常见属性 . . . 67

8.2.3.2 连接的对象属性 . . . 68

8.2.3.3 封装属性 . . . 68

8.2.3.4 焊盘属性 . . . 68

8.2.3.5 布线和圆弧属性 . . . 69

8.2.3.6 过孔属性 . . . 69

8.2.3.7 覆铜和规则区域属性 . . . 69

8.2.3.8 图形形状属性 . . . 70

8.2.3.9 文本属性 . . . 70

8.2.3.10 表达式函数 . . . 70

8.2.4 自定义设计规则示例 . . . 71

8.3 脚本 . . . 72

8.3.1 Python 脚本位置 . . . 72

8.3.2 操作插件 . . . 72

8.3.3 封装向导 . . . 72

8.3.4 使用脚本控制台 . . . 73

8.3.5 编写外部脚本 . . . 73

8.3.6 编写操作插件 . . . 73

8.4 Working With IDF Component Outlines . . . 73

8.4.1 Specifying component models for use by the exporter . . . 73

8.4.2 Creating a component outline file . . . 75

8.4.3 Guidelines for creating outlines . . . 76

8.4.3.1 Package naming . . . 76

8.4.3.2 Comments . . . 76

8.4.3.3 Geometry and Part Number entries . . . 76

8.4.3.4 Pin orientation and positioning . . . 76
8.4.3.5 Tips on dimensions . . . 77
8.4.4 IDF Component Outline Tools . . . 77
8.4.4.1 idfcyl . . . 77
8.4.4.2 idfrect . . . 78
8.4.4.3 dxf2idf . . . 79
8.4.4.4 idf2vrml . . . 80

9 操作参考 81

9.1 PCB 编辑器 . . . 81
9.2 3D 查看器 . . . 86
9.3 Common . . . 88

参考手册

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.

Copyright

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.

本指南中的所有商标均属于其&#x

Contributors

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

翻译人员

taotieren <admin@taotieren.com>, 2019, 2020, 2021.

Telegram 简体中文交流群: https://t.me/KiCad_zh_CN

译者注：英文双引号包含的中&#x65

Feedback

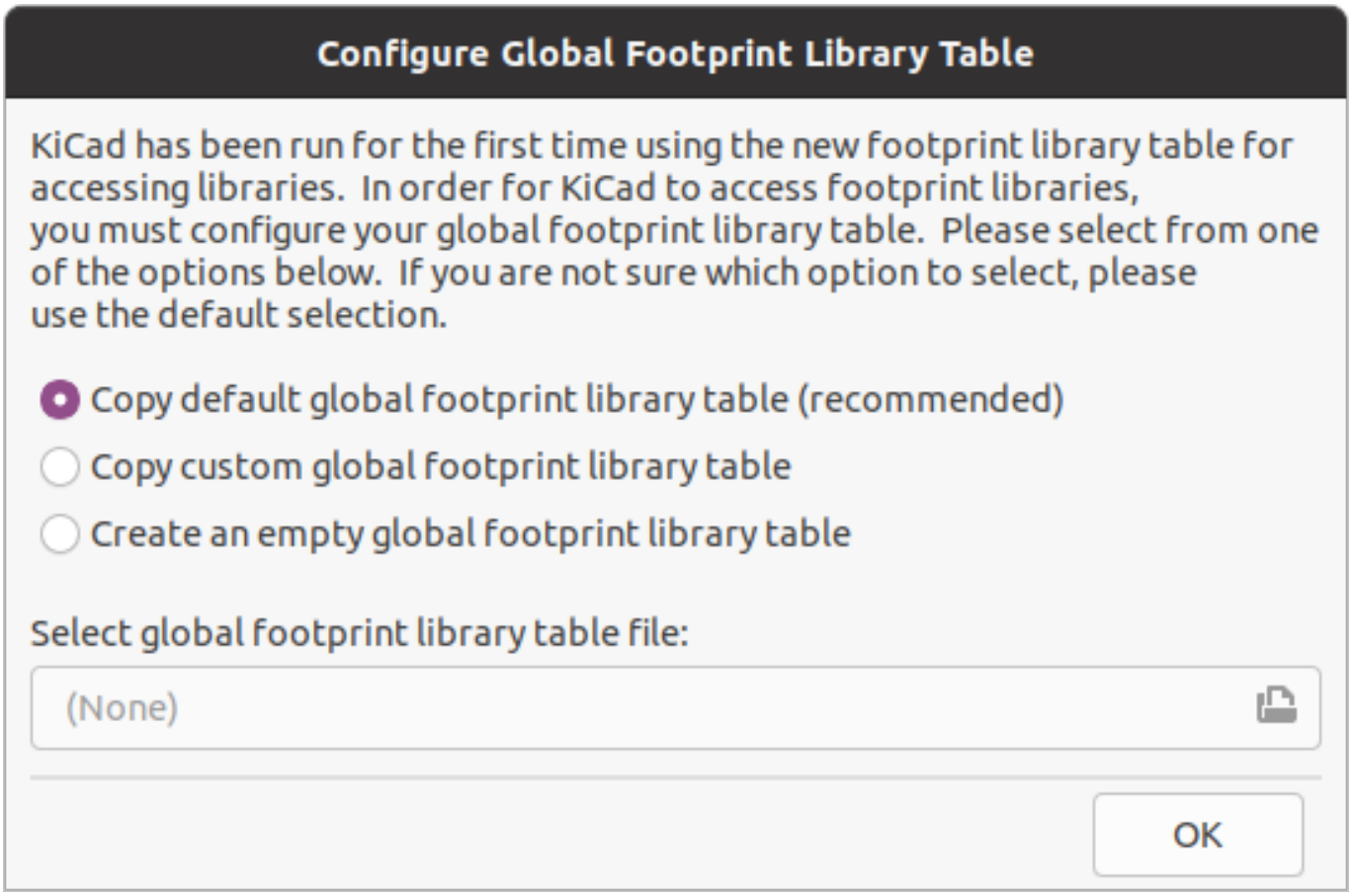
The KiCad project welcomes feedback, bug reports, and suggestions related to the software or its documentation. For more information on how to sumbit feedback or report an issue, please see the instructions at https://www.kicad.org/help/report-an-issue/

Chapter 1

Pcbnew 简介

1.1 初始配置

当 Pcbnew 第一次运行时，如果在 KiCad 配
`fp-lib-table`，那么 Pcbnew 将询问如何创建这



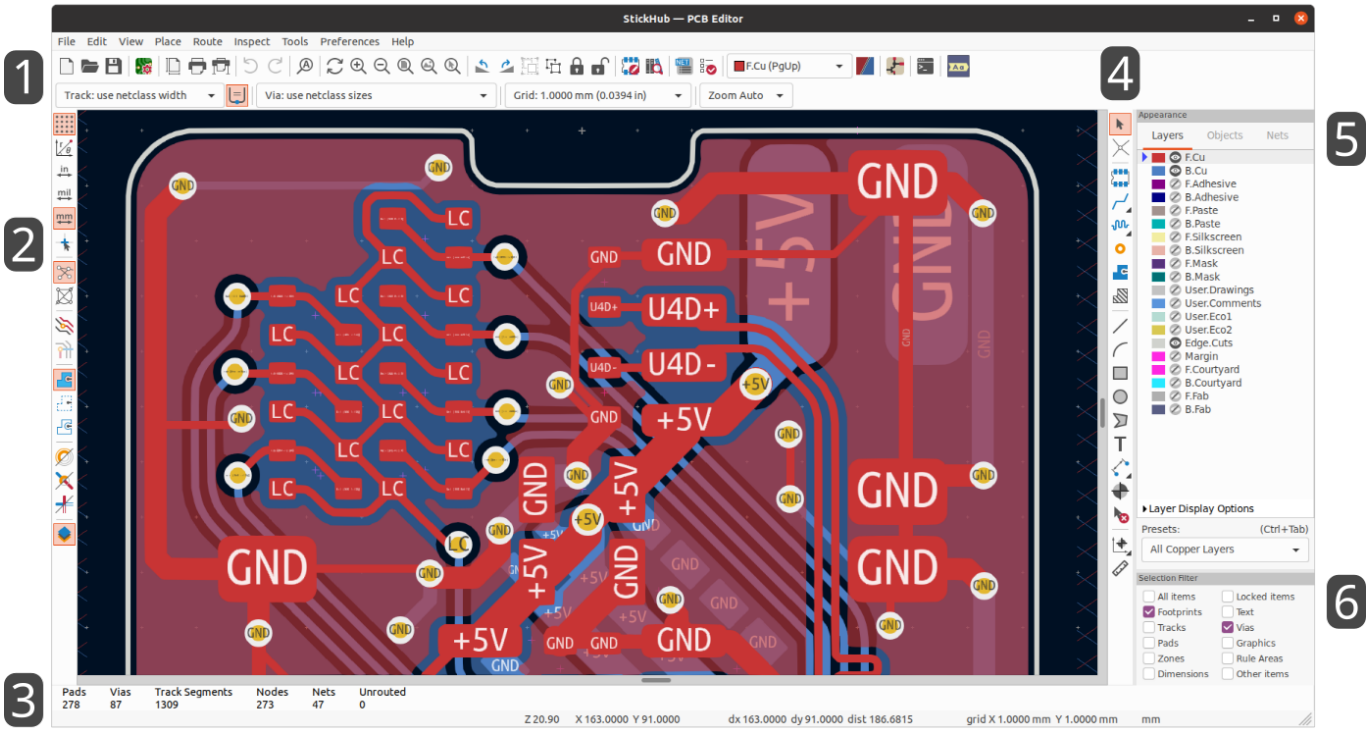
The first option is recommended (**Copy default global footprint library table (recommended)**). The default footprint library table includes all of the standard footprint libraries that are installed as part of KiCad.

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 用户界面



主 Pcbnew 用户界面如上所示，其中&#x63

1. 顶部工具栏 (文件管理、缩放&#x5de
2. 左侧工具栏 （显示选项）
3. 消息面板和状态栏
4. 右侧工具栏 (绘图和设计工具)
5. 外观面板
6. 选择过滤器面板

1.3 导航编辑画布

编辑画布是正在设计的板上的&#x8

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 快捷键

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
通过热键可用的许多操作也可在&#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

显示和选择&#x6

2.1 板层

Pcbnew 中的层表示线路板上的物理&#x94c
在编辑器中始终有一个处于活&#x5
活动图层被画在其他层之上，&#x5e
活动图层在顶部工具栏的层选&#x6
若要变更活动图层，可以左键&#x53
可以隐藏图层以简化电路板视&#x56

2.1.1 电路板层的显示顺序

Note
TODO：写下这一节。

2.2 外观面板

外观面板提供用于管理 Pcbnew 绘图&#x75

2.2.1 图层控件

在外观面板的图层选项卡中，&#x6
活动图层在色块的左边有一个&#x7
左键点击一个图层来选择它作&#x4
左键单击相应的可见性图标，&#x57
双击或中击色块来改变该图层&#x7

Note
必须先在首选参数中创建自定义&
然后才能在“外观”面板中更改图层&

在图层列表下方是一个包含图&#x5
"暗显" 或 "隐藏" 时，不能选择非活&
kbd:[Ctrl+H] 快速切换这些显示模式。

Flip board view will show the board as if you are looking from the bottom (that is, mirrored around the Y-axis). This option is also available in the View menu.

Note
翻转电路板视图不会更改可视层&#x98

2.2.2 对象控件

外观面板的 "对象" 选项卡与 "图层"
选项卡类似。 主要区别在于，&#x6
这里的不透明度设置将与图层&#x98
默认情况下，所有对象都是完&#x51

2.2.3 图层预设

图层预设存储了哪些图层和对&#x8
有几个内置的图层预设，您可&#x4e
自定义预设存储在一个电路板&#x7

要加载一个预设，请从外观面&#x67
kbd:[Ctrl] 并按 kbd:[Tab] 来使用快速切换器。
一旦快速切换器窗口出现，你&#x53
kbd:[Tab] 和 kbd:[Shift+Tab] 来循环浏览可用的预&#x8bbe
当你放开 kbd:[Ctrl] 键时，高亮显示的&#x98

要保存一个自定义的预设，首&#x5
给您的预设一个名字，它现在&#x5
要修改一个自定义预设，请遵&#x5f
要删除一个自定义预设，从下&#x62

2.2.4 网络和网络类控件

外观面板的网络选项卡显示电&#x8

每个网络和网络类还可以指定&#x4e0
(或网络类中的所有网络) 的飞线&#x

Note
默认网络类不能分配颜色，
因为该类中的网络将仅使用颜色&#

您还可以通过外观面板选择并&#x9a

网络类列表下面是一个包含网&#x7
(焊盘、布线、过孔和区域) 都将&#x
当选择没有”时，网络和网络类&#x9

第二个选项控制如何绘制飞线&#x3

2.3 选择和选择筛选器

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.

可以通过在单击或拖动的同时&#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.

拖动以执行选框时，将应用以&#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 窗口右下角的选择过滤器面&#x67

当一个连接的铜线项目被选中&#x6
"扩展选择" 命令或热键 kbd:[U] 将选择&#
第一次运行这个命令时，选择&#x5c
第二次，选择将被扩展到所有&#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.

按 kbd:[Esc] 将始终取消当前工具或操&#x
kbd:[Esc] 将清除当前选择。

2.4 网络高亮

电气网络（或一组网络）可以&#x572
PCB 编辑器中被高亮显示，以显示&
PCB 上布线的。 通过在 PCB 编辑器中&#x
当网络高亮激活时，高亮的网&#x621

有三种方法可以点击一个或多&#x4
PCB 编辑器中高亮：点击铜对象后&
kbd:[`] ，使用任何铜对象的上下文菜
(热键 kbd:[~]) 清除高亮。

选择一个或多个网络进行高亮&#x65
kbd:[Ctrl+`] 访问)。此操作将打开或关闭&

2.5 从原理图交叉探测

KiCad 允许在原理图和 PCB 之间进行双&#
有几种不同类型的交叉探测。

Selection cross-probing allows you to select a symbol or pin in the schematic to select the corresponding footprint or pad in the PCB (if one exists) and vice-versa. By default, cross-probing will result in the display centering on the cross-probed item and zooming to fit. This behavior can be disabled in the Display Options section of the Preferences dialog.

Highlight cross-probing allows you to highlight a net in the schematic and PCB at the same time. If the option "Highlight cross-probed nets" is enabled in the Display Options section of the Preferences dialog, highlighting a net or bus in the schematic editor will cause the corresponding net or nets to be highlighted in the PCB editor.

2.6 左侧工具栏显示控件

左侧工具栏提供了更改 Pcbnew 中项&#x76

	打开/关闭栅格显示。 **注意：** 默认情况下，隐藏网格将禁&#x 可以在偏好设置的显示选项&#x
	在状态栏中的极坐标和笛卡&#x
	以英寸、密耳或毫米为单位&#x
	在全屏和小编辑光标 (十字光标) 之间切换。
	打开/关闭飞线显示。
	在直线型和弧线型飞线之间&#x
	在正常和暗显之间切换非活&# **注意：** 当非活动层显示模式为暗显&#x 在这两种情况下，按下按钮&#x 隐藏模式只能通过外观面板&#x kbd:[Ctrl+H] 进入。
	选择要高亮的网络时，会打&#x **注意：** 当没有高亮任何网络时，此&#x6 要高亮网络，可使用热键 kbd:[`]， 右击网络中的任何铜对象并&#x 或右击外观面板的网络选项&#x

	显示分区填充区域。
	仅显示区域轮廓。
	将分区填充区域显示为轮廓&#x
	在填充模式和轮廓模式之间&#x
	在填充模式和轮廓模式之间&#x
	在填充模式和轮廓模式之间&#x
	显示敷铜填充区域。
	仅显示敷铜轮廓。
	将敷铜填充区域显示为轮廓&#x
	在填充模式和轮廓模式之间&#x
	在填充模式和轮廓模式之间&#x
	在填充模式和轮廓模式之间&#x
	显示或隐藏编辑器右侧的外&#x

Chapter 3

创建 PCB

3.1 基本 PCB 概念

KiCad 中的印刷电路板通常由代表&#x753
封装、定义这些焊盘如何彼此&#x8
网络、形成每个网络中焊盘之&#x9
布线、**过孔** 和 **填充区** 以及定义&

KiCad 通常会将 PCB 上的网络信息与相&#x
PCB 编辑器中创建和编辑网络。

3.2 性能

KiCad 能够创建多达 32 个铜层、14 个技&#
(丝印、阻焊、元件粘合剂、焊&#
和 13 个通用绘图层的印刷电路板&

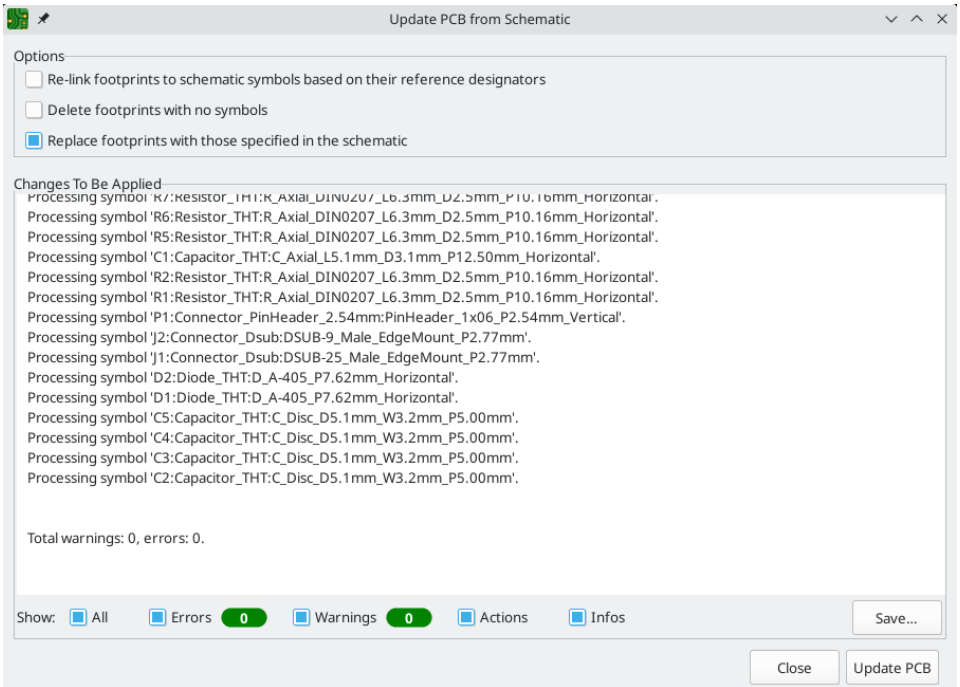
The internal measurement resolution of all objects in KiCad is 1 nanometer, and measurements are stored as 32-bit integers. This means it is possible to create boards up to approximately 4 meters by 4 meters.

