

KiCad

The KiCad Team

Table of Contents

Introduzione	2
System Requirements	2
Cartelle e file di KiCad	2
Installing and Upgrading KiCad	5
Importing settings	5
Migrating files from previous versions	6
Uso del gestore del progetto di KiCad	7
Finestra del gestore progetti	7
La vista ad albero del progetto	7
Side toolbar	8
Creazione di un nuovo progetto	8
Importing a project from another EDA tool	9
KiCad configuration	10
Common preferences	10
Mouse and touchpad preferences	12
Hotkey preferences	13
Configurazione percorsi	13
Libraries configuration	15
Modelli utente	16
Uso dei modelli	16
Posizione dei modelli:	17
Creazione dei modelli	17
Plugin and Content Manager	21

Manuale di riferimento

Copyright

This document is Copyright © 2010-2021 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.

Tutti i marchi registrati all'interno di questa guida appartengono ai loro legittimi proprietari.

Collaboratori

Jean-Pierre Charras, Fabrizio Tappero, Jon Evans.

Traduzione

Marco Ciampa <ciampix@posteo.net>, 2014-2018.

Feedback

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

Introduzione

KiCad is an open-source software suite for creating electronic circuit schematics and printed circuit boards (PCBs). KiCad supports an integrated design workflow in which a schematic and corresponding PCB are designed together, as well as standalone workflows for special uses. KiCad also includes several utilities to help with circuit and PCB design, including a PCB calculator for determining electrical properties of circuit structures, a Gerber viewer for inspecting manufacturing files, and an integrated SPICE simulator for inspecting circuit behavior.

KiCad runs on all major operating systems and a wide range of computer hardware. It supports PCBs with up to 32 copper layers and is suitable for creating designs of all complexities. KiCad is developed by a volunteer team of software and electrical engineers around the world with a mission of creating free and open-source electronics design software suitable for professional designers.

The latest version of this documentation is available at <https://docs.kicad.org>.

System Requirements

KiCad is capable of running on a wide variety of hardware and operating systems, but some tasks may be slower or more difficult on lower-end hardware. For the best experience, a dedicated graphics card and display with 1920x1080 or higher resolution is recommended.

Please check the KiCad website for the latest system requirements: <https://kicad.org/help/system-requirements/>

Cartelle e file di KiCad

KiCad crea e usa file con le seguenti estensioni (e cartelle) per la modifica di schemi e schede.

File del gestore progetti:

*.kicad_pro	Project file, containing settings that are shared between the schematic and PCB
*.pro	Legacy (KiCad 5.x and earlier) project file. Can be read and will be converted to a <i>.kicad_pro</i> file by the project manager.

File dell'editor degli schemi elettrici:

*.kicad_sch	Schematic files containing all info and the components themselves.
*.kicad_sym	Schematic symbol library file, containing the component descriptions: graphic shape, pins, fields.
*.sch	Legacy (KiCad 5.x and earlier) schematic file. Can be read and will be converted to a <i>.kicad_sch</i> file on write.
*.lib	Legacy (KiCad 5.x and earlier) schematic library file. Can be read but not written.
*.dcm	Legacy (KiCad 5.x and earlier) schematic library documentation. Can be read but not written.
*-cache.lib	Legacy (KiCad 5.x and earlier) schematic component library cache file. Required for proper loading of a legacy schematic (<i>.sch</i>) file.
sym-lib-table	Symbol library list (<i>symbol library table</i>): list of symbol libraries available in the schematic editor.

File e cartelle dell'editor di circuiti stampati:

*.kicad_pcb	Board file containing all info but the page layout.
*.pretty	Footprint library folders. The folder itself is the library.
*.kicad_mod	Footprint files, containing one footprint description each.
*.kicad_dru	Design rules file, containing custom design rules for a certain <i>.kicad_pcb</i> file.
*.brd	Legacy (KiCad 4.x and earlier) board file. Can be read, but not written, by the current board editor.
*.mod	Legacy (KiCad 4.x and earlier) footprint library file. Can be read by the footprint or the board editor, but not written.
fp-lib-table	Footprint library list (<i>footprint library table</i>): list of footprint libraries available in the board editor.
fp-info-cache	Cache to speed up loading of footprint libraries.