KiCad 目前支持每个工程/原理图一&#x4e2a

3.3 从原理图开始

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 选项

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 从头开始

也可以创建没有匹配原理图的&#x75
PCB 编辑器(而不是从 KiCad 工程管理器&#
"另存为..." 从文件菜单选择保存电

3.5 电路板设置

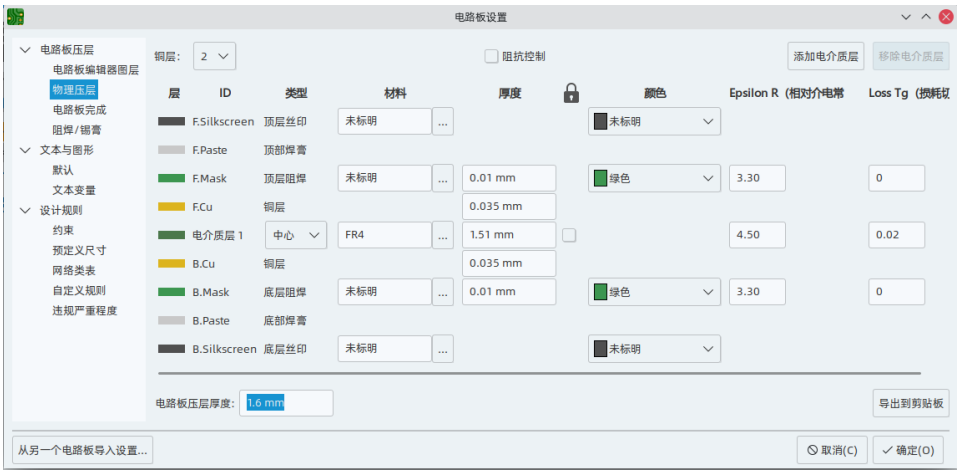
在开始您的线路板设计之前，&#x4f7


图标或选择 "电路板设置..." 从文&#x

3.5.1 配置电路板压层和物理&#x53c

在电路板设置中有两个部分用&#x4
(非铜) 层，并在需要时为层指定&#x

要配置电路板的压层，从物理&#x53



在左上角设置铜层的数量，然&#x54
这些参数可以保留其默认值，&#x4f
3D 模型时，将使用电路板的厚度&#x

Note
KiCad 目前仅支持铜层数为偶数的压&#x
要创建具有奇数层的设计
(例如，柔性印刷电路板或金属芯&#x
只需选择下一个最高的偶数，而&

接下来，如果需要，可以使用&#x753
`B.Silkscreen` 层旁边的复选框。



Note

在电路板编辑器层部分，可以将&#x9
本指南仅供用户参考。 无论在此&#

在电路板编辑器对话框的电路&#x67
(如镶边或边缘电镀) 的设置。请&#x
Gerber 作业文件一部分的电路板属&#x6027

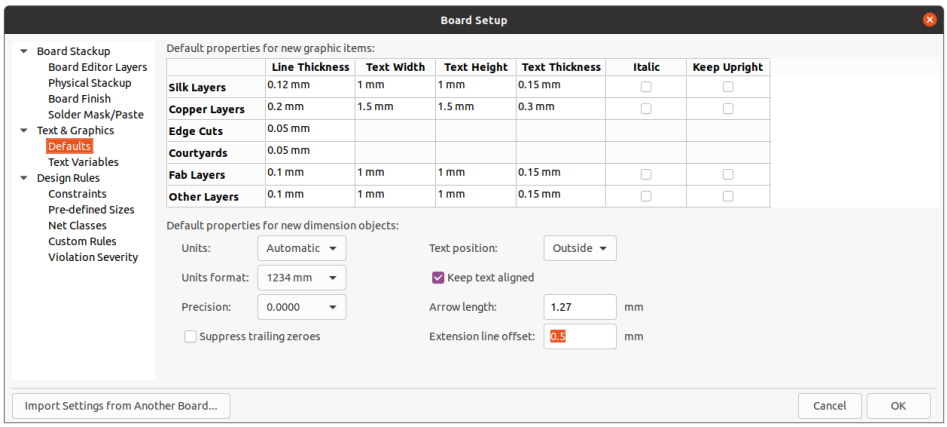
阻焊/锡膏部分允许全局调整电&#x
这些值将被添加到在个别封装&#x6
正的间隙值将导致阻焊层或锡&#x8
更大。 负的间隙值将导致开口&#x6b
更小。

Warning

大多数商业 PCB 制造商希望这些&#x5
CAM 过程中自行调整阻焊和粘贴&#x5f
通常最好将这些值保留为默认&#
除非您自己制作 PCB， 或者您的&#x523

3.5.2 配置默认文本和图形设&#x7f6

电路板设置对话框的文本和图&#x5f



可以为对话框中显示的六种不&#x5

文本替换变量可以在文本变量&#x90
这些变量允许你将变量名称替&#x63
这种替换发生在变量名称在 ${VARIABLEN
的变量替换语法内的任何地方&#x30

例如，您可以创建一个名为 VERSION
的变量，并将文本替换设置为 1.0&#
现在，在 PCB 上的任何文本对象中&
${VERSION}，KiCad 将替代 1.0’。 如果你把&#x66
`2.0’，每个包括 `${VERSION} 的文本对&#x8c61
你也可以混合使用普通文本和&#x53
例如，你可以创建一个文本对&#x8c6
版本：${VERSION}，它将被替换为 版&#

3.5.3 配置设计规则

设计规则控制交互式布线器的&#x8

3.5.3.1 约束

基本设计规则是在电路板设置&#x5b
这一部分的约束条件适用于整&#x4
这里设置的任何最小值都是一&#x4e2
绝对(absolute) 的最小值，不能被更具&#x4f
例如，如果你需要电路板的一&#x90
0.2mm，其余部分为 0.3mm，你必须在约&#
0.2mm 的最小铜间隙，并使用网络类
0.3mm 间隙。



除了设置最小间隙外，还可以&#x572

设置	描述
允许盲&#x[illegible]	启用此设置，然& 盲孔是机械钻孔，从外& 埋孔是机械钻孔，在内

设置	描述
允许微调	孔位用布线器放置微孔& 微孔是典型的激光钻孔 KiCad 支持单独的微孔尺寸限
由线段组成的轨&#x8ff9

3.5.3.3 网络类

网络类部分允许你为不同网类&#x9
在 KiCad 中，每个网络都是一个网络&
如果你不把一个网络添加到一&#x4
飞线可以通过原理图设置和电&#x8d
PCB 编辑器中添加到网络类。



网络类部分的上部包含一个表&#x6
每个网络类都有铜间隙、布线&#x5b
这些值将在创建布线和过孔时&#x4f

Note

任何规则都不能覆盖电路板设置&#x7
例如，如果您将网络类间距设置&#x4
0.1 mm，但约束条件部分中的最小间&#x8
0.2 mm， 则该类网络的间距将为 0.2
mm。

当 PCB 编辑器中的布线宽度和过孔&
"使用网络类值" 时，就会使用为&#x6
这些宽度和尺寸被认为是该网&#x7e
它们不是最小或最大值。 手动&#x
DRC。 要将布线宽度或过孔大小限

3.5.3.4 自定义规则

自定义规则部分包含一个文本&#x
自定义规则用于创建基本约束&#x62

只有在自定义规则定义中没有&#x9
在关闭电路板设置之前，使用&#x68

请参阅高级主题一章中的自定&#x4



3.5.3.5 违规严重程度

违规严重性部分允许你配置每&#x7
每条规则可以被设置为创建一&#x4e

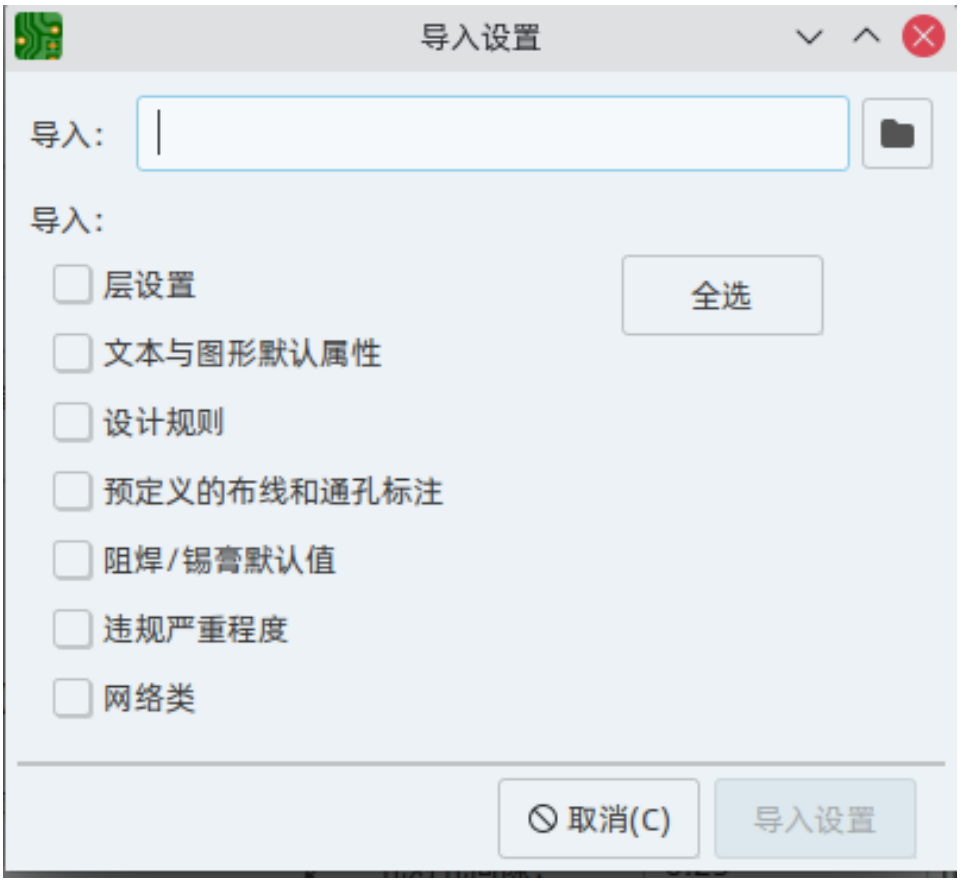
Note

在设计规则检查器中可能会忽略&#x
在违规程度部分中将规则设置为&
请谨慎使用此设置。



3.5.3.6 正在导入设置

您可以从现有电路板导入部分&#x62
"模板" 电路板，然后将这些设置&#x



要导入设置，请点击电路板设&#x7f6
从另一个电路板导入设置... 按钮&#
`kicad_pcb` 文件。 选择你想导入的设置&

Chapter 4

编辑电路板

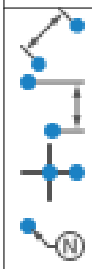
4.1 放置和绘制操作

放置和绘图工具位于右边的工&#x51
当一个工具被激活时，它将一&#x76
kbd:[Esc] 键取消该工具。 当任何其他&#x

某些工具栏按钮在调色板中有&#x

要显示调色板，你可以在工具&#x4e
调色板关闭时将显示最近使用&#x76

	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. **Note:** 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. **Note:** 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 捕捉

移动、拖动和绘制电路板元素&#x6

快捷键	**效果**
kbd:[Ctrl]	关闭网格捕捉。
kbd:[Shift]	关闭对象捕捉。

4.3 编辑对象属性

所有对象都有可在对话框中编&#x8
使用热键 kbd:[E] 或从右键菜单中选&#x62

只有当你选择的所有项目都是&#x5
要想一次编辑不同类型的项目&#x7

在属性对话框中，任何包含数&#x5b
例如，一个尺寸可以被输入为 2
* 2mm，结果是 4mm 的值。 支持基本算&#

4.4 使用封装

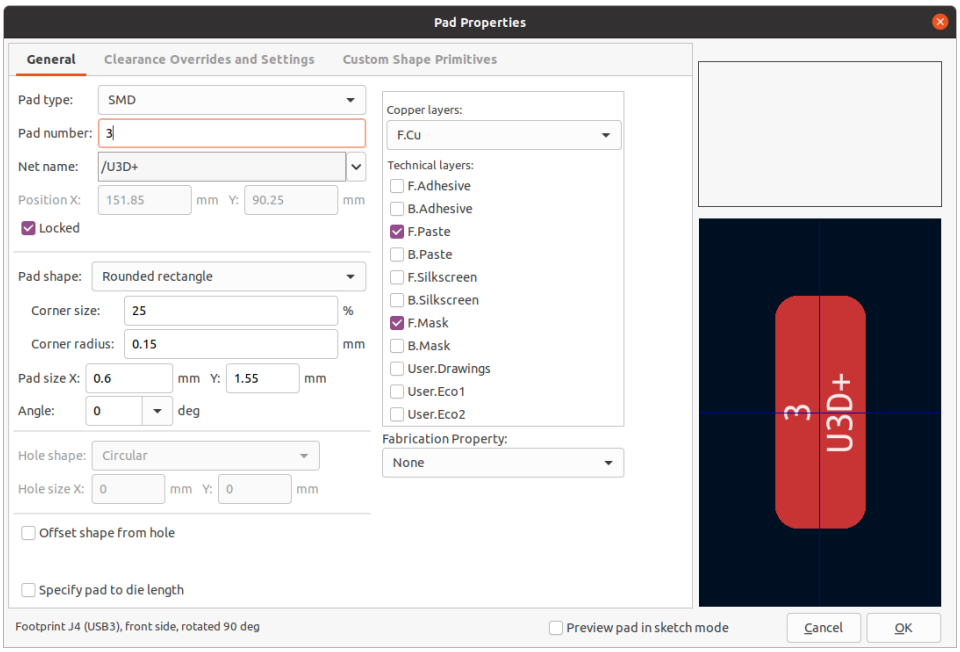
Note
TODO: 编写这一节—包括封装属性，从&#

4.5 使用焊盘

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*.

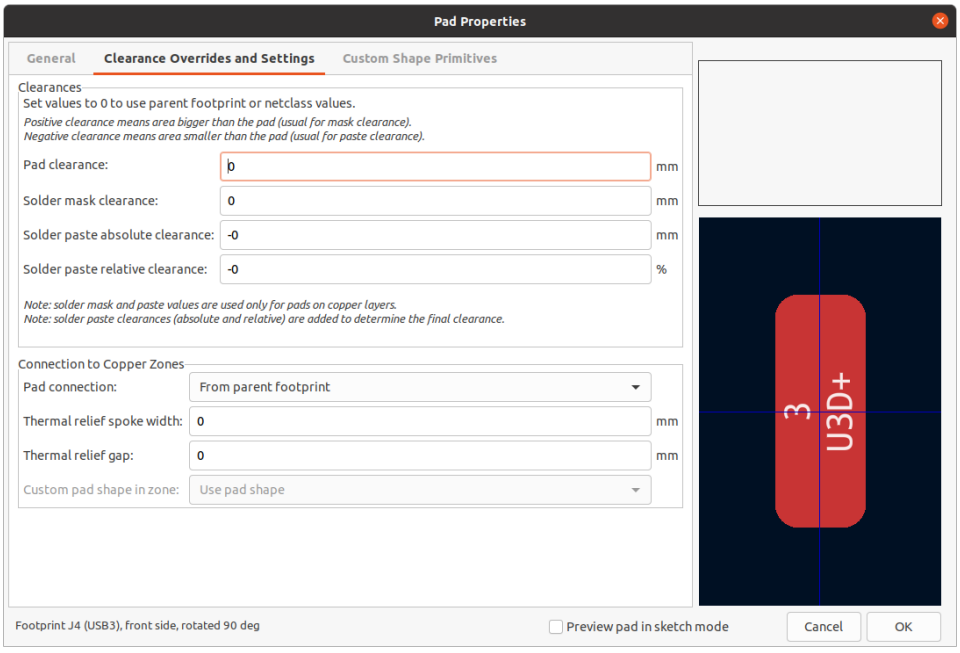
For through-hole pads, it is possible to remove the pad shape from copper layers where the pad is not electrically connected to other copper (tracks or filled zones). Setting the copper layers to **connected layers only** will remove the pad shape from any unconnected layers, and setting to **F.Cu, B.Cu, 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.



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 使用区域

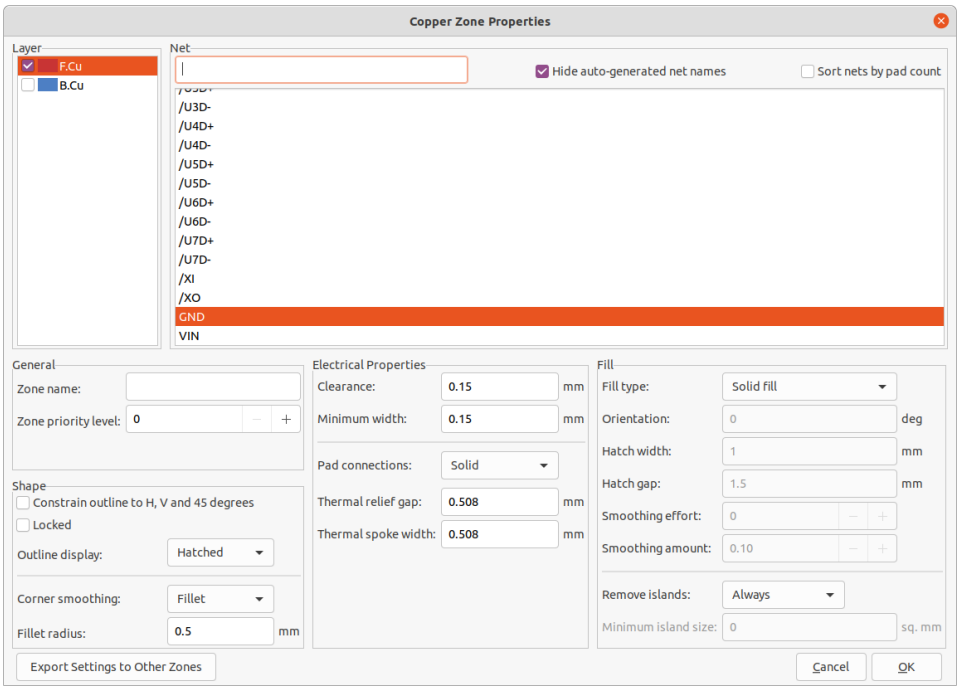
Copper zones, also sometimes called copper pours or fills by other EDA tools, are solid or hatched areas of copper assigned to a particular net that automatically keep clearance from other copper objects. Zones are commonly used to fill in all free space on a board layer (or a portion of a layer) in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



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.

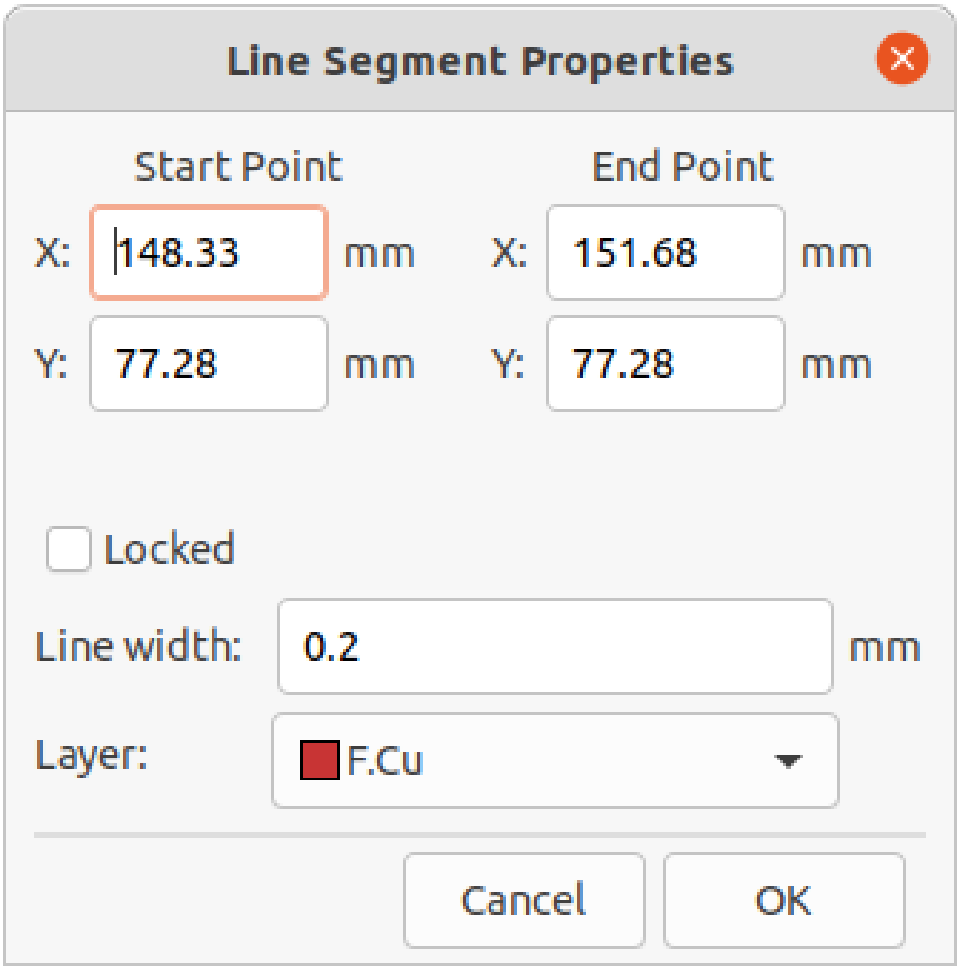
4.7 图形对象

图形对象 (直线、圆弧、矩形、&#x5
"0" 以禁用轮廓。

4.7.1 正在创建图形形状

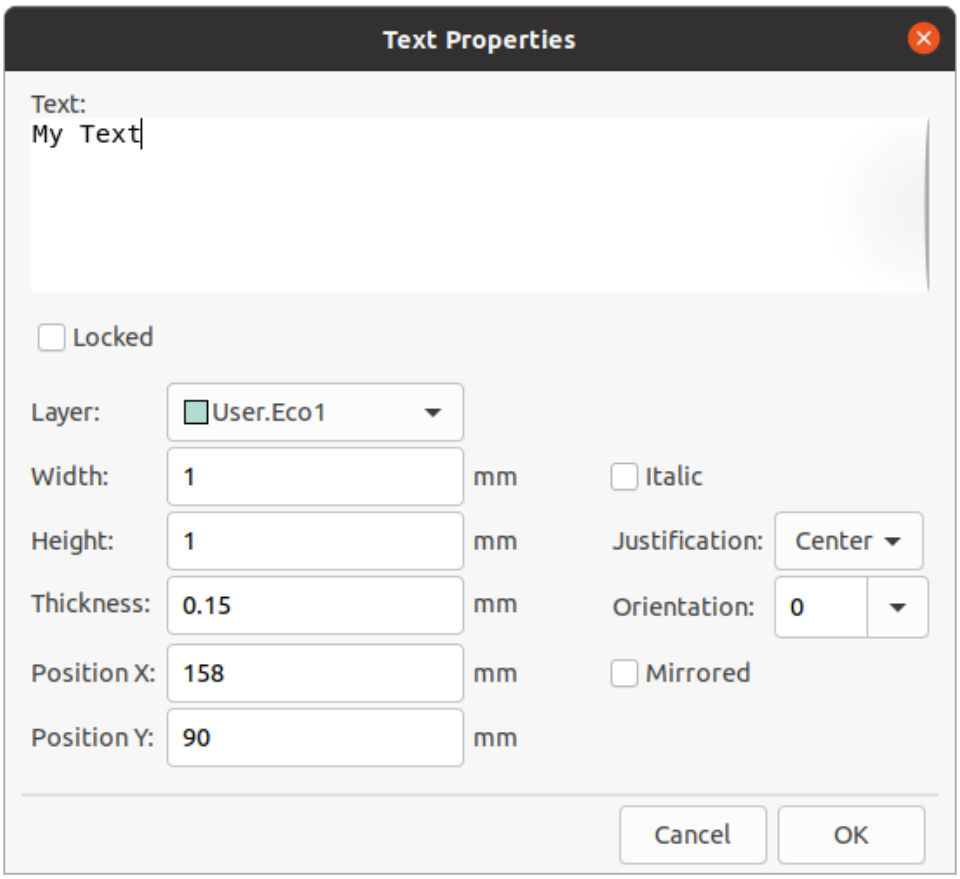
The right toolbar can be used to create lines (, default hotkey kbd:[Ctrl+Shift+L]), arcs (, default hotkey kbd:[Ctrl+Shift+A]), rectangles ( ), circles (, default hotkey kbd:[Ctrl+Shift+C]), and polygons (, default hotkey kbd:[Ctrl+Shift+P]).

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.



4.7.2 正在创建文本对象

Graphical text may be placed by using the (T) 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 电路板轮廓 (边缘切割)

KiCad 使用 `Edge.Cuts` 层上的图形对象来定&#x4
该轮廓必须是一个连续（封闭&#xff0
如果没有定义电路板的轮廓，&#x62
3D 查看器和一些设计规则的检查&

4.8 标注

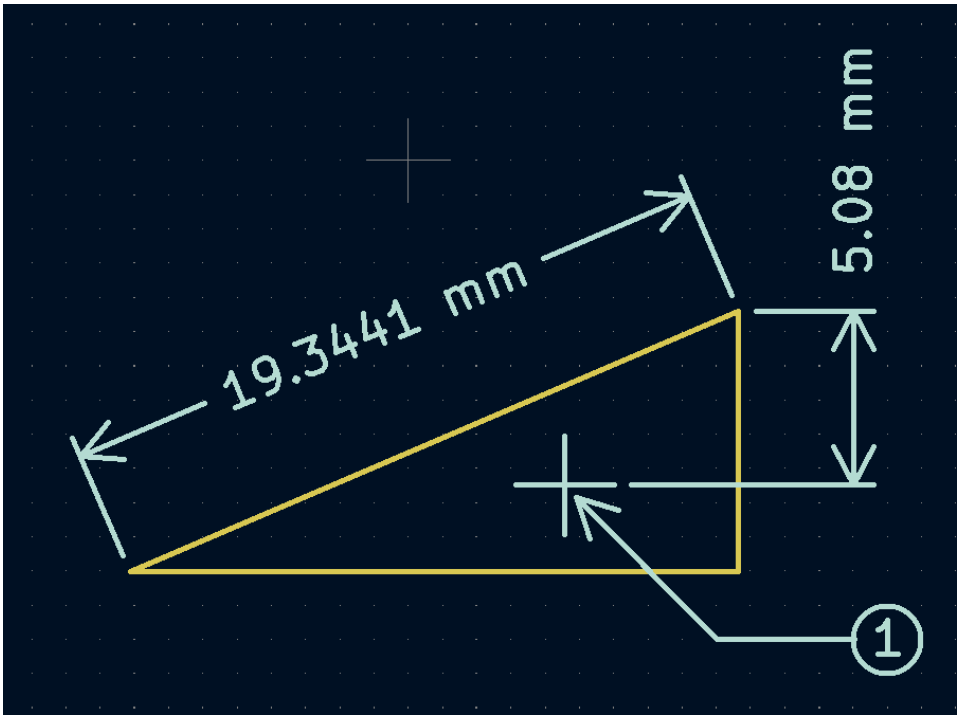
标注是用于显示测量值或电路&#x6
它们可以被添加到任何绘图层&#x4e
KiCad 目前支持四种不同类型的标&#x6ce

Aligned dimensions ( ) show a measurement of distance between two points. The measurement axis is the line that connects those two points, and the dimension graphics are kept parallel to that axis.

Orthogonal dimensions ( ) also measure the distance between two points, but the measurement axis is either the X or Y axis. In other words, these dimensions show the horizontal or vertical component of the distance between two points. When creating orthogonal dimensions, you can select which axis to use as the measurement axis based on where you place the dimension after selecting the two points to measure.

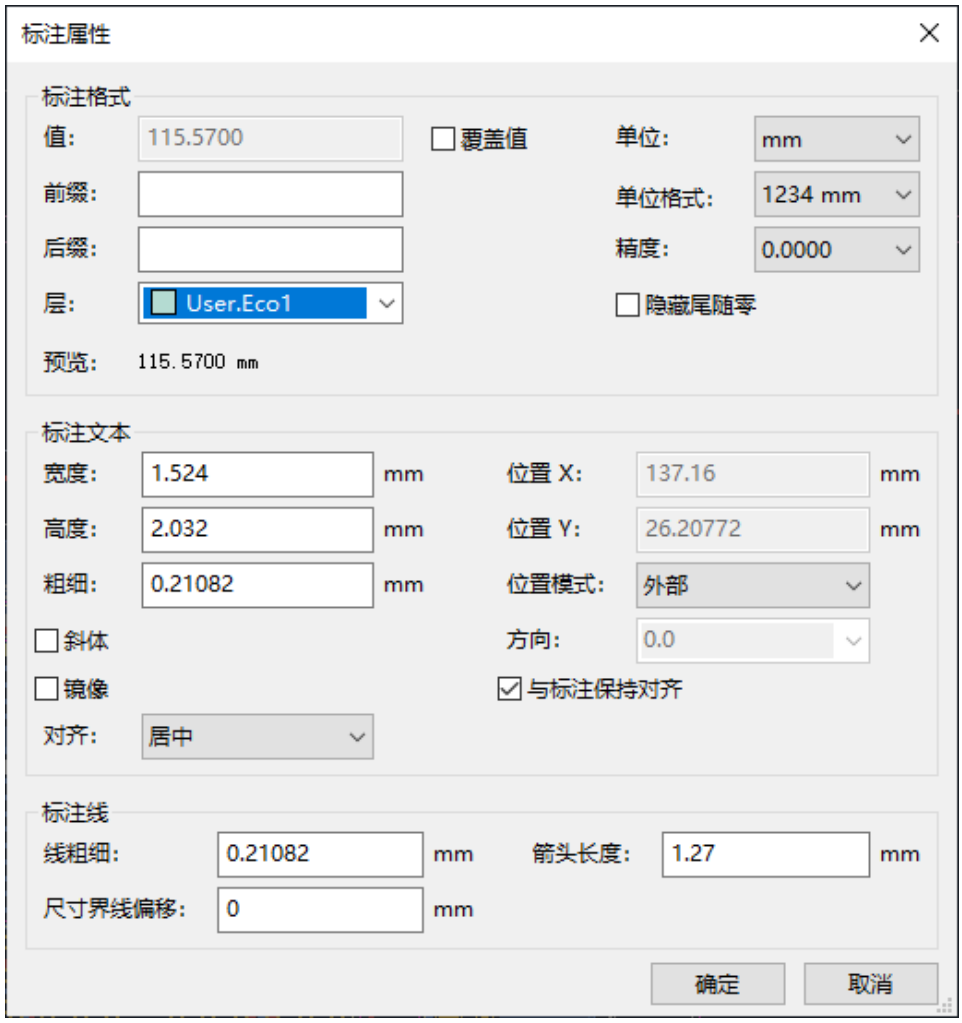
Center dimensions ( ) create a cross mark to indicate a point or the center of a circle or arc.

Leader dimensions ( ) create an arrow with a leader line connected to a text field. This text field can contain any text, and an optional circular or rectangular frame around the text. This type of dimension is often used to call attention to parts of the design for reference in fabrication notes.



创建一个标注后，可以编辑其&#x5c5
kbd:[E]）以改变显示数字的格式以&#x53ca

Note
您可以在电路板设置对话框的文&#x



4.8.1 标注格式选项

Override value: When enabled, you may enter a measurement value directly into the **Value** field that will be used instead of the actual measured value.

Prefix: Any text entered here will be shown before the measurement value.

Suffix: Any text entered here will be shown after the measurement value.

Layer: Selects which layer the dimension object exists on.

Units: Selects which units to display the measured value in. **Automatic** units will result in the dimension units changing when the display units of the board editor are changed.

Units format: Select from several built-in styles of unit display.

Precision: Select how many digits of precision to display.

4.8.2 标注文本选项

大多数标注文本选项与其他图&#x
也有一些特殊的选项适用于标&#x

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

Keep aligned with dimension: When enabled, the orientation of the dimension text will be adjusted automatically to keep the text parallel with the measurement axis.

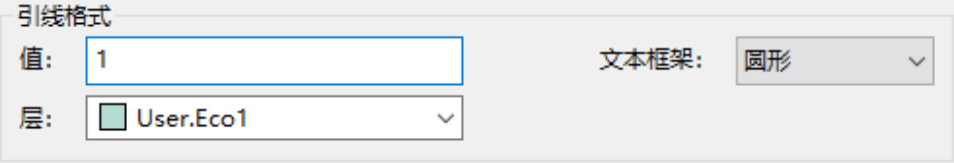
4.8.3 标注线选项

Line thickness: Sets the thickness of the graphical lines that make up a dimension's shape.

Extension line offset: Sets the distance from the measurement point to the start of the extension lines.

Arrow length: Sets the length of the arrow segments of the dimension's shape.

4.8.4 引线选项



Value: Enter the text to show at the end of the leader line.

Text frame: Select the desired border around the text (circle, rectangle, or none).

4.9 布线

KiCad 具有交互式布线器的功能：

- 允许对单轨和差分对进行手动&#x
  (半自动) 布线。
- 可通过以下方式修改现有设计&#x
  - 拖动现有布线时重新布线现有&
  - 拖动封装时重新布线连接到封&
- 允许通过插入蛇形线 + 来调整布&
  为具有严格时序要求的设计调&#

默认情况下，布线器在放置布&#x7e
如果需要的话，可以通过使用&#x9a
DRC 违规选项来禁用这种行为（见

布线器有三种模式，可以随时&#x900
布线器模式用于布线新布线，&#x4f4
kbd:[D]）命令拖动现有布线时。 这&#x4e9

- **Highlight Collisions**: in this mode, most of the router features are disabled and routing is fully manual. When routing, *collisions* (clearance violations) will be highlighted in green and the newly-routed tracks cannot be fixed in place if there is a collision unless the Allow DRC Violations option is turned on. In this mode, up to two track segments may be placed at a time (for example, one horizontal and one diagonal segment).
- **Shove**: in this mode, the track being routed will walk around obstacles that cannot be moved (for example, pads and locked tracks/vias) and *shove* obstacles that can be moved out of the way. The router prevents DRC violations in this mode: if there is no way to route to the cursor position that does not violate DRC, no new tracks will be created.
- **Walk Around**: in this mode, the router behaves the same as in Shove mode, except no obstacles will be moved out of the way.