File comuni:

*.kicad_wks	Page layout (drawing border and title block) description file
*.net	Netlist file created by the schematic, and read by the board editor. This file is associated to the <i>.cmp</i> file, for users who prefer a separate file for the component/footprint association.
*.kicad_prl	Local settings for the current project, helps Kicad remember the last used settings such as layer visibility or selection filter. May not need to be distributed with the project or put under version control.

Altri file:

*.cmp	Associazione tra componenti usati nello schema elettrico e le loro impronte. Possono essere creati da Pcbnew, e importati da Eeschema. Lo scopo è la reimportazione da Pcbnew a Eeschema, per utenti che cambiano impronte da dentro Pcbnew (per esempio usando il comando <i>Scambia impronte</i>) e vogliono importare questi cambiamenti nello schema.
-------	--

Altri file:

Sono generati da KiCad per la fabbricazione o per la documentazione.

*.gbr	File gerber, per la fabbricazione
*.drl	File di foratura (formato Excellon), per la fabbricazione
*.pos	File di posizionamento (formato ascii), per le macchine per l'inserzione automatica
*.rpt	File di rapporto (formato ascii), per documentazione
*.ps	File di tracciatura (formato postscript), per documentazione
*.pdf	File di tracciatura (formato pdf), per documentazione
*.svg	File di tracciatura (formato svg), per documentazione
*.dxf	File di tracciatura (formato dxf), per documentazione
*.plt	File di tracciatura (formato HPGL), per documentazione

Storing and sending KiCad files

KiCad schematic and board files contain all the schematic symbols and footprints used in the design, so you can back up or send these files by themselves with no issue. Some important design information is stored in the project file (*.kicad_pro*), so if you are sending a complete design, make sure to include it.

Some files, such as the project local settings file (*.kicad_prj*) and the *fp-info-cache* file, are not necessary to send with your project. If you use a version control system such as Git to keep track of your KiCad projects, you may want to add these files to the list of ignored files so that they are not tracked.

Installing and Upgrading KiCad

Importing settings

Each major release of KiCad has its own configuration, so that you may run multiple KiCad versions on the same computer without the configurations interfering. The first time you run a new version of KiCad, you will be asked how to initialize the settings:



If a previous version of KiCad is detected, you will have the option to import the settings from that version. The location of the previous configuration files is detected automatically, but you may override it to choose another location if desired.

By default, the schematic symbol and footprint library tables from the previous version of KiCad will also be imported. If you would like to start with no library configuration, uncheck the **Import library configuration from previous version** checkbox.

You may also choose to start with default settings if you do not want to import settings from a previous version.

KiCad stores the settings files in a folder inside your user directory. Each KiCad version will store its settings in a subfolder of that folder (except for KiCad 5.1 and earlier, which did not use subfolders). Those folders are:

Windows	%APPDATA%\kicad
Linux	~/.config/kicad
Mac OS	/Users/<username>/Library/Preferences/kicad

Migrating files from previous versions

Modern versions of KiCad can open files created in earlier versions, but can only write files in the latest formats. This means that in general, there are no special steps to migrate files from a previous version besides opening the files. In some cases, the file extension for a file has changed from one KiCad version to the next. After opening these files, they will be saved in the new format with the new file extension. The old files will not be deleted automatically.

In general, files created or modified by one version of KiCad **cannot** be opened by older versions of KiCad. For this reason, it is important to keep backup copies of your projects when testing a new KiCad release, until you are confident that you will not need to use the older KiCad version anymore.

NOTE

Hotkey configurations are not imported from previous versions at this time. You can manually import hotkey configurations by copying the various `*.hotkeys` files from the old version configuration directory to the new one. If you do so, please note that KiCad will not automatically detect conflicts such as one key being assigned to multiple actions.

Uso del gestore del progetto di KiCad

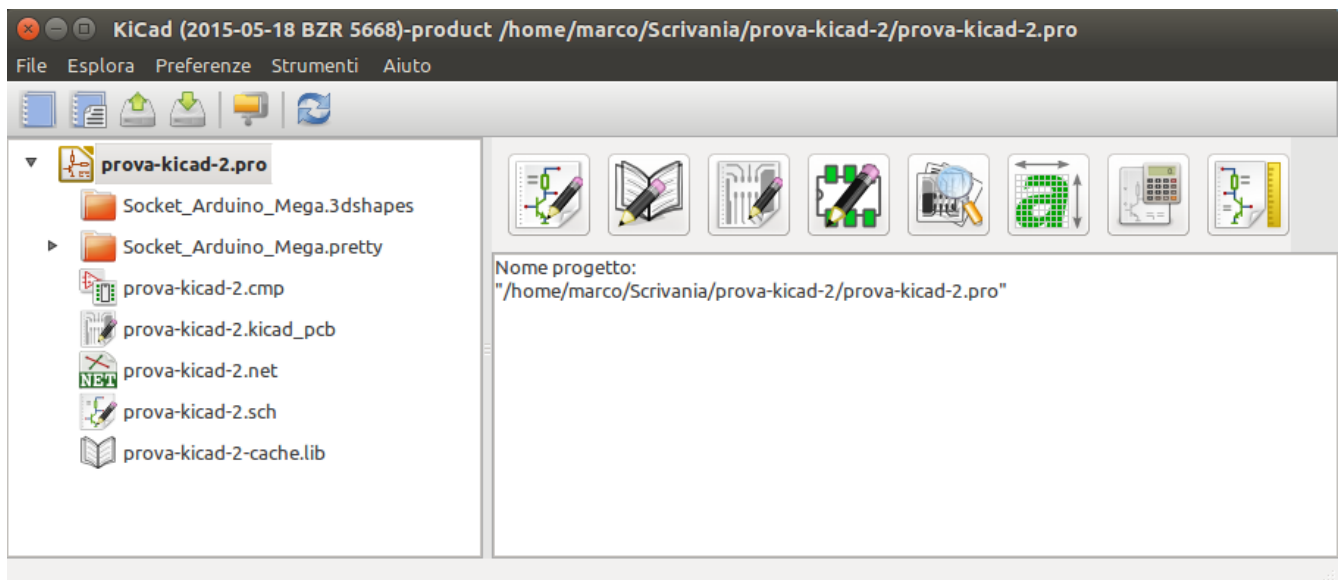
KiCad project manager (kicad or kicad.exe) is a tool which can easily run the other tools (schematic and board editors, Gerber viewer and utility tools) when creating a design.

L'esecuzione degli altri strumenti dal gestore del progetto di KiCad ha alcuni vantaggi:

- controllo incrociato tra editor di schemi elettrici e editor di circuiti stampati.
- synchronization of the design between the schematic editor and board editor (without creating netlist files)

KiCad currently only supports having one project open at a time. When running the schematic and board editors from the KiCad project manager, you can only edit the schematics and board associated with the open project. When these tools are run in *stand alone* mode, you can open any file in any project, but cross probing between tools can give strange results.

Finestra del gestore progetti



The KiCad project manager window is composed of a tree view on the left showing the files associated with the open project, and a launcher on the right containing shortcuts to the various editors and tools.

La vista ad albero del progetto







The tree view shows a list of files inside the project folder. Double-clicking on a file in the tree view will open it in the associated editor. Right-clicking on a file will open a context menu with some file manipulation commands.



NOTE Only files that KiCad understands how to open are displayed in the project tree view.

Side toolbar

The toolbar on the left side of the window provides shortcuts for common project operations:

	Create a new project.
	Open an existing project.
	Create a zip archive of the whole project. This includes schematic files, libraries, PCB, etc.
	Extract a project zip archive into a directory. Files in the destination directory will be overwritten.
	Refresh the tree view, to detect changes made on the filesystem.
	Open the project working directory in a file explorer.

Creazione di un nuovo progetto

Most KiCad designs start with the creation of a project. There are two ways to create a project from the KiCad project manager: you may create an empty project, or create a project based on an existing template. This section will cover the creation of a new, empty project. Creating projects from templates is covered in the [Project Templates](#) section.

To create a new project, use the **New Project...** command in the **File** menu, click the **New Project** button in the top toolbar, or use the keyboard shortcut (**Ctrl+N** by default).

You will be prompted for a name to give your project. By default, a directory will be created for your project with the same name. For example, if you enter the name `MyProject`, KiCad will create the directory `MyProject` and the project file `MyProject/MyProject.kicad_pro` inside it.

If you already have a directory to store your project files in, you can uncheck the *Create a new directory for the project* checkbox in the **New Project** dialog.