使用哪种模式是一个偏好问题&#x30
对于大多数用户，我们建议使&#x752
45 度 (H/V/45) 布线段。如果需要使用 H/V/45
以外的角度布线段，则必须使&#x75

有五个主要的布线功能。单轨&#x5
(相位)。 所有这些都存在于顶部&#
上面介绍了重载图标的使用。
一个用于两个布线功能，一个&#x752
此外，布线菜单允许选择设置&#x5

要布线，请点击布线 图标（从&
布线 下）或使用热键 kbd:[X]。 点击&#x4e
被布线的网络会自动高亮显示&#xff0
可以通过改变 "偏好设置" 对话框&#
"间隙轮廓" 设置来禁用间隙轮廓&#x

Note
间隙轮廓显示从布线网络到 PCB
上任何其他铜线的最大间隙。
可以使用自定义设计规则为不同&#
布线器将考虑这些间隙，但仅直&#x8

当布线器处于活动状态时，将&#x4e
(*unfixed* temporary) 对象，它们显示当您使用&#x5d
kbd:[Enter] 键来固定布线 (*fix* the route) 时将创建哪&
kbd:[Esc] 键或通过选择另一个工具退&#x5
kbd:[End]) 将固定所有布线并退出布线&#x5

在布线时，可以使用撤消最后&#x4e0
(热键 kbd:[Backspace]) 取消固定最近固定的&#x5e

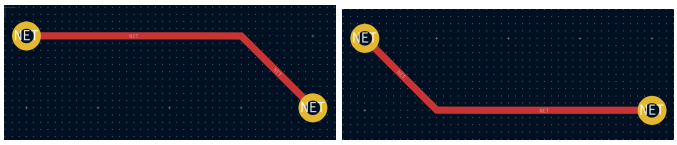
在以前的 KiCad 版本中，使用鼠标&#x5de6
kbd:[Enter] 来固定已布线的线段会固定&#x6
在 KiCad 6 中，这种行为现在是可选&#x768
包括 在鼠标光标位置结束的线&#x6
通过在交互式布线器设置对话&#x684
"点击后固定所有线段" 选项，可&#x

布线时，可以按住 kbd:[Ctrl] 键禁用网&#x68
kbd:[Shift] 键禁用对焊盘和过孔等对象&#x

Note
也可以通过更改首选项话框的&#x
我们建议您在一般情况下保持启&#

4.9.1 布线形态

在垂直(H) / 水平 (V) / 45 度模式下布线时&#
是指一组两个线段如何连接单&#x4e
H/V/45 度线段无法到达的两个点。
在这种情况下，这些点将由一&#x67
(45度) 线段连接。形态指的是这些&



KiCad 的布线器试图根据一系列因&#x7d20
"坏" 拐角 (如锐角)。当从焊盘布线&
将选择将路线与焊盘最长边缘&#x5b

在某些情况下，KiCad 无法正确猜&#x6d4b
(热键 kbd:[/])。

在没有明显的“最佳”形态的情况&#
将使用鼠标光标的移动来选择&#x5
(水平或垂直) 线段开始，请在水&#x

Note
如果使用切换布线形态命令覆盖
KiCad 选择的形态，则在当前布线操&#x4f

4.9.2 布线转角模式

当以 H/V/45 模式布线时，KiCad 的布线器&#
(热键 kbd:[Ctrl+/])。使用圆角布线时，每&#

选择所需布线后，还可以在布&#x7e

Note
尚不支持使用圆弧拖动布线。当&#x

4.9.3 布线宽度

被布线的线段的宽度是通过以&#x4e

按钮被启用，宽度将被设置为
否则，如果顶部工具栏中的布&#x7
"使用网络类宽度"，则宽度将取&#x
最后，如果布线宽度下拉菜单&#x88

Note
布线宽度永远不能低于在电路板&#x

KiCad 的布线器支持活动路线的单&#x4e00
换句话说，要在线段中间改变&#x5b
要改变活动线段的宽度，可使&#x75
kbd:[W] 和 kbd:[Shift+W]，在电路板设置对话框&#

4.9.4 放置过孔

在布线线段时，切换层会在当&#x524
一旦你放置了过孔，布线将继&#x7ee
有几种方法可以选择一个新层&#x5

- 使用热键选择特定的图层，如 kbd:[PgUp] 选择 `F.Cu` 或 kbd:[PgDn] 选择 `B.Cu`。
- 通过使用 "下一层" 或 "上一层" 热&#x952 (kbd:[+] 和 kbd:[-])。
- 通过使用 "放置过孔" 热键 (kbd:[V])，它&#
- 通过使用 "选择图层并通过通孔&#
  操作(热键 kbd:[<])，将打开一个对话&#x6

过孔的尺寸将从激活的过孔尺&#x5b
(kbd:[']) 和减小过孔尺寸 (kbd:[\]) 热键访问&#x300
时，将使用 "电路板设置" 的 "网络&#
部分中配置的过孔大小 (除非被&#x

如果在电路板设置对话框的约&#x6
使用热键 kbd:[Ctrl+V] 来放置微孔，kbd:[Alt+Shift+V]
来放置盲/埋孔。 微孔只能被放&#x7
盲孔/埋孔可以放置在任何一层&#x4

布线器放置的过孔被认为是已&#x5e
这意味着过孔网络可以自动更&#x65
PCB 时改变了线段的网络名。 在&#x67d0
对于特定的过孔，可以通过关&#x95
"自动更新过孔网络" 复选框来禁&#
使用 "添加独立过孔" 工具放置的&#

4.9.5 差分对布线

KiCad 中的差分对被定义为具有共&#x540
基数名称 和正负后缀的网路。
KiCad 支持使用 + 和 −，或者 `P` 和 `N` 作为后
例如，`USB+` 和 `USB−` 构成一对差分，`USB_P`
和 `USB_N` 也是如此。 在第一个例子&#x4e
`USB`，第二个例子中是 `USB_`。 后缀样&#x5
`USB+` 和 `USB_N` 不构成差分对。 请确保你
PCB 编辑器中使用差分对布线器。

要对差分对进行布线，请点击&#x5e0



图标（从绘图工具栏或从顶部&
布线 下）或使用热键 kbd:[6]。 点击&#x4e
你可以从差分对的正网或负网&#x5f0

Note
目前不可能在现有差分对布线的&#

差分对布线器将尝试用设计规&#x5
"扇出" 部分，以最大限度地缩短&#

当交换层或使用放置过孔 (kbd:[V]) 操&#x4

4.9.6 修改布线

线段被布线后，可以通过移动&#x621
当选择一个线段时，热键 kbd:[U] 可&#x4ee5
第一次按下 kbd:[U] 将选择与焊盘或&#x8
第二次按 kbd:[U] 将再次扩大选择范&#x5
用这种技术选择线段可以用来&#x5

有两种不同的拖动命令可用于&#x4
(45 度模式) 命令热键 kbd:[D] 用于通过布&#
热键 kbd:[G] 用于将布线段一分为二&#xff0

Note

目前还不能拖动包含圆弧的布线&#
(kbd:[D]) 来调整其大小。使用此命令调&#
DRC 检查。

移动命令（热键 kbd:[M]）也可以在&#x7ebf
该命令将拾取选定的线段，而&#x5f
使用移动命令移动线段时，不&#x4f1
DRC 检查。

在移动封装的同时，可以对附&#x77
要做到这一点，在选择了一个&#x5
任何以封装的一个焊盘为终点&#x7
这个功能有一些限制：它只在&#x9a
此外，只有以封装的焊盘为终&#x7
仅仅穿过焊盘或在原点以外的&#x7

可以使用编辑布线和过孔对话&#x684

4.9.7 长度调整

长度调谐工具可用于在布线后&#x4e
要调整线段的长度，首先要挑&#x9

单线调谐工具（图标 或热键 kbd:[7]&

差分对调谐工具（图标  或热&#x952e
kbd:[8]）将为差分对做同样的事情&#x300

差分对偏斜调整工具（图标  &#x6216
kbd:[9]）将为差分对中较短的成员&#x589e
与 "布线" 图标一样，"调谐" 图标可&
"布线" 菜单下拉框和右侧的绘图&#x

要选择长度调整工具的目标长&#x5
kbd:[Ctrl+L] 打开 "长度调整设置" 对话框：



此对话框还可用于配置曲折形&#x72

配置好目标长度后，在你希望&#x5f
沿着线段移动鼠标光标，蛇形&#x5f
光标旁边会出现一个状态窗口&#xff
再次点击，完成放置当前的蛇&#x5f
如果需要，可以在同一条布线&#x4e

Note
长度调整工具仅支持调整两个焊&

4.9.8 交互式布线设置

交互式布线器设置可通过路径&#x83d



设置	**说明**
模式	设置用于创建新布线和 有关详细信息，请参阅&#
自由角度模式以任何角度布线，& 45 度增量布线。 仅当布线模式设置为突& 此选项才可用。
绕过障&#x (如焊盘) 后面移动碰撞布线。
移除多& 仅保留回路中最近布线& 和其他不需要的布线&#x3002
平滑拖& 以最大限度地减少方&#x5411

设置	说明
允许违规则 DRC 规则	在高亮碰撞模式下，允& DRC 规则的布线和过孔。 在其他模式下不起作用
优化正在处理内容。 KiCad	在处理内容。布线段将& KiCad 优化屏幕上可见的其余& 优化过程去除了不必要 禁用时，不会对正在拖& 在拖动布线时尝试优化
使用鼠标从布局器件库中选择单个器件。	如果鼠标从开始位置开& 如果鼠标主要水平或&#x5782 当鼠标离开布线起始位& 并且可以通过移回起始&
点击时将布局器件库中的器件。	线时单击& 包括在鼠标光标结束&#x7684 禁用，最后一个线段 (在鼠标光标处结束的&#x7eb 将不会固定在适当位置& 可以通过进一步的鼠标&

4.10 向前和向后批注

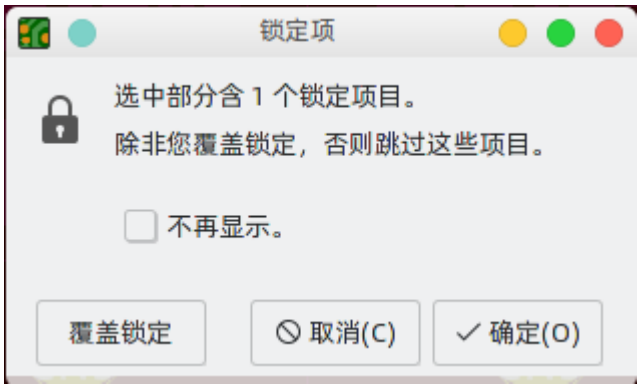
Note
TODO：写下这一节

4.10.1 位置重新批注

Note
TODO：写下这一节

4.11 锁定

大多数对象可以通过其属性对&#x8
"切换锁定" 热键（kbd:[L]）来锁定。
被锁定的对象不能被选择，除&#x9
"被锁定的项目" 复选框被启用。
试图移动锁定的项目将导致一&#x4e



在这个对话框中选择 "覆盖锁定" 将允许移动锁定的项目。 选择 "确定" 将允许你在选择中移动任&#x 选择 "不再显示" 将使你在剩下的&#

4.12 批量编辑工具

Note
TODO：写下这一节

4.13 清理工具

Note
TODO：写下这一节

4.14 正在导入图形

Note
TODO：写下这一节

4.14.1 从 DXF 和 SVG 文件导入矢量图

Note
TODO：写下这一节

4.14.2 正在导入位图图像

Note
TODO：写下这一节

Chapter 5

检查电路板

5.1 测量工具

测量工具允许你在 PCB 上的各点&#x4e4b
要激活该工具，请点击右侧工&#x51

图标，或使用热键 kbd:[Ctrl + Shift + M]。 一&#x65

Note

测量工具用于不需要永久显示的&#
您所做的任何测量都将仅在该工&#
要创建将显示在打印输出和打印&#

5.2 设计规则检查

设计规则检查器用于验证 PCB 是&#x5426
KiCad 可以在布线线段时自动防止&#x4e00
这意味着在为 PCB 创建制造文件之

要使用设计规则检查器，请点&#x51

图标，或者从检查菜单中选&#x62e9



DRC 控制窗口的顶部部分包含一&#x4e9b

Refill all zones before performing DRC: when enabled, zones will be refilled every time the design rule checker is run. Disabling this option may result in incorrect DRC results if zones have not been refilled manually.

Report all errors for each track: when enabled, all clearance errors will be reported for each track segment. When disabled, only the first error will be reported. Enabling this option will result in the design rule checker running more slowly.

Test for parity between PCB and schematic: when enabled, the design rule checker will test for differences between the schematic and PCB in addition to testing the PCB design rules. This option has no effect when running the PCB editor in standalone mode.

运行 DRC 后，任何违规行为都会显&
DRC 控制窗口的中间部分。 违反&#x89c4
PCB 之间的差异会显示在三个不同&
违规列表下面的控件可以用来&#x6
在运行 DRC 后，可以使用保存按钮&



每项违规行为涉及 PCB 上的一个或
在违章列表中，涉及的对象列&#x5
点击列表中的违规行为将移动
PCB 编辑器的视图，使受影响的区&
点击违章所涉及的对象之一将&#x7

右键单击列表视图中的违规行&#x4
除非 DRC 控制窗口底部的 "排除" 复&#x
被排除的违规行为在设计规则&#x

5.2.1 间隙和约束解析

间隙和约束解决工具允许你检&#x
当设计具有复杂设计规则的 PCB &#x65f6

要检查两个对象之间适用的间&#x9
间隙报告对话框将显示每个铜&#x5

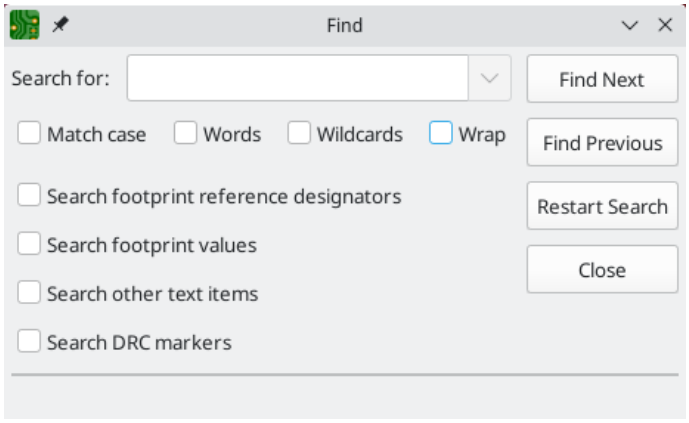


To inspect the design constraints that apply to an object, select it and choose Constraints Resolution from the Inspect menu. The Constraints Report dialog will show any constraints that apply to the object.



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 ( ) 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.

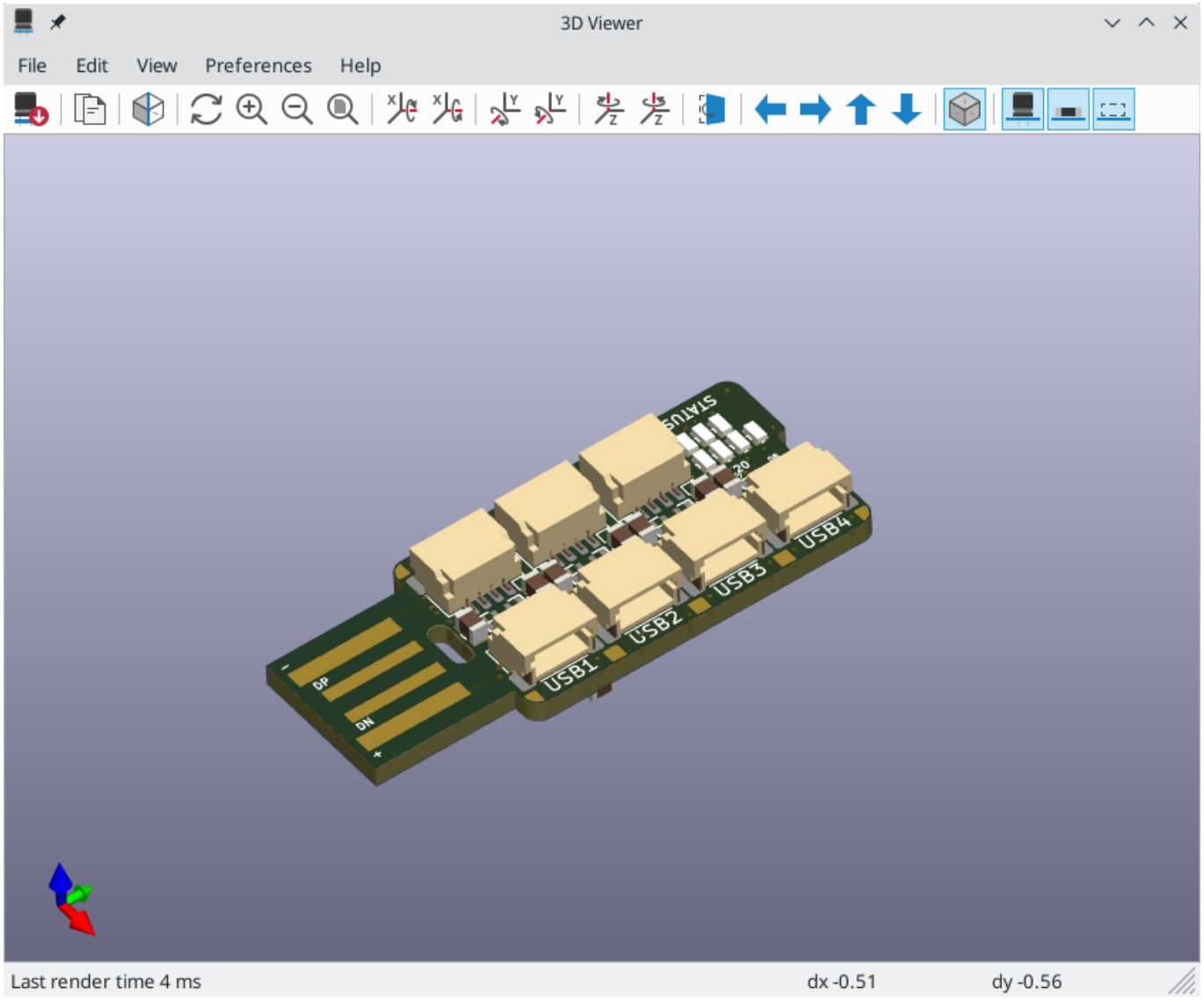
Search footprint values: Selects whether the search should apply to footprint value fields.

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 查看器

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.

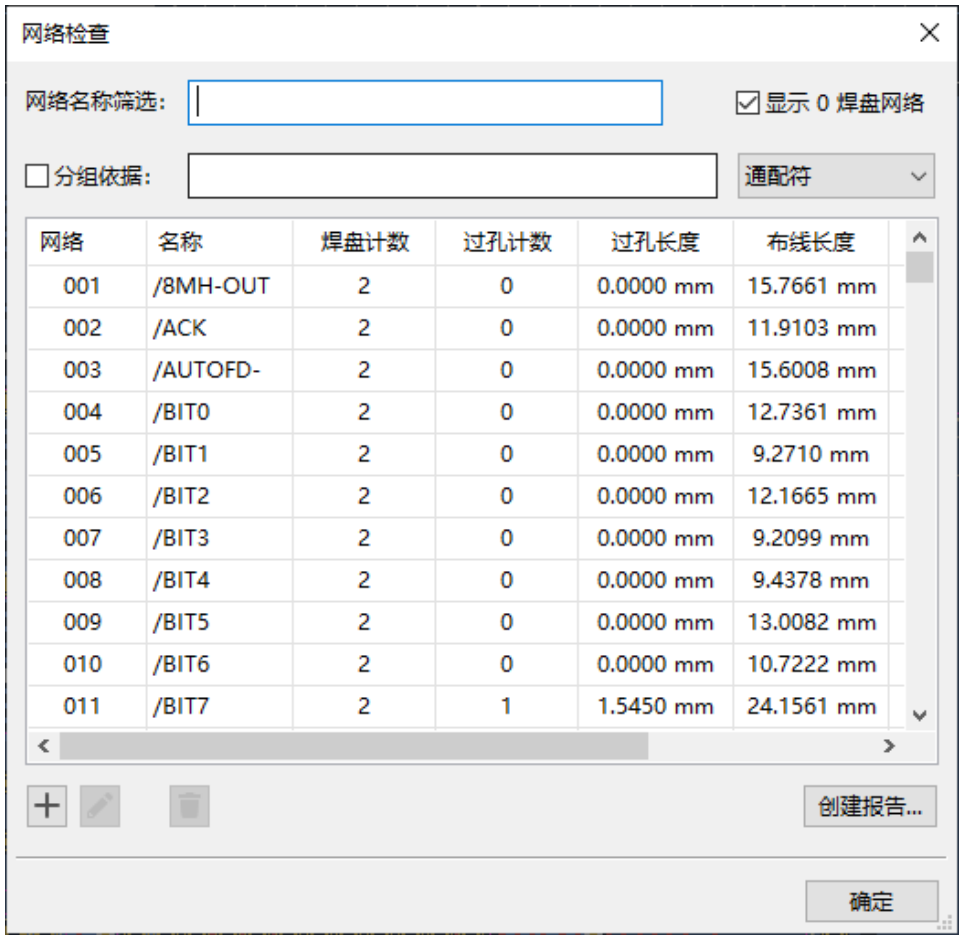
	Reload the 3D model
	Copy 3D image to clipboard
	Render current view using raytracing
	Zoom in
	Zoom out

	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 *other*

5.5 网络检查

网络检查器允许你查看电路板&#x4
要打开检查器，请点击外观面&#x6


图标，或者从检查菜单中选择



点击网络列表中的一个网络会&#x57
单击列标题允许您按该列对网&#x7

分组依据字段允许您将不同的&#x
例如，如果您有两个名为 `DATA0` 和
`DATA0_EXT` 的网络，则使用值为 `DATA0*` 的分&#
通过将分组依据模式从通配符&#x66
RegEx (正则表达式)，可以创建更复&#x6742
分组依据模式的子字符串 (Substr) 变&#x4f

例如，如果您有 `U1D+`，`U1D-`，`U2D+` 和 `U2D-`，&#
`U*D` 将在通配符模式下匹配所有四
`U*D`。 在通配符子串模式下，它将&
和 `U2D`。

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

网络检查器中显示的长度不同于&
因为网络检查器显示连接到网络&#
而长度调整工具显示最近的两个&
有关长度调整工具的更多信息，&#x

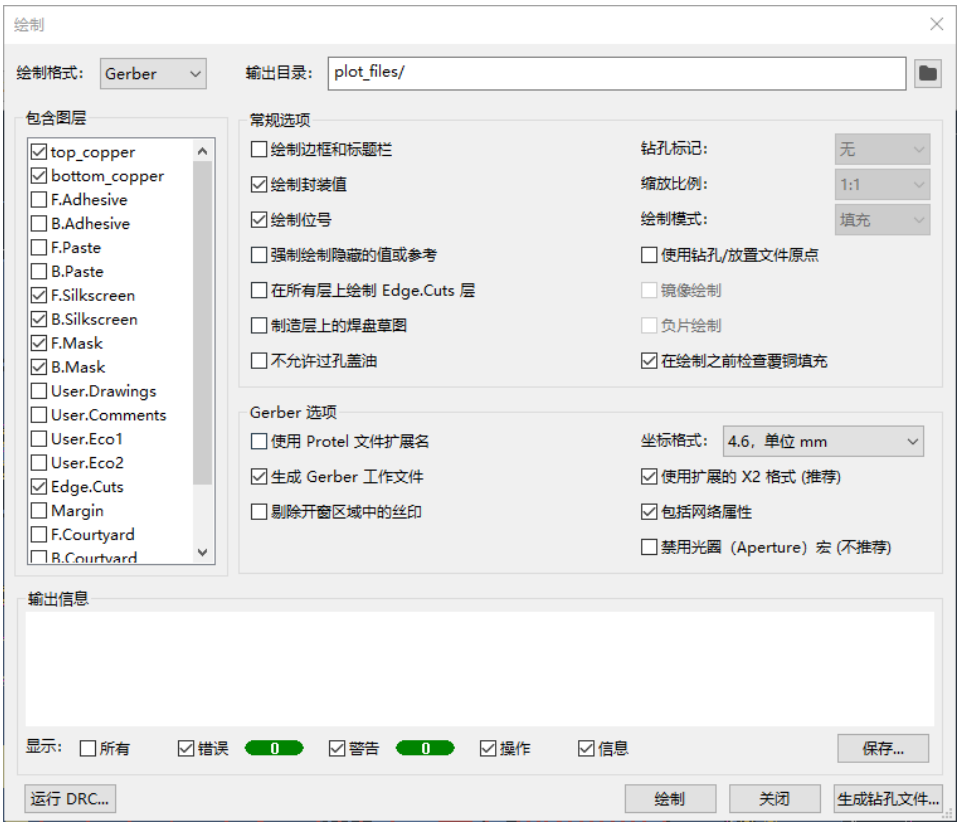
Chapter 6

生成输出

KiCad 可以生成和导出多种不同格式&
PCB 和与外部软件的接口很有用。
该功能可在文件菜单的几个不&#x54
制造输出部分包含准备制造 PCB &#x6240
输出部分包含生成可由外部软&#x4e
绘图功能允许你以各种格式导&#x51f
PCB 的 2D 线图。 打印功能允许你将
PCB 的视图发送到 2D 打印机上。

6.1 制造输出和绘制

KiCad 使用 Gerber 文件作为其 PCB 制造的主&#x8
要创建 Gerber 文件，请从文件菜单&#x4e2d
Gerbers。 绘图对话框将打开，允许&#x4f6
Gerber 文件。



6.1.1 绘制选项

Include Layers: Check that every layer used on your board is enabled in the list. Disabled layers will not be plotted.

Output directory: Specify the location to save plotted files. If this is a relative path, it is created relative to the project directory.

Plot border and title block: If enabled, the drawing sheet border and title block will be plotted on each layer. This should usually be disabled when plotting Gerber files.

Plot footprint values: If enabled, the Value field of each footprint will be plotted on whatever layer it exists on (unless the field visibility is disabled for a specific footprint).

Plot reference designators: If enabled, the Reference Designator field of each footprint will be plotted on whatever layer it exists on (unless the field visibility is disabled for a specific footprint).

Force plotting of invisible values / refs: If enabled, all footprint values and reference designators will be plotted, even if the field visibility is disabled for some of these fields.

Plot Edge.Cuts on all layers: If enabled, the Edge.Cuts (board outline) layer will be added to all other layers. Check with your manufacturer to see what the correct value of this setting is for their manufacturing process.

Sketch pads on fabrication layers: If enabled, footprint pads on fabrication (F.Fab, B.Fab) layers will be drawn as unfilled outlines rather than filled shapes.

Do not tent vias: If enabled, vias will be left uncovered on the solder mask layers (F.Mask, B.Mask). If disabled, vias will be covered by solder mask (tented).

Note

KiCad 不支持覆盖或开窗特定过孔。
覆盖只能是全局控制的(一块板上&

Drill marks: For plot formats other than Gerber, marks may be plotted at the location of all drilled holes. Drill marks may be created at the actual size (diameter) of the finished hole, or at a smaller size.

Scaling: For plot formats that support scaling other than 1:1, the plot scale may be set. The Auto scaling setting will scale the plot to fit the specified page size.

Plot mode: For some plot formats, filled shapes may be plotted as outlines only (sketch mode).

Use drill/place file origin: When enabled, the coordinate origin for plotted files will be the drill/place file origin set in the board editor. When disabled, the coordinate origin will be the absolute origin (top left corner of the worksheet).

Mirrored plot: For some plot formats, the output may be mirrored horizontally when this option is set.

Negative plot: For some plot formats, the output may be set to negative mode. In this mode, shapes will be drawn for the empty space inside the board outline, and empty space will be left where objects are present in the PCB.

Check zone fills before plotting: When enabled, zone fills will be checked (and refilled if outdated) before generating outputs. Plot outputs may be incorrect if this option is disabled!

6.1.2 Gerber 选项

Use Protel filename extensions: When enabled, the plotted Gerber files will be named with file extensions based on Protel (`.GBL`, `.GTL`, etc). When disabled, the files will have the `.gbr` extension.

Generate Gerber job file: When enabled, a Gerber job file (`.gbrjob`) will be generated along with any Gerber files. The Gerber job file is an extension to the Gerber format that includes information about the PCB stackup, materials, and finish. More information about Gerber job files is available at the Ucamco website.

Coordinate format: Configure how coordinates will be stored in the plotted Gerber files. Check with your manufacturer for their recommended setting for this option.

Use extended X2 format: When enabled, the plotted Gerber files will use the X2 format, which includes information about the netlist and other extended attributes. This format may not be compatible with older CAM software used by some manufacturers.

Include netlist attributes: When enabled, the plotted Gerber files will include netlist information that can be used for checking the design in CAM software. When X2 format mode is disabled, this information is included as comments in the Gerber files.

Disable aperture macros: When enabled, all shapes will be plotted as primitives rather than by using aperture macros. This setting should only be used for compatibility with old or buggy CAM software when requested by your manufacturer.

6.1.3 Postscript 选项

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.

Force A4 output: When enabled, the generated PostScript file will be A4 size even if the KiCad board file is a different size.

6.1.4 SVG 选项

Units: Controls the units that will be used in the SVG file. Since the SVG format has no specified units system, you must export using the same units setting that you want to use for importing into other software.

Precision: Controls how many significant digits will be used to store coordinates.

6.1.5 DXF 选项

Plot graphic items using their contours: Graphic shapes in DXF files have no width. This option controls how graphic shapes with a width (thickness) in a KiCad board are plotted to a DXF file. When this option is enabled, the outer contour of the shape will be plotted. When this option is disabled, the centerline of the shape will be plotted (and the shape's thickness will not be visible in the resulting DXF file).

Use KiCad font to plot text: When enabled, text in the KiCad design will be plotted as graphic shapes using the KiCad font. When disabled, text will be plotted as DXF text objects, which will use a different font and will not appear in exactly the same position and size as shown in the KiCad board editor.

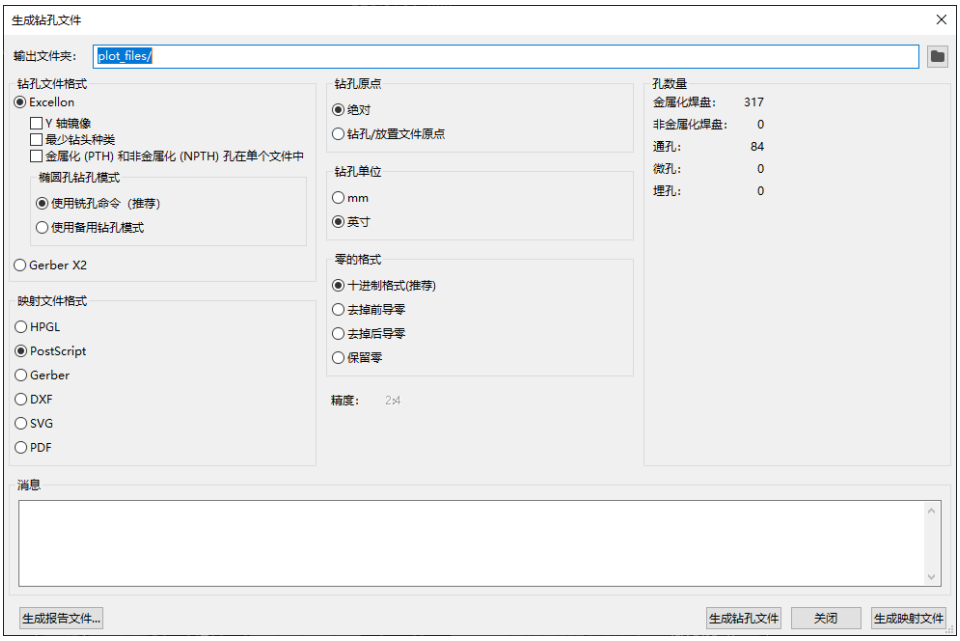
Export units: Controls the units that will be used in the DXF file. Since the DXF format has no specified units system, you must export using the same units setting that you want to use for importing into other software.

6.1.6 HPGL 选项

Default pen size: Controls the plotter pen size used to create graphics.

6.2 钻孔文件

KiCad 可以生成大多数 PCB 制造工艺所&#x
Excellon 或 Gerber X2 格式的数控钻孔文件。
KiCad 还可以生成钻孔图：一个显示
从 "制造输出" 菜单中选择 "生成钻&
选项来打开该对话框：



Output folder: Choose the folder to save generated drill and map files to. If a relative path is entered, it will be relative to the project directory.

Drill file format: Choose whether to generate Excellon drill files (required by most PCB manufacturers) or Gerber X2 files.

Mirror Y axis: For Excellon files, choose whether or not to mirror the Y-axis coordinate. This option should in general not be used when having PCBs manufactured by a third party, and is provided for convenience for users who are making PCBs themselves.

Minimal header: For Excellon files, choose whether to output a minimal header rather than a full file header. This option should not be enabled unless requested by your manufacturer.

PTH and NPTH in single file: By default, plated holes and non-plated holes will be generated in two different Excellon files. With this option enabled, both will be merged into a single file. This option should not be enabled unless requested by your manufacturer.

Oval holes drill mode: Controls how oval holes are represented in an Excellon drill file. The default setting, **Use route command**, is correct for most manufacturers. Only choose the **Use alternate drill mode** setting if requested by your manufacturer.

Map file format: Choose the output format for plotting a drill map.

Drill origin: Choose the coordinate origin for drill files. **Absolute** will use the page origin at the top left corner. **Drill/place file origin** will use the origin specified in the board design.

Drill units: Choose the units for drill coordinates and sizes.

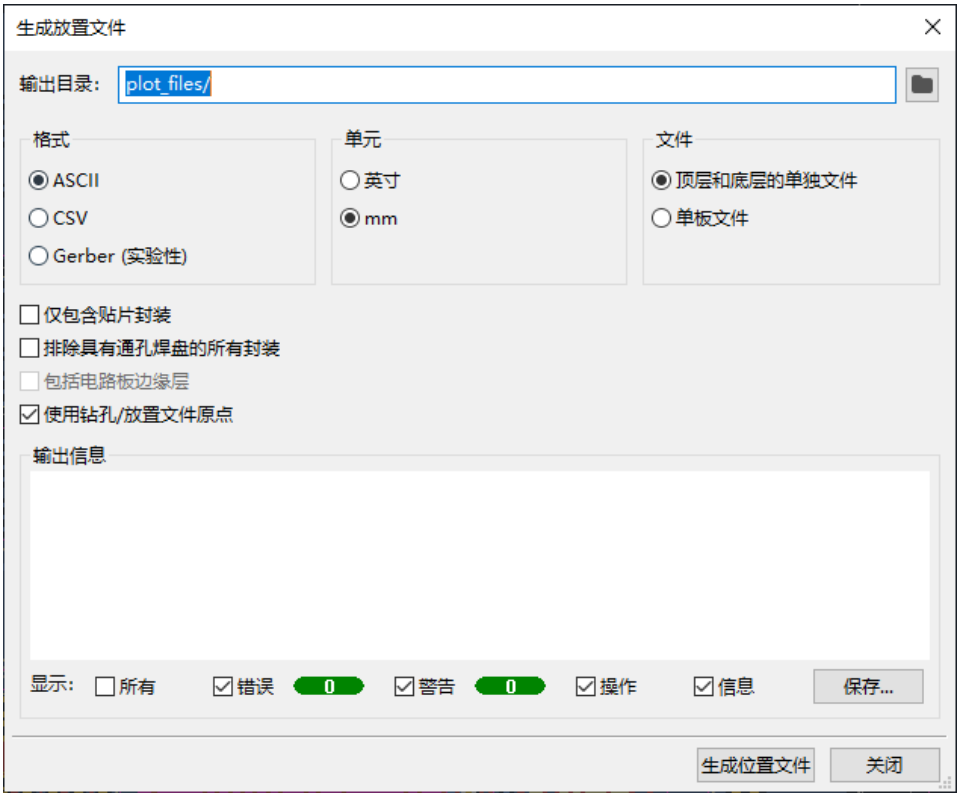
Zeros format: Controls how numbers are formatted in an Excellon drill file. Select an option here based on your manufacturer's recommendations.

6.3 元件放置文件

元件放置文件是一种文本文件&#xff0
这些文件通常用于对取放机进&#x88
PCB，你的制造商可能需要这些文&

Note

如果为生成的封装启用了“从放置&
这可用于排除不代表要组装的物&



Format: Choose between generating a plain text (ASCII), comma-separated text (CSV), or Gerber placement file format.

Units: Choose the units for component locations in the placement file.

Files: Choose whether to generate separate files for footprints on the front and back of the board or a single file combining both sides.

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.

Exclude all footprints with through hole pads: When enabled, footprints will be excluded from the placement file if they contain any through-hole pads, even if their fabrication type is set to SMD.

Include board edge layer: For Gerber placement files, controls whether or not the board outline is included with the footprint placement data.

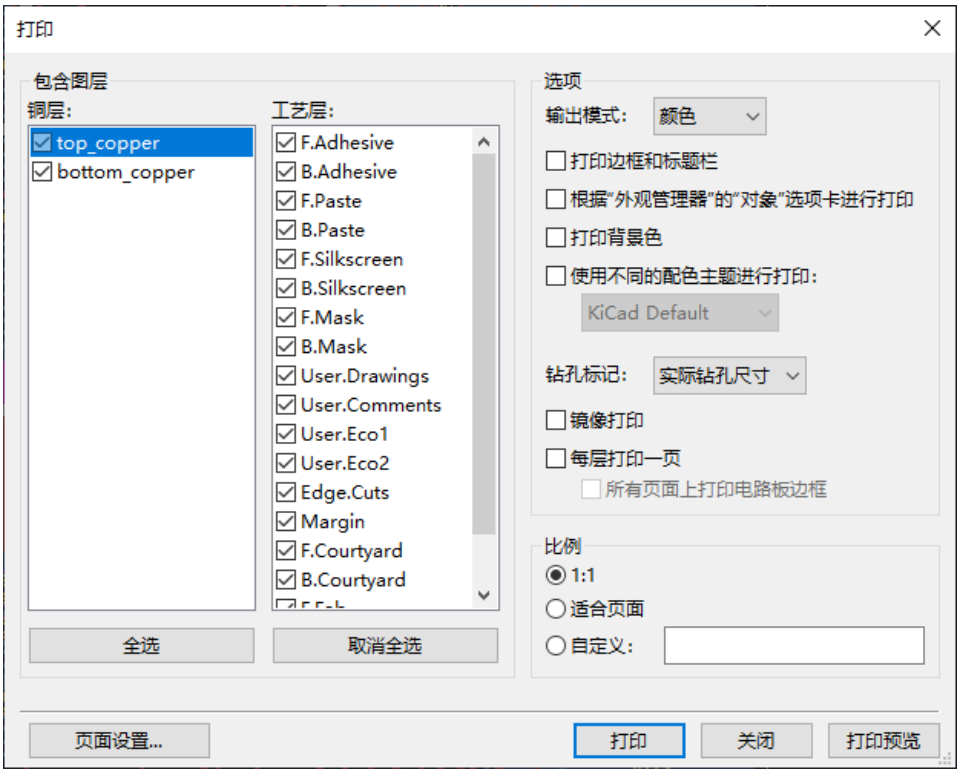
Use drill/place file origin: When enabled, component positions will be relative to the drill/place file origin set in the board design. When disabled, the positions will be relative to the page origin (upper left corner).