NOTE It is strongly recommended that you store each KiCad project inside its own directory.

Once you select the name of your project, KiCad will create the following files inside the project directory:

example.kicad_pro	KiCad project file.
example.kicad_sch	Main schematic file.
example.kicad_pcb	Printed circuit board file.

Importing a project from another EDA tool

KiCad is able to import files created by some other software packages. Currently the following types of project are supported:

*.sch, *.brd	Eagle 6.x or newer (XML format)
*.csa, *.cpa	CADSTAR archive format

To import a project from one of these tools, choose the appropriate option from the **Import Non-KiCad Project** submenu of the **File** menu.

You will be prompted to select either a schematic or a board file in the import file browser dialog. The imported schematic and board files should have the same base file name (e.g. project.sch and project.brd). Once the requested files are selected, you will be asked to select a directory to store the resulting KiCad project.

KiCad configuration

The KiCad preferences can always be accessed from the **Preferences** menu, or by using the hotkey (default `Ctrl+,`). The Preferences dialog is shared between the running KiCad tools. Some preferences apply to all tools, and some are specific to a certain tool (such as the schematic or board editor).

Common preferences



Accelerated graphics antialiasing: KiCad can use different methods to prevent aliasing (jagged lines) when rendering using a graphics card. Different methods may look better on different hardware, so you may want to experiment to find the one that looks best to you.

Fallback graphics antialiasing: KiCad can also apply antialiasing when using the fallback graphics mode. Enabling this feature may result in poor performance on some hardware.

Text editor: Choose a text editor to use when opening text files from the project tree view.

PDF viewer: Choose a program to use when opening PDF files.

Show icons in menus: Enables icons in drop-down menus throughout the KiCad user interface.

NOTE | Icons in menus are not displayed on some operating systems.

Icon theme: Sets whether to use the icon theme designed for light window backgrounds or dark window backgrounds. The default setting of *Automatic* will choose the theme based on the lightness of the operating system window theme.

Icon scale: Sets the size of the icons used in menus and buttons throughout KiCad. Choose *Automatic* to pick an appropriate icon scale automatically based on your operating system settings.

Canvas scale: Sets the scale of the drawing canvas used in the KiCad editors. Choose *Automatic* to pick an appropriate canvas scale automatically based on your operating system settings.

Apply icon scaling to fonts: This setting will scale fonts used in the UI according to the icon scale setting. This is not needed for most users, but may improve the look of KiCad on certain Linux platforms when using a high-DPI display.

Warp mouse to origin of moved object: When enabled, the mouse cursor will be repositioned (warped) to the origin of an object when you start a move command on that object.

First hotkey selects tool: When disabled, pressing the hotkey for a command such as *Add Wire* will immediately start the command at the current cursor location. When enabled, pressing the hotkey the first time will just select the *Add Wire* tool but will not immediately begin a wire.

Remember open files for next project launch: When enabled, KiCad will automatically re-open any files that were previously open when a project is re-opened.

Auto save: When editing schematics and board files, KiCad can automatically save your work periodically. Set to 0 to disable this feature.

File history size: Configure the number of entries in the list of recently-opened files

3D cache file duration: KiCad creates a cache of 3D models in order to speed up the 3D viewer. You can configure how long to keep this cache before deleting old files.

Automatically backup projects: When enabled, KiCad projects will be archived to ZIP files automatically according to the settings below. The archives will be stored in a subfolder of the project folder. Backups are created when saving files in the project.

Create backups when auto save occurs: When enabled, a backup will be created every time an automatic file save occurs (if the backup is permitted by the settings below). This setting has no effect if the auto save interval is set to 0 (disabled).

Maximum backups to keep: When creating a new backup, the oldest backup file will be deleted to keep the total number of backup files below this limit.

Maximum backups per day: When creating a new backup, the oldest backup file created on the current day will be deleted to stay below this limit.

Minimum time between backups: If backup is triggered (for example, by saving a board file), the backup will not be created if an existing backup file is newer than this limit.

Maximum total backup size: When creating a new backup file, the oldest backup files will be deleted to keep the total size of the backup files directory below this limit.

Remember open files for next project launch: When checked, KiCad will re-open the schematic and board editor if they were open the last time you closed the project manager.