6.4 额外的制造产出

KiCad 还可以从电路板设计生成封装
D-356 网表文件和物料清单 (BOM)。 这些

6.5 打印

KiCad 可以使用文件菜单中的打印&#x64cd



Included layers: Select the layers to include in the printout. Unselected layers will be invisible.

Output mode: Choose whether to print in black and white or full color.

Print border and title block: When enabled, the page border and title block will be printed.

Print according to objects tab of appearance manager: When enabled, any objects that have been hidden in the Objects tab of the Appearance panel will be hidden in the printout. When disabled, these objects will be printed if the layer they appear on is selected in the Included Layers area.

Print background color: When printing in full color, this option controls whether or not the view background color will be printed.

Use a different color theme for printing: When printing in full color, this option allows a different color theme to be used for printing. When disabled, the color theme used by the board editor will be used for printing.

Drill marks: Controls whether to show drilled holes at their actual size, at a small size, or hide them from the printout.

Print mirrored: When enabled, the printout will be mirrored horizontally.

Print one page per layer: When enabled, each layer selected in the Included Layers area will be printed to an individual page. If this option is enabled, the **Print board edges on all pages** option controls whether to add the Edge.Cuts layer to each printed page.

Scale: controls the scale of the printout relative to the page size configured in Page Setup.

6.6 正在导出文件

KiCad 可以将电路板设计导出为各种
这些功能可以在文件菜单的导&#x51

Specctra .DSN: creates a file suitable for importing into certain third-party autorouter software. This exporter has no configurable options.

Note
TODO：文档 GenCAD 导出器

Note
TODO: 文档 VRML 导出器

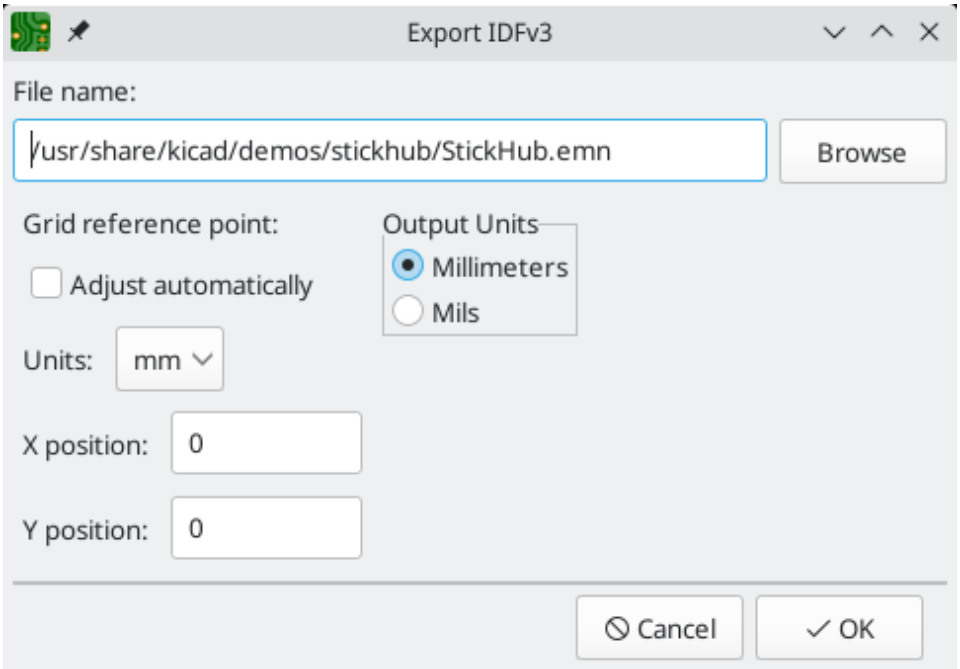
6.6.1 IDF Exporter

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: 文档 STEP 导出器

Note
TODO: 文档 SVG 导出器

Note
TODO: 文档 CMP 文件导出器

Hyperlynx: creates a file suitable for importing into Mentor Graphics (Siemens) HyperLynx simulation and analysis software.

Chapter 7

封装和封装&#x5

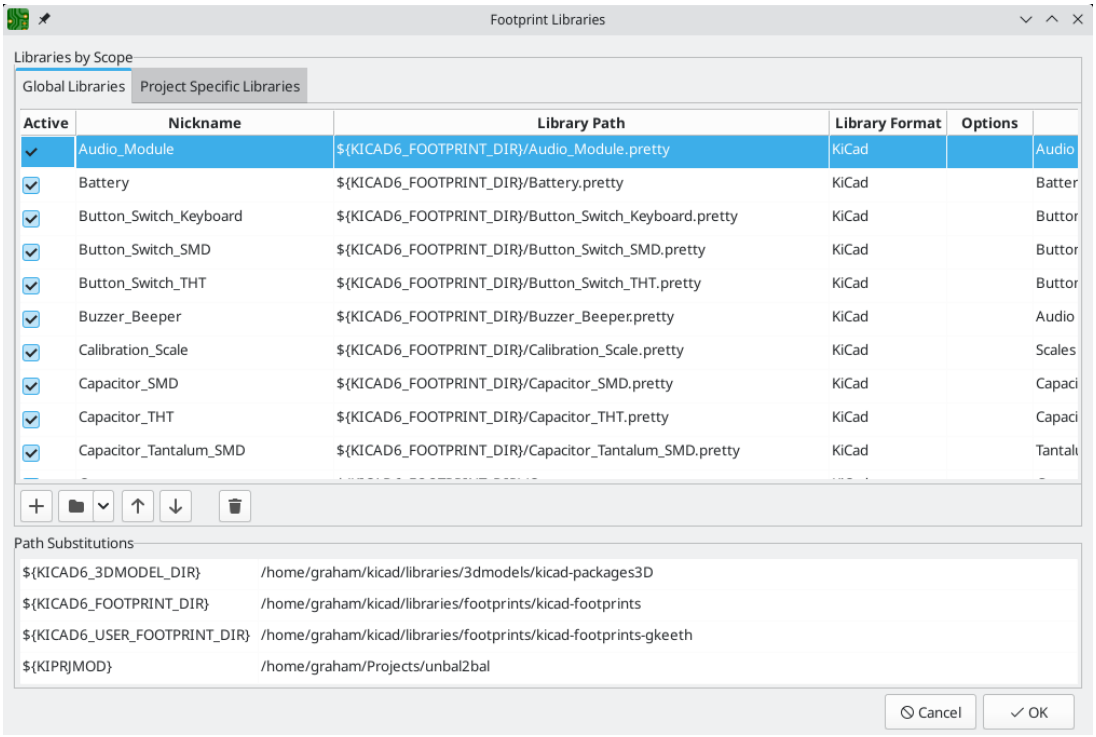
7.1 管理封装库

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 创建和编辑封装

Note
TODO：写下这一节

7.2.1 自定义焊盘形状

7.2.2 封装属性

Note
在这里提到网节点

7.2.3 封装向导

有关创建新的封装向导的更多&#x4

Chapter 8

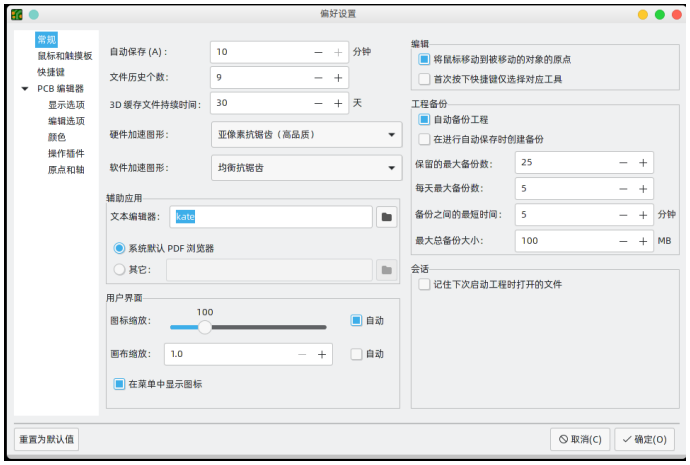
高级主题

8.1 配置和自定义

Pcbnew 有各种偏好设置，可以通过&#x504f
KiCad 的所有部分一样，Pcbnew 的偏好设
KiCad 次要版本之间相互独立，从而

偏好设置对话框的第一部分 (通&#x
在所有 KiCad 程序之间共享。KiCad 手册&
Pcbnew 的快捷键只有在 Pcbnew 运行时才&#x4f1a

8.1.1 显示选项



Rendering Engine: Controls if Accelerated graphics or Fallback graphics are used.

Grid style: Controls how the alignment grid is drawn.

Grid thickness: Controls how thick grid lines or dots are drawn.

Min grid spacing: Controls the minimum distance, in pixels, between two grid lines. Grid lines that violate this minimum spacing will not be drawn, regardless of the current grid setting.

Snap to grid: Controls when drawing and editing operations will be snapped to coordinates on the active grid. "Always" will enable snapping even when the grid is hidden; "When grid shown" will enable snapping only when the grid is visible.

Note
按住 kbd:[Ctrl] 可以暂时禁用网格捕捉。

Cursor shape: Controls whether the editing cursor is drawn as a small crosshair or a full-screen crosshair (a set of lines covering the entire drawing canvas). The editing cursor shows where the next drawing or editing action will occur and will be snapped to a grid location if snapping is enabled.

Always show crosshairs: Controls whether the editing cursor is shown all the time or only when an editing or drawing tool is active.

Net names: Controls whether or not net name labels are drawn on copper objects. These labels are guides for editing only and do not appear in fabrication outputs.

Show pad numbers: Controls whether or not pad number labels are drawn on footprint pads.

Show pad <no net> indicator: Controls whether or not pads with no net are indicated with a special marker.

Track clearance: Controls whether or not clearance outlines around tracks and vias are shown. Clearance outlines are shown as thin shapes around objects that indicate the minimum clearance to other objects, as defined by constraints and design rules.

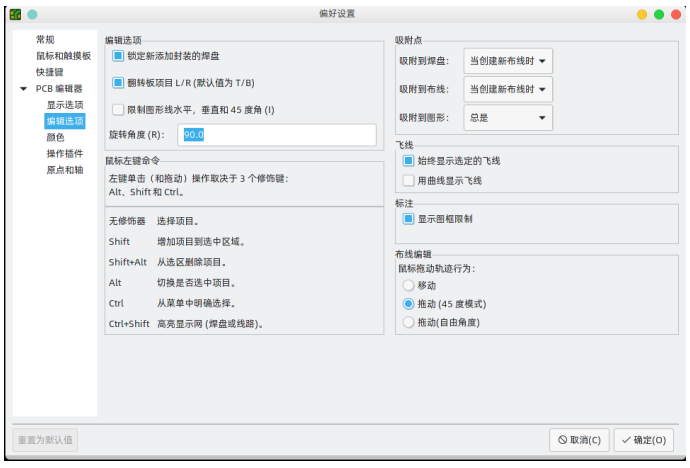
Show pad clearance: Controls whether or not clearance outlines around pads are shown.

Center view on cross-probed items: When both Eeschema and Pcbnew are running, controls whether clicking a component or pin in Eeschema will center the Pcbnew view on the corresponding footprint or pad.

Zoom to fit cross-probed items: Controls whether the view will be zoomed to show a cross-probed footprint or pad.

Highlight cross-probed nets: Controls whether or not nets highlighted in Eeschema will be highlighted in Pcbnew when the highlight tool is activated in both tools.

8.1.2 编辑选项



Flip board items L/R: Controls the direction board items will be flipped when moving them between the top and bottom layers. When checked, items are flipped Left-to-Right (about the Vertical axis); when unchecked, items are flipped Top-to-Bottom (about the Horizontal axis).

Step for rotate commands: Controls how far the selected object(s) will be rotated each time the Rotate command is used.

Allow free pads: Controls whether or not the pads of footprints can be unlocked and edited or moved separately from the footprint.

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.

Snap to pads: Controls when the editing cursor will snap to pad origins.

Snap to tracks: Controls when the editing cursor will snap to track segment endpoints.

Snap to graphics: Controls when the editing cursor will snap to graphic shape points.

Always show selected ratsnest: When enabled, the ratsnest for a selected footprint will always be shown even if the global ratsnest is hidden.

Show ratsnest with curved lines: Controls whether ratsnest lines are drawn straight or curved.

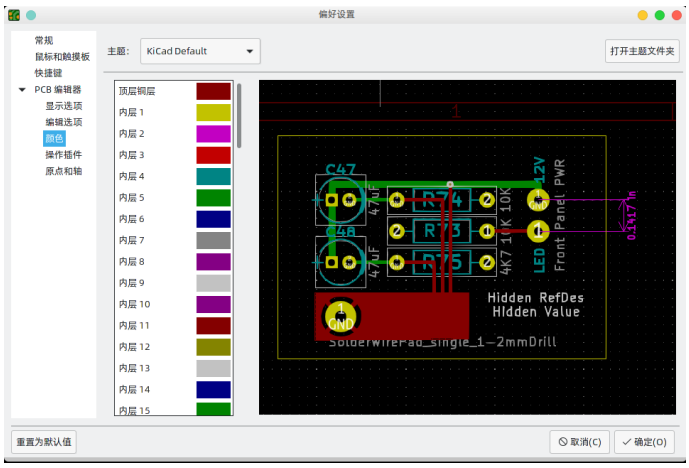
Mouse drag track behavior: Controls the action that will occur when you drag a track segment with the mouse: "Move" will move the track segment independent of any others. "Drag (45 degree mode)" will invoke the push-and-shove router to drag the track, respecting design rules and keeping other track segments attached. "Drag (free angle)" will move the nearest corner of the track segment, highlighting collisions with other objects but not moving them out of the way.

Limit actions to 45 degrees from start Controls whether lines drawn with the graphic drawing tools can take on any angle. Note that this only affects drawing new lines: lines can be edited to take on any angle.

Show page limits: Controls whether or not the page boundary is drawn as a rectangle.

Refill zones after Zone Properties dialog: Controls whether or not zones are automatically refilled after editing the properties of any zone. This may be disabled on complicated designs or slower computers to improve responsiveness.

8.1.3 颜色



Pcbnew 支持在不同的颜色主题之间&#x52
6.0 有两个内置的颜色主题："KiCad 默&#x
是一个新主题，设计用于大多&#x6
经典版" 是 KiCad 5.1 及更早版本的默认&#
Pcbnew 的外观，也可以安装其他用&#x623

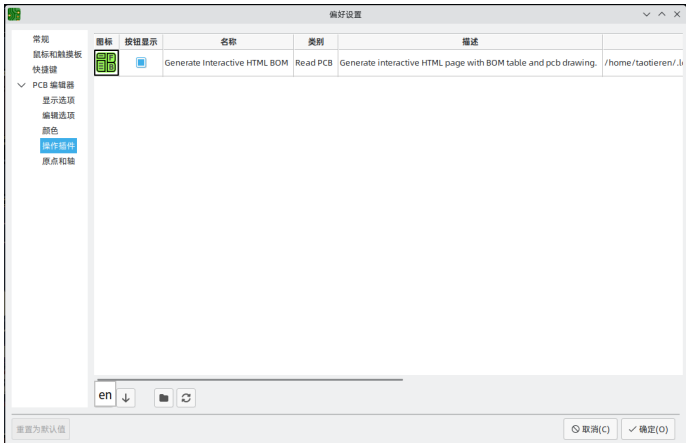
颜色主题存储在位于 KiCad 配置目&#x5f5
`Colors` 子目录中的 JSON 文件中。“打开&#x4e3
KiCad。如果文件是有效的颜色主&#x989

要创建一个新的颜色主题，从&#x98
为你的主题输入一个名称，然&#x54
新主题中的颜色将从你创建新&#x4

要更改颜色，请双击或中键单&#x51
默认”颜色主题中的相应条目。

颜色主题会自动保存；当您关&#x95

8.1.4 操作插件



KiCad PCB 编辑器支持用 Python 编写的插件&#xff
这些插件可以使用内置的插件&#x548
KiCad 章节），或者将插件文件放在
详见下面的脚本部分。

每个被检测到的插件都会在这&#x4e
插件可以在 PCB 编辑器的顶部工具
如果一个插件的 "显示按钮" 控制&
"工具" > "外部插件" 菜单中访问。

列表底部的箭头控制允许改变&#x
文件夹按钮将启动一个文件资&#x6
刷新按钮将扫描插件文件夹中&#x7

8.1.5 原点和轴



Display origin: Determines which coordinate origin is used for coordinate display in the editing canvas. The page origin is fixed at the corner of the page. The drill/place file origin and the grid origin can be moved by the user.

X axis: Controls whether X-coordinates increase to the right or to the left.

Y axis: Controls whether Y-coordinates increase upwards or downwards.

8.2 自定义设计规则

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.