Mouse and touchpad preferences

Preferences

Common
Mouse and Touchpad
Hotkeys

Pan and Zoom

☒ Center and warp cursor on zoom ☐ Automatically pan while moving object

☐ Use zoom acceleration

Zoom speed: ☒ Automatic Auto pan speed:

Drag Gestures

Left button drag: Drag selected objects; otherwise draw selection rectangle ▼

Middle button drag: Pan ▼

Right button drag: Pan ▼

Scroll Gestures

Vertical touchpad or scroll wheel movement:

	–	Ctrl	Shift	Alt
Zoom:	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>	<input type="radio"/>
Pan up/down:	<input type="radio"/>	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>
Pan left/right:	<input type="radio"/>	<input checked="" type="radio"/>	<input type="radio"/>	<input type="radio"/>

☐ Pan left/right with horizontal movement

Reset to Mouse Defaults

Reset to Trackpad Defaults

Reset Mouse and Touchpad to Defaults

Cancel OK

Center and warp cursor on zoom: When enabled, zooming using the hotkeys or mouse wheel will cause the view to be centered on the cursor location.

Use zoom acceleration: When enabled, scrolling the mouse wheel or touchpad faster will cause the zoom to change faster.

Zoom speed: Controls how much the zoom changes for a given amount of scrolling the mouse wheel or touchpad. Use *Automatic* to set a default value depending on your operating system.

Automatically pan while moving object: When enabled, the view can be panned while moving an object by moving close to the edge of the canvas.

Auto pan speed: Controls how fast the canvas pans while moving an object.

Mouse buttons: You can set the behavior of dragging the middle and right mouse buttons to zoom the view, pan the view, or have no effect. You can also set the behavior of dragging the left mouse button depending on whether or not any objects are already selected in the editing canvas.

NOTE | The left mouse button is always used for selecting and manipulating objects.

Mouse wheel and touchpad scrolling: You can set the behavior of scrolling the mouse wheel or vertical motion of the touchpad while pressing certain modifier keys.

Pan left/right with horizontal movement: When enabled, you can pan the view using the touchpad or horizontal scroll wheel (if present on your mouse).

Hotkey preferences



You can use this dialog to customize the hotkeys used to control KiCad. The hotkeys in the *Common* section are shared between every KiCad program. Hotkeys for each specific KiCad program are shown when that program is running. You can assign the same hotkey to a different action in different KiCad programs (for example, the schematic editor and the board editor), but you cannot assign a hotkey to more than one action in the same program.

There are many available commands, and so not all of them have a hotkey assigned by default. You can add a hotkey to any command by double-clicking on the command in the list. If you choose a hotkey that is already assigned to a different command, you can choose to use that hotkey on your chosen command, which will remove the hotkey assignment from the conflicting command.

Changes that you have made to hotkey assignments are shown with a * character at the end of the command name. You can undo changes to a specific command by right-clicking that command and selecting *Undo Changes*, or you can undo all changes with the button below the command list.

Importing hotkeys

Hotkey preferences are stored in `.hotkeys` files in the KiCad settings directory (see the [Settings](#) section for information about where the settings directory is on your operating system). If you have configured KiCad hotkeys the way you like on one computer, you can transfer that configuration to another computer by importing the appropriate `.hotkeys` file(s).

Configurazione percorsi

In KiCad, si possono definire alcuni percorsi usando *variabili ambiente*. Alcune variabili ambiente vengono definite internamente da KiCad, e possono essere usate per definire percorsi per librerie, forme 3D, eccetera.

Ciò è utile quando i percorsi assoluti non sono conosciuti o sono soggetti a cambiamenti (per es. quando si trasferisce un progetto su un altro computer), o anche quando un percorso base viene condiviso con altri

simili. Si consideri per esempio i seguenti che possono essere installati in posizioni variabili:

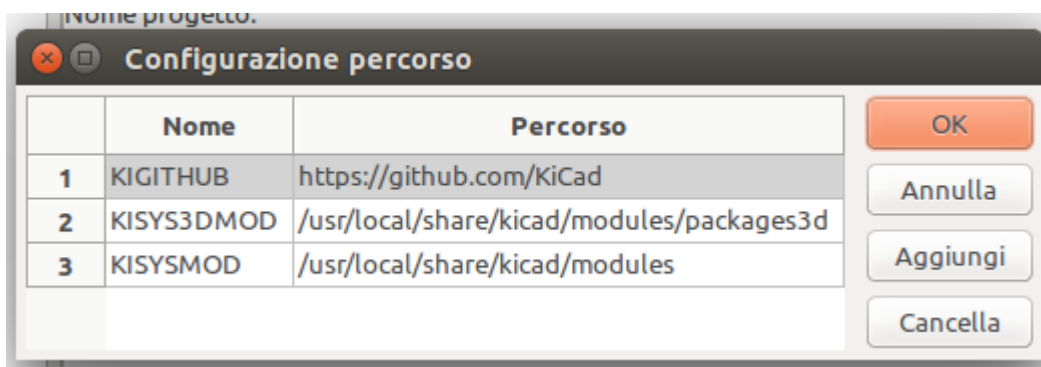
- Schematic symbol libraries
- Footprint libraries
- Forme 3D usate nelle definizioni delle impronte

Per esempio, il percorso completo della libreria di impronte **connect.pretty**, quando si usa la variabile ambiente **KISYSMOD**, sarà definito come **\${KISYSMOD}/connect.pretty**

The **Preferences** → **Configure Paths...** menu allows you to define paths for some built-in KiCad environment variables, and add your own environment variables to define personal paths, if needed.

Variabili ambiente KiCad:

KICAD6_SYMBOL_DIR	Base path of symbol library files.
KICAD6_FOOTPRINT_DIR	Base path of footprint library files.
KICAD6_3DMODEL_DIR	Base path of 3D models used in footprints.
KICAD6_TEMPLATE_DIR	Location of project templates installed with KiCad.
KICAD_USER_TEMPLATE_DIR	Location of personal project templates.
KICAD6_SCRIPTING_DIR	Location of Python scripts installed with KiCad.
KICAD6_USER_SCRIPTING_DIR	Location of personal Python scripts.



NOTE

You cannot override an environment variable that has been set outside of KiCad by using the Configure Paths dialog. Any variable that has been set externally will be shown as read-only in the dialog.

Some advanced environment variables can be set to customize KiCad's behavior. These variables are not shown in the environment variable configuration dialog by default. Changing these variables will not result in KiCad moving any files from the default location to the new location, so if you change these variables you will need to copy any desired settings or files manually.

Additional environment variables:

KICAD_CONFIG_HOME	Base path of KiCad configuration files. Subdirectories will be created within this directory for each KiCad minor version.
KICAD_DOCUMENTS_HOME	Base path of KiCad user-modifiable documents, such as templates, Python scripts, libraries, etc. Subdirectories will be created within this directory for each KiCad minor version.

Si noti anche che la variabile **KIPRJMOD** è **sempre** internamente definita da KiCad, ed è il **percorso assoluto del progetto corrente**.

Per esempio, ***\${KIPRJMOD}/connect.pretty*** è sempre la cartella ***connect.pretty*** (libreria di impronte in formato pretty) trovata ***dentro la cartella del progetto corrente***.

Se si modifica la configurazione dei percorsi, uscire e riavviare KiCad, per evitare qualsiasi problema nella gestione dei percorsi.

Libraries configuration

The **Preferences** → **Manage Symbol Libraries...** menu let you manage the library list files called **symbol library table** (sym-lib-table).

Likewise, use the **Preferences** → **Manage Footprint Libraries...** menu to manage the library list files called **footprint library table** (fp-lib-table).

There are 2 library list files: the first (located in the user home directory) is global for all projects and the second (located in the project directory) is optional and specific to the project.

Modelli utente

L'uso di un modello di progetto facilita la creazione di un nuovo progetto, includendo alcune preimpostazioni. I modelli possono contenere: profili di scheda predefiniti, posizioni di connettori, elementi dello schema, regole di progettazione, ecc. Persino schemi elettrici o circuiti stampati completi possono venire inclusi come spunti per il nuovo progetto.