自定义设计规则存储在一个扩&#x
`kicad_dra` 的单独文件中。当您开始向&
`kicad_dra` 文件与 `kicad_pcb` 和 `kicad_pro` 文件一起保&#

Note
`kicad_dra` 文件由 KiCad 自动管理，不应使用&
始终使用电路板设置对话框的自&#x

8.2.1 自定义规则编辑器

自定义规则编辑器位于电路板&#x8b

最好在编辑自定义规则后使用
检查规则语法 按钮，以确保没&#x

8.2.2 自定义规则语法

自定义设计规则语言基于 s 表达&#x
条件，以及一个定义要应用于&#x5
约束。

该语言使用圆括号 ((和)) 来定义&#x76f8
(必须有匹配的) 。在子句中，标&
" 或 ' 引起来，也可以不加引号。&#
" 作为外引号字符，使用 ' 作为内&#
(反之亦然)，从而实现单层嵌套&#x

在下面的语法描述中，< 尖括&#x53f7
> 中的项表示必须存在的标记，[
方括号] 中的项表示可选或仅&#x6

自定义规则文件必须以定义规&#x5
从 KiCad 6.0 开始，版本是 1。 版本头的&#
(version<number>) 。因此，在 KiCad 6.0 中，标题应

```
(version 1)
```

在版本标题之后，您可以输入&#x4e

例如，如果您创建一条规则来&#x9
HV 中的布线与任何其他网络中的&#
HV 网络中的布线落在规则区域内&#x

每条规则必须有一个名称和一&#x4
约束 (constraint) 子句。该名称可以是&
DRC 报告中引用该规则。约束 (constrai
定义了规则的行为。规则还可&#x4
条件 (condition) 子句，决定哪些对象&#
层 (layer) 子句，指定该规则适用于&

```
(rule <name>
    [(layer <layer_name>)]
    [(condition <expression>)]
    (constraint <constraint_type> [constraint_arguments]))
```

定制规则文件还可以包括描述&#x89
字符开头的行表示 (不包括空格)&

```
# Clearance for 400V nets to anything else
# 400V  ↩
    &#x7f51;&#x7edc;&#x4e0e;&#x4efb;&#x4f55;&#x5176;&#x4ed6;&#x7f51;&#x7edc;&#x4e4b;&#x95f4;&#x7684;&
(rule HV
    (condition "A.NetClass == 'HV'")
    (constraint clearance (min 1.5mm))))
```

8.2.2.1 图层子句

层 (layer) 子句确定规则将对哪些层&
约束 (constraint) 子句中进行测试，但&
层 (layer) 子句效率更高。

层 (layer) 子句的值可以是任何线路&#
(F.Cu 和 B.Cu) 匹配的快捷方式 外 (outer) ，&#x4
内 (inner) 。

如果省略 层 (layer) 子句，则该规则

下面是一些示例：

```
# Do not allow footprints on back layer (no condition clause means this rule always applies ↩
    )
# &#x4e0d;&#x5141;&#x8bb8;&#x5728;&#x5e95;&#x5c42;&#x4e0a;&#x7559;&#x4e0b;&#x5c01;&#x88c5;  ↩
    ( ↩
    &#x65e0;&#x6761;&#x4ef6;&#x5b50;&#x53e5;&#x8868;&#x793a;&#x6b64;&#x89c4;&#x5219;&#x59cb;&#x7ec8;&

(rule "Top side footprints only"
    (layer B.Cu)
    (constraint disallow footprint))

# This rule does the same thing, but is less efficient
#  ↩
    &#x6b64;&#x89c4;&#x5219;&#x6267;&#x884c;&#x76f8;&#x540c;&#x7684;&#x64cd;&#x4f5c;&#xff0c;&#x4f46;&
(rule "Top side footprints only"
    (condition "A.Layer == 'B.Cu'")
    (constraint disallow footprint))

# Larger clearance on outer layers (inner layer clearance set by board minimum clearance)
# &#x5916;&#x5c42;&#x95f4;&#x9699;&#x8f83;&#x5927; ( ↩
    &#x5185;&#x5c42;&#x95f4;&#x9699;&#x7531;&#x7535;&#x8def;&#x677f;&#x6700;&#x5c0f;&#x95f4;&#x9699;&

(rule "clearance_outer"
    (layer outer)
    (constraint clearance (min 0.25mm)))
```

8.2.2.2 条件子句

规则 **条件** 是一个包含在文本字&#
该表达式是针对设计规则检查&#x56

例如，当检查铜对象之间的间&#x96
如果存在一个自定义规则，其&#x8

被测对象在表达式语言中称为
`A` 和 `B`。这两个对象的顺序并不重&#
`A` 为布线，`B` 为过孔。 有一些表达&#
`AB` 作为对象名。

条件中的表达式必须解析为布&#x5c
(`真 (true` 或 `false`)。如果表达式解析为
`true`，则规则应用于给定的对象。

每个被测对象都有可以比较的
属性，以及可以执行特定测试&#x7
函数。属性和函数的使用语法&#x5
`<object>.<property>` 和 `<object>.<function>([arguments])` 。译者注&#xff1
`<对象>.<属性>` 和 `<对象>.<函数>([&#x53c2`

Note
当您在文本编辑器 (`A`.、`B`. 或 `AB`.)
中键入 `<对象 (object)>`. 时，会出现一&#x4e2

使用 **布尔运算符** 比较对象属性&#
C/C++ 语法，并支持以下运算符：

`==`	等于
`!=`	不等于
`>`, `>=`	大于、大于或等于
`<`, `\<=`	小于、小于或等于
`&&`	和
`||`	或

例如，`A.NetClass == 'HV'` 将适用于任何属&#x4e8e
"HV" 网类的对象，`A.NetClass != B.NetClass` 将适用&#x

有些属性表示物理测量，比如&#x5
在这些属性上，**单位后缀** 可以&#x5
如果没有使用单位后缀，属性&#x76
支持以下后缀：

`mm`	毫米
`mil`, `th`	千分之一英寸 (mils)
`in`, `"`	英寸
`deg`	度
`rad`	弧度

Note
自定义设计规则中使用的单位独&#
PCB 编辑器中的显示单位。

8.2.2.3 约束

规则的 **约束** 子句定义了规则在&#
约束类型 和一个或多个设置约&#x

(如 `间隙 (clearance)`、`布线宽度 (trace_width)`)。

许多约束条件的参数指定了一&#x4
这些约束条件支持最小值、最&#x4f
"min/opt/max"）。 **最小** 和 **最大** 值用于设&#x8ba
DRC 错误。 **最优** 值仅用于某些约&#x675
KiCad 默认使用的 "最优" 值。 例如，&#x6
`diff_pair_gap` 是由布线器在放置新的差&#x5
如果后来修改了差分对，使得&#x5d
在所有接受最小/最大/最优值的&#

最小/最优/最大值被指定为 `(min<value>)`, `(opt<value>)`, 和 `(max<value>)` 。例如，布线宽度&#x7ea6
`(constraint track_width (min 0.5mm) (opt 0.5mm) (max 1.0mm))` ，如果只&#x7
`(constraint track_width (min 0.5mm))。

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 `clearance` constraint with the `min` value less than zero (for example, `-1`).
`courtyard_clearance`	min	Checks the clearance between footprint courtyards and generates an error if any two courtyards are closer than the `min` 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, `(constraint disallow track)` or `(constraint disallow track via pad)`. 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 `Edge.Cuts` 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 `min` value (if specified) or above the `max` 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 `min` value (if specified) and the larger (major) diameter will be tested against the `max` 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 对象属性和函数参考

可以在自定义规则表达式中测&#x8

8.2.3.1 常见属性

这些属性适用于所有 PCB 对象。

属性数据类型		
`Layer`	string	对象所在的板层。对& 该属性将返回第一层 (例如，对于大多数通& `F.Cu`)。
`Locked`	boolean	如果对象已锁定，则& True。
`Parent`	string	返回此对象的父对象& 这些属性适用于所有 PCB 对象。
`Position_X`	dimension	对象原点在 X 轴上的位置。请注意&# 例如，封装的原点是&# (0, 0) 坐标的位置， 但是封装的设计可能&#
`Position_Y`	dimension	对象原点在 Y 轴上的位置。请注意&# 始终在内部使用从屏&# Y 坐标， 即使您已将设置配置&# Y 坐标。
`Type`	string	"封装 (Footprint)", "焊盘 (Pad)", "图形形状 (Graphic Shape)", "电路板文本 (Board Text)", "封装文本 (Footprint Text)", "敷铜 (Zone)", "布线 (Track)", "过孔 (Via)", "敷铜 (Zone)", 或 "组合 (Group)" 之一.

8.2.3.2 连接的对象属性

这些属性适用于可以分配网的&#x9

属性数据类型		
Net	integer	铜对象的网络码。 请注意，不能依赖网&#x Net 可以用来比较两个性& 例如 A.Net == B.Net 比 A.NetName == B.NetName 快。
NetClass	string	铜对象的网络类的名&
NetName	string	铜对象的网络名称。

8.2.3.3 封装属性

These properties apply to footprints.

属性数据类型		
Clearance_Override	dimension	为封装设置的铜间隙&
Orientation	double	封装的方向 (旋转) (单位：度)。
Reference	string	封装的位号。
Solderpaste_Margin_Override	dimension	为封装设置的焊膏边&
Solderpaste_Margin_Ratio_Override	dimension	为封装设置的焊膏余&
Thermal_Relief_Gap	dimension	为封转设置的散热间&
Thermal_Relief_Width	dimension	为封装设置的散热连&
Value	string	封装的 "值" 字段的内容。

8.2.3.4 焊盘属性

These properties apply to footprint pads.

属性数据类型		
Clearance_Override	dimension	为焊盘设置的铜间隙&
Fabrication_Property	string	"无"、"BGA 焊盘"、"基准, 全局到电路板"、"基准 本地到封装"、"测试&#x70b "散热片焊盘"、"蜂窝&#x72b 之一。
Hole_Size_X	dimension	焊盘在 X 轴上的通孔/槽的大小&
Hole_Size_Y	dimension	焊盘在 Y 轴上的通孔/槽的大小&
Orientation	double	焊盘的方向 (旋转) (单位：度)。
Pad_Number	string	焊盘的 "编号"，可以是字符串 (例如，BGA 中的 "A1")。
Pad_To_Die_Length	dimension	焊盘的 "焊盘到芯片长度" 属性的值，它是在计&

属性	数据类型	
Pad_Type	string	"通孔"、"贴片"、"板边&#x8 或 "非导通孔，机械" 之一。
Pin_Name	string	焊盘的名称 (通常是原理图中相应&#x
Pin_Type	string	焊盘的电气类型 (通常取自原理图中相& "输入"、"输出"、"双向"&#x3 "电源输入"、"电源输&#x51fa 或 "未连接" 之一。
Round_Radius_Ratio	double	对于圆形矩形焊盘，&#
Shape	string	"圆形"、"矩形"、"椭圆&#x5 或 "自定义" 之一。
Size_X	dimension	焊盘在 X 轴上的大小。
Size_Y	dimension	焊盘在 Y 轴上的大小。
Soldermask_Margin_Override	dimension	为焊盘设置的阻焊边&#
Solderpaste_Margin_Override	dimension	为焊盘设置的焊膏边&#
Solderpaste_Margin_Ratio_Override	dimension	为焊盘设置的焊膏边&#
Thermal_Relief_Gap	dimension	为焊盘设置的散热间&
Thermal_Relief_Width	dimension	为焊盘设置的散热连&

8.2.3.5 布线和圆弧属性

These properties apply to tracks and arc tracks.

属性	数据类型	
Origin_X	dimension	起点的 X 坐标。
Origin_Y	dimension	起点的 Y 坐标。
End_X	dimension	终点的 X 坐标。
End_Y	dimension	终点的 Y 坐标。
Width	dimension	布线或圆弧的宽度。

8.2.3.6 过孔属性

These properties apply to vias.

属性	数据类型	
Diameter	dimension	过孔焊盘的直径。
Drill	dimension	过孔成品通孔的直径&
Layer_Bottom	string	过孔压层中的最后一&
Layer_Top	string	过孔压层中的第一层&
Via_Type	string	"通孔"、"盲孔/埋孔" 或 "微孔" 之一。

8.2.3.7 覆铜和规则区域属性

这些属性适用于铜区和非铜区&#xf

属性	数据类型	
Clearance_Override	dimension	为覆铜设置的铜间隙&

属性数据	类型	
Min_Width	dimension	覆铜中允许的填充区
Name	string	用户指定的名称 (默认情况下为空)。
Pad_Connections	string	"继承"、"无"、"焊盘散&#x70 之一
Priority	int	覆铜的优先级别。
Thermal_Relief_Gap	dimension	为覆铜设置的散热间&#
Thermal_Relief_Width	dimension	为覆铜设置的散热连&#

8.2.3.8 图形形状属性

这些属性适用于图形线、圆弧&#x30

属性数据	类型	
End_X	dimension	终点的 X 坐标。
End_Y	dimension	终点的 Y 坐标。
Thickness	dimension	形状画笔的粗细。

8.2.3.9 文本属性

这些属性适用于文本对象（封&#x88

属性数据	类型	
Bold	boolean	如果文本为粗体，则 true。
Height	dimension	字体中字符的高度。
Horizontal_Justification	string	水平文本对齐 (对齐)："向左对齐"、"&#x5c4 或 "向右对齐" 之一。
Italic	boolean	如果文本为斜体，则 true。
Mirrored	boolean	如果文本为镜像，则 true。
Text	string	文本对象的内容。
Thickness	dimension	字体笔划的粗细。
Width	dimension	字体中字符的宽度。
Vertical_Justification	string	垂直文本对齐方式："& 或 "向下对齐" 之一。
Visible	boolean	如果文本对象可见 (显示)，则为 true。

8.2.3.10 表达式函数

可以对自定义规则表达式中的&#x5b

Function	Objects	Description
existsOnLayer('layer_id')	Any	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
`inDiffPair('x')`	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, `inDiffPair('/USB_')` or `inDiffPair('/USB') return `true` for objects in the nets `/USB_P` and `/USB_N`. The `*` can be used as a wildcard, so `inDiffPair('/USB*')` matches `/USB1_P` and `/USB1_N`. 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 `/USB_P` but no net named `/USB_N`, this function returns false.
`insideArea('x')`	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.
`insideCourtyard('x')` `insideFrontCourtyard('x')` `insideBackCourtyard('x')`	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 `\*` wildcard can be used in the reference: `insideCourtyard('R*')` would check all footprints with references that start with `R`.
`isBlindBuriedVia()`	A or B	Returns true if the object is a blind/buried via.
`isCoupledDiffPair(A,B)`	A	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 `/USB+` and `B` is in net `/USB-`.
`isMicroVia()`	A or B	Returns true if the object is a microvia.
`isPlated()`	A or B	Returns true if the object is a plated hole (in a pad or via).
`memberOf('x')`	A or B	Returns true if the object is a member of the named group `x`.

8.2.4 自定义设计规则示例

```
(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 脚本

脚本允许您使用 Python 语言自动执&#x884
KiCad 中的任务。可以通过 Python "操作插&
向 KiCad 添加功能，这些插件可以添&
KiCad 文件交互的独立脚本，例如，&

本手册涵盖了一般脚本编写概&#x
https://docs.kicad.org/doxygen-python/namespaces.html 上的 Doxygen 文档。

KiCad 6 或更新版本需要 Python 3 来支持脚&#x67
2 已不再被支持。

8.3.1 Python 脚本位置

PCB 编辑器的插件脚本可以通过插&
每个插件都应该在 `plugins` 文件夹内&#x
`plugins` 文件夹的位置默认为：

平台	路径
Linx	`~/.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 操作插件

Note
TODO：编写本部分 (如何安装新的操&#x4

8.3.3 封装向导

封装向导是可以从封装编辑器&#x8b
Python 脚本的集合。 如果调用封装&#x5b

如果插件未正确分发到您的系&#x7
KiCad 源代码树中的链接中找到最&#x65b

8.3.4 使用脚本控制台

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：记录其他有用的 PyAlaMode 函数

8.3.5 编写外部脚本

Note
TODO：写下这一节

8.3.6 编写操作插件

Note
TODO：写下这一节

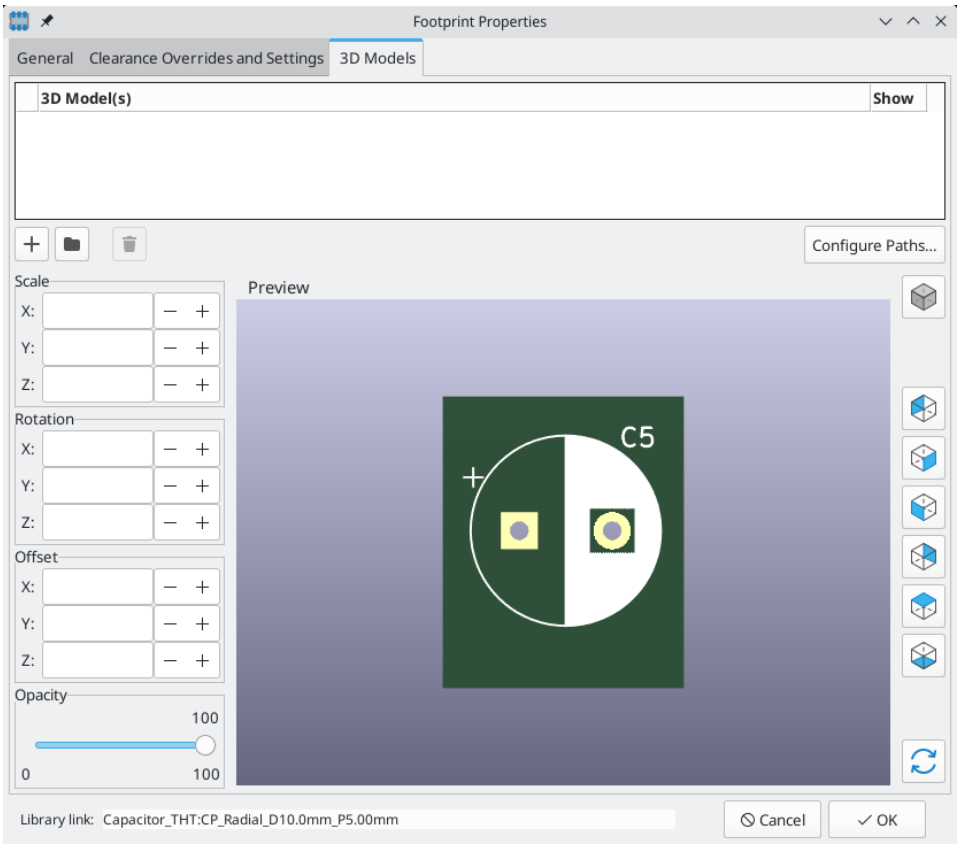
8.4 Working With IDF Component Outlines

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.

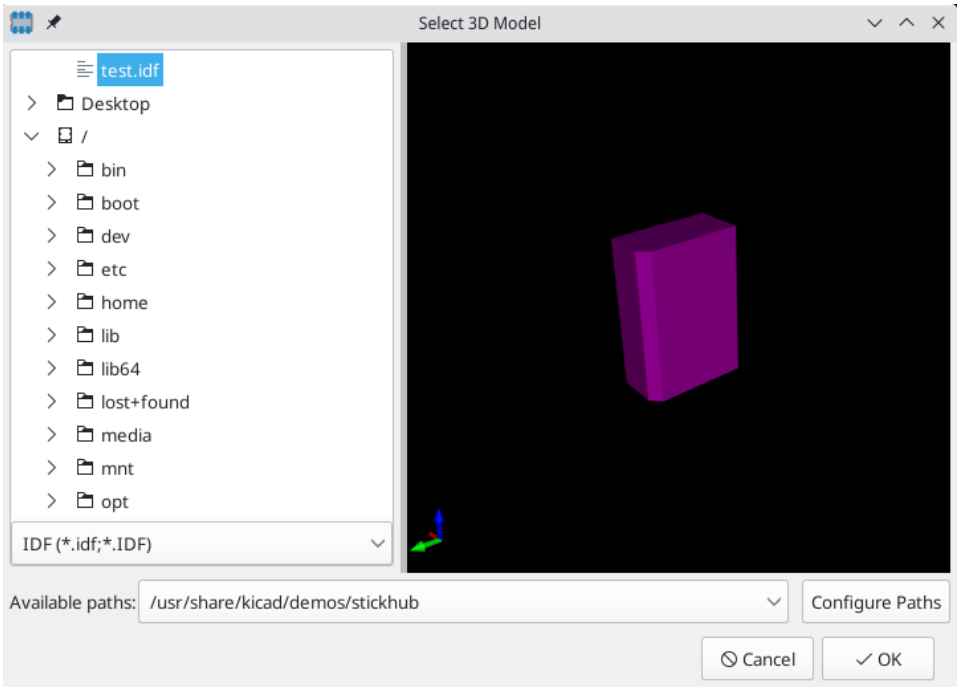
8.4.1 Specifying component models for use by the exporter

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

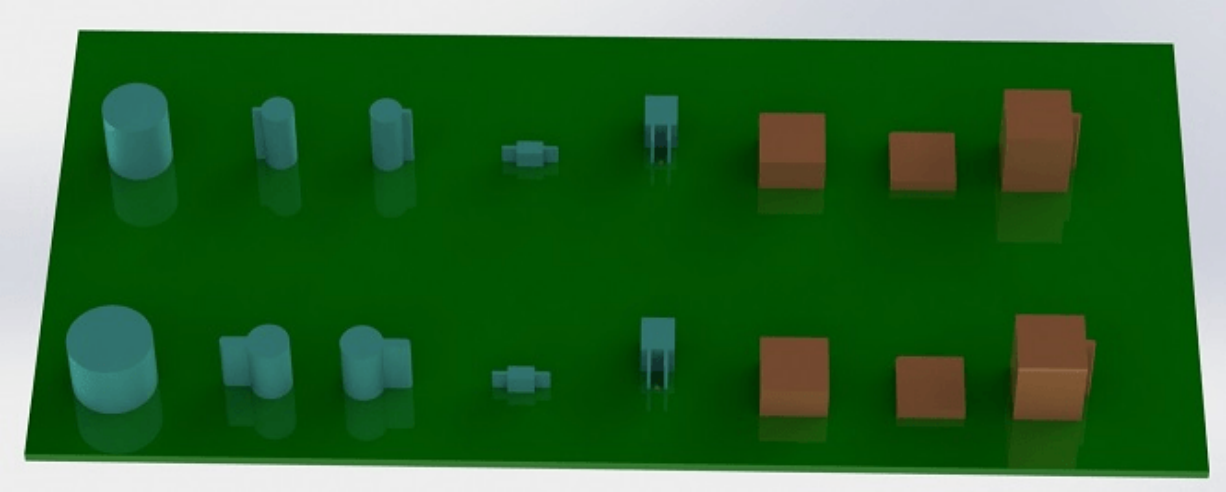
Use comments in the IDF file to give users more information about the outline, for example a reference to the source used for dimensional information.

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 leaded cylinder, (b) vertical axial leaded cylinder with wire on left, (c) vertical axial leaded cylinder with wire on right, (d) horizontal axial leaded cylinder, (e) horizontal radial leaded 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. `dxf2idf`: 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
idfcyl - 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 idf2vrml

The `idf2vrml` 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 `idf2vrml` without any arguments will result in the display of a usage message:

```
>./idf2vrml
Usage: idf2vrml -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: idf2vrml -f input.emn -s 0.3937008 -k
```

Note

The `idf2vrml` 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

操作参考

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

9.1 PCB 编辑器

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

操作	默认快捷键	
向下对齐		将选定项目与下边
与垂直中心对齐		将选定项目垂直居
与水平中心对齐		将选定项目与水平
向左对齐		将选定项目与左边
向右对齐		将选定项目与右边
向上对齐		将选定项目与上边
水平分布		沿水平轴分布选定
垂直分布		沿垂直轴分布选定
放置电路板外轮廓		放置电路板区域外
放置选择的元件选择的元件		
翻转电路板元件		反面看板
草图图形项目		轮廓模式下显示
降低图层不透明度		前图层的透
增加图层不透明度		前图层的透
切换到内层 (F.Cu)	切换层件	
切换到内层 1		
切换到内层 2		
切换到内层 3		
切换到内层 4		
切换到内层 5		
切换到内层 6		
切换到内层 7		

操作	默认设置键
切换到内层 8	
切换到内层 9	
切换到内层 10	
切换到内层 11	
切换到内层 12	
切换到内层 13	
切换到内层 14	
切换到内层 15	
切换到内层 16	
切换到内层 17	
切换到内层 18	
切换到内层 19	
切换到内层 20	
切换到内层 21	
切换到内层 22	
切换到内层 23	
切换到内层 24	
切换到内层 25	
切换到内层 26	
切换到内层 27	
切换到内层 28	
切换到内层 29	
切换到内层 30	
切换到内层 (B.Cu) 层	下一层
切换到内层 [1]	上一层
切换图层 [2]	在活动层对中的&#x5c42
网络检查器	显示网络检查器
高亮飞线	显示所选项目的&#x98de
草图焊盘	在轮廓模式下显示&
弯曲的飞线	用曲线显示飞线
修复电路板	运行各种诊断程序

操作	默认影响	影响
显示外观管理器/隐藏外观管理		
显示焊盘编辑器		显示焊盘编号
脚本控制台		显示 Python 脚本控制台
显示飞线		显示电路板飞线
草图文本项		在行模式下显示封
草图布线[K]		在轮廓模式下显示
草图过孔		在轮廓模式下显示
线框区域		仅显示区域边界
填充覆铜		显示覆铜的填充&#x533a
草图区域		在轮廓模式下显示
切换区域[A]	显示	在显示填充区域&#x3001
自动缩放以适应		封装时缩放以
转换为圆弧		将选定的线段转换
转换为规则圆弧		从所选内容创建规
转换为线		从所选内容创建图
转换为多边形		从所选内容创建图
转换为布线		将选定的图形线转
转换为覆铜		从所选内容创建覆
设计规则检查		显示设计规则检&#x67e5
在封装过程中使用软件辑器中打		
附加电路板...		打开另一个电路板
电路板设置...		编辑电路板设置，
清除网络[L]	高亮	清除任何现有网络
钻孔/放置文...		钻孔文件和元
导出 Specctra DSN...		导出 Specctra DSN 布线信息
BOM...		从电路板创建 BOM 表
IPC-D-356 网表文件...		生成 IPC-D-356 网表文件
钻孔文件 (.drl)...		生成 Excellon 钻孔文件
Gerbers (.gbr)...		为制造生成 Gerbers
元件放置 (.pos)...		为贴片和插件生成
封装报告 (.rpt)...		从当前电路板创建
组合		将所选项目组合，
进入组合		进入要编辑项目的
离开组合		离开当前组合
隐藏网络		隐藏所选网络的飞
高亮网络[C]		高亮网络中的所有
高亮网络		高亮网络中的所有
导入网表...		辻取网表并更新电
导入 Specctra 会话...		导入布线的 Specctra session (*.ses) 文件
锁定		防止项目在画布&#x4e0a
添加封装[S]		添加封装
添加图层对象		添加图层对齐目标
删除项目		从组合中删除项&#x76ee
切换到原理图		在编辑器 中打开原理图
显示网络		显示所选网络的飞
切换上次网络		上两个高亮的

操作	默认快捷键	
切换锁定	kbd:[Ctrl+L]	锁定或解锁选定&#x9879
切换网格视图	kbd:[Ctrl+']	切换网络高亮
将布线转换为多边形	kbd:[Shift+W]	将图形线转换为多边形或圆弧改为
将布线转换为圆弧	kbd:[W]	将图形线转换为多边形或圆弧改为
解组		取消对任何选定组
解锁		允许在画布上移&#x52a8
减小过孔尺寸	kbd:[,]	减小过孔尺寸更改为
增大过孔尺寸	kbd:[.]	增大过孔尺寸更改为
将区域复制到新的位置		将区域复制到新的位置廓复制&#x5230
合并区域		合并区域
更改封装…		从库中分配不同&#x7684
更改封装…		从库中分配不同&#x7684
清除图形…		清除多余项目等&#x3002
清除布线和过孔		清除多余项目、&#x77ed
编辑文本和属性		编辑文本和图形属性
编辑布线和属性		编辑布线和属性布线和过
全局删除…		从电路板中删除布
移除未使用的层		移除未使用的层或换复通孔&#x710a
交换层…		将布线或图形从一&#x
更新封装…		更新封装以包括库
从库中更新		更新封装以包括库
间隙解析…		显示两个选定对象
约束解析…		显示选定对象的约
显示电路板		显示电路板统计信
添加对象	kbd:[Ctrl+Shift+C]	添加对齐的线性标
绘制圆弧	kbd:[Ctrl+Shift+A]	绘制圆弧
切换圆弧方向	kbd:[Ctrl+Shift+D]	切换圆弧方式
添加中心标记		添加中心标注
绘制圆	kbd:[Ctrl+Shift+C]	绘制圆
关闭轮廓		关闭正在进行的轮
减小线宽	kbd:[Ctrl+-]	减小线宽
删除最一个	kbd:[Ctrl+Backspace]	删除添加到当前&#x9879
绘制图形	kbd:[Ctrl+Shift+P]	绘制图形多边形
增大线宽	kbd:[Ctrl++]	增大线宽
添加引线		添加引线标注
绘制线	kbd:[Ctrl+Shift+L]	绘制线
将线限制在45 度		将图形线限制为垂在45 度
添加正交标记		添加正交标注
放置导线端点	kbd:[Ctrl+Shift+E]	放置图形
绘制矩形		绘制矩形
添加规则区域(禁止布线区)	kbd:[Ctrl+Shift+K]	添加规则区域(禁止布线区)
放置封装(锚点)	kbd:[Ctrl+Shift+N]	放置封装的坐标原(锚点)
添加相关线段	kbd:[Ctrl+Shift+F]	添加线用与现有分区相
添加文本	kbd:[Ctrl+Shift+T]	添加文本项
添加过孔	kbd:[Ctrl+Shift+V]	添加独立过孔
添加填充区域	kbd:[Ctrl+Shift+Z]	添加填充区域
添加区域连接	kbd:[Shift+C]	添加现有分区的&#x526a
获取并使用库中的位号选择封装	kbd:[U]	获取并使用库中的位号选择封装
更改布线宽		更新选定的布线和
创建阵列	kbd:[Ctrl+T]	创建阵列
删除整个区域	kbd:[Shift+Del]	删除选定项目和&#x94dc
重复和存在	kbd:[Ctrl+Shift+D]	复制所选项目，递
弧形布线		添加与所选直线轨

操作	默认快捷键	
更改侧边栏	kbd:[F7]	显示项目翻转到边栏或底部面板
镜像		镜像选择项
精确移动	kbd:[Ctrl+M]	按精确的数量移动一个像素
属性...	kbd:[E]	显示项目属性对话框
逆时针旋转	kbd:[R]	逆时针旋转所选项目
顺时针旋转	kbd:[Shift+R]	顺时针旋转所选项目
带位号复制		将选定项目复制到剪贴板
移动	kbd:[M]	移动选定项目
随位号移动		移动具有指定起点和方向的随机位号
自动完成当前布线	kbd:[C]	自动完成当前布线或对象
分割布线		将布线段拆分为在交点处断开的多个线段
自定义线/多边形	kbd:[Q]	自定义线/多边形工具激活时不显示在屏幕上
差分对之后	kbd:[4]	对比分对交互布线
差分对标注		打开差分对标注设置
拖动 (45 度模式)	kbd:[D]	拖动布线段，同时旋转45 度。
拖动 (自由角度)	kbd:[G]	拖动布线中最近的端点
完成布线	kbd:[Enter]	停止当前布线。
布线高亮模式		将布线切换到高亮模式，便于查看布线细节
分割布线		将布线段分割为在交点处断开的多个线段
放置盲点	kbd:[Alt+Shift+V]	在当前布线的末端点上放置一个新的盲点
放置微调	kbd:[V]	在当前布线的末端点上放置微调点
放置通孔	kbd:[V]	在当前布线的末端点上放置通孔
选择图层	kbd:[Alt+5]	在当前布线的末端点上放置通孔，然后在其他布线上放置匹配的通孔
选择图层	kbd:[5]	在当前布线的末端点上放置通孔，然后在其他布线上放置匹配的通孔
设置层对...		更改布线的活动层，仅影响当前布线的设置
交互式切换	kbd:[Shift+5]	切换当前布线交互式布线
布线推挤模式		将布线切换到推挤模式，便于精确调整位置
单轨交互	kbd:[2]	单轨交互式布线
切换布线形状	kbd:[Shift+7]	切换当前交互式布线
改变拐组类	kbd:[Ctrl+7]	切换当前交互式布线
撤消一	kbd:[Ctrl+Z]	当前布线。
布线绕走模式		将布线切换到绕走模式，便于在不影响其他布线的情况下调整路径
取消选择网格	kbd:[Ctrl+Shift+D]	取消选择网格。
过滤选中项		按类型从选区删除不需要的元素
选择/扩展	kbd:[Shift+F]	按择连接或将现有选区扩展到包含整个连接的完整元素
选择网中的孤立点	kbd:[Shift+I]	选择孤立的点，即不属于任何线段的点
图框		选择原理图图框中的所有对象
同一层次图元素	kbd:[Alt+2]	同一层次图元素对象的快速选择工具
减小幅度	kbd:[-]	将弯曲幅度降低一个等级
增大幅度	kbd:[=]	将弯曲幅度增加一个等级
结束布线	kbd:[Enter]	当前轨迹处停止布线。
长度优先设置	kbd:[Ctrl+L]	长度优先设置当前布线项长度
减小间距	kbd:[,]	将弯曲间距降低一个等级
增大间距	kbd:[.]	将弯曲间距增加一个等级
新建布线	kbd:[N]	开始新的布线。
调整差分对	kbd:[6]	布线长度
调整差分对	kbd:[6]	布线歪斜
调整单轨	kbd:[6]	布线长度
添加微波多边形		添加微波应用点列表创建微波多边形对象
添加微波间距		为微波应用创建指定间距的微波多边形
添加微波线		为微波应用创建指定线段的微波多边形
添加微波短路微波应用创建指定短路微波多边形 (Stub)		为微波应用创建指定短路微波多边形 (Stub)

操作	默认快捷键	
添加微波弧线(Stub)	[illegible]	在布线用创建指引的弧线(弧线)
封装检查器		显示封装检查器&#x7a97
复制封装		
创建封装...		使用封装向导创建&
剪切封装		
从库中删除封装		
编辑封装		在编辑器画布上显
导出封装...		
封装属性...		编辑封装属性
导入封装...		
新建封装	kbd:[Ctrl+N]	创建一个新的空封
粘贴封装		
显示封装树视图	[illegible]	显示封装树视图
将默认焊盘属性设置为当前封装	[illegible]	将默认焊盘属性设置为当前封装
将焊盘属性设置为当前封装	[illegible]	将焊盘属性设置为当前封装
将焊盘属性设置为当前封装	[illegible]	将焊盘属性设置为当前封装
默认焊盘属性	[illegible]	默认焊盘属性设置为当前封装
焊盘重新编辑	[illegible]	焊盘重新编辑封装
将焊盘重新编辑	[illegible]	将焊盘重新编辑封装
添加焊盘		添加焊盘
完成焊盘编辑	[illegible]	完成焊盘编辑
创建拐角	kbd:[Ctrl+R]	创建拐角
移除拐角		移除拐角
相対位置	kbd:[Shift+P]	根据所选项目相对
位置重新批准PCB	[illegible]	位置顺序重新批
填充		填充区域
填充所有	[illegible]	填充所有区域
取消填充		取消填充区域
取消填充所有	[illegible]	取消填充所有区&#x57df

9.2 3D 查看器

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.

操作	默认快捷键	
添加基板		在电路板下面添加(慢)
抗锯齿		在最终渲染中以&#x6700(慢)
切换表面(SMD) 3D 模型	kbd:[S]	切换“表贴”属性的 3D 模型
切换直插 3D 模型	kbd:[T]	切换“直插”属性的 3D 模型
切换虚拟 3D 模型	kbd:[V]	切换“虚拟”属性的 3D 模型
翻转电路板		翻转电路板视图
主视图	kbd:[Home]	主视图
CAD 颜色样式		根据材质的漫反&#x5c04 CAD 颜色样式
仅使用漫反射	[illegible]	仅使用 3D 模型文件中的漫反

操作	默认快捷键	
使用所有属性		使用每个 3D 模型文件中的所&#x6709
下移电路板	kbd:[Down]	下移电路板
左移电路板	kbd:[Left]	左移电路板
右移电路板	kbd:[Right]	右移电路板
上移电路板	kbd:[Up]	上移电路板
不显示 3D 网格		不显示 3D 网格
中心轴旋转	kbd:[Space]	中心轴旋转 (点击鼠标中键)
后期处理		在最终渲染中，应 (慢)
过程纹理		将过程纹理应用于 (慢)
渲染阴影		渲染阴影
重置视图	kbd:[R]	重置视图
沿 Z 轴旋转 45 度	kbd:[Tab]	
沿 X 轴顺时针旋转		沿 X 轴顺时针旋转
沿 X 轴逆时针旋转		沿 X 轴逆时针旋转
沿 Y 轴顺时针旋转		沿 Y 轴顺时针旋转
沿 Y 轴逆时针旋转		沿 Y 轴逆时针旋转
沿 Z 轴顺时针旋转		沿 Z 轴顺时针旋转
沿 Z 轴逆时针旋转		沿 Z 轴逆时针旋转
3D 网格 1 mm		3D 网格 1 mm
3D 网格 2.5 mm		3D 网格 2.5 mm
3D 网格 5 mm		3D 网格 5 mm
3D 网格 10mm		3D 网格 10 mm
显示 3D 轴线		显示 3D 轴线
显示模型边框		显示模型边框
反射		在最终渲染中渲&#x67d3 (慢)
折射		在最终渲染中渲&#x67d3 (慢)
切换胶粘层显示		切换胶粘层的显&#x793a
切换电路板显示		切换电路板显示
切换注释和绘图层		切换注释和绘图层
切换 ECO 层显示		切换 ECO 层的显示
切换正交投影用/禁用正交投&#x5f71		
切换真实模型		切换真实模式
切换丝印层显示		切换丝印层的显&#x793a
切换阻焊层显示		切换阻焊层的显&#x793a
切换锡膏层显示		切换锡膏层的显&#x793a

操作	默认快捷键	
切换区域显示		切换区域显示
后视图	kbd:[Shift+Y]	后视图
底视图	kbd:[Shift+Z]	底视图
正视图	kbd:[Y]	正视图
左视图	kbd:[Shift+X]	左视图
右视图	kbd:[X]	右视图
顶视图	kbd:[Z]	顶视图

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