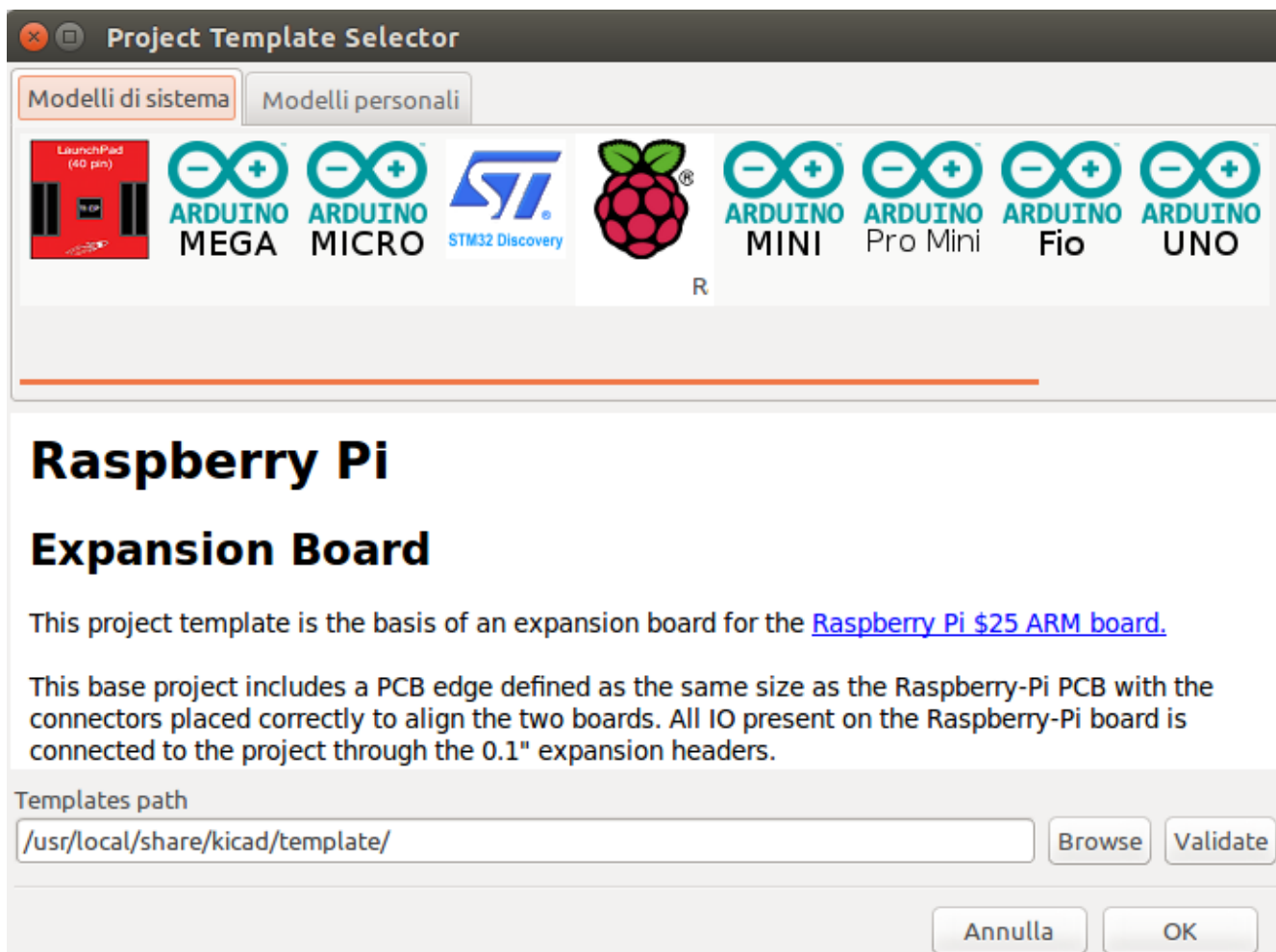
Uso dei modelli

Il menu **File** → **Nuovo progetto** → **Nuovo progetto da modello** aprirà la finestra di dialogo di selezione dei modelli:



Un singolo clic sull'icona di un modello mostrerà le informazioni del modello stesso, mentre un ulteriore clic sul pulsante OK farà partire la creazione del nuovo progetto. I file del modello verranno copiati nella posizione del nuovo progetto e rinominati per riflettere il nome del nuovo progetto.

Dopo la selezione di un modello:



Posizione dei modelli:

KiCad cerca i file dei modelli nei seguenti percorsi:

- percorso definito nella variabile ambiente KICAD_USER_TEMPLATE_DIR
- percorso definito nella variabile ambiente KICAD_TEMPLATE_DIR
- Modelli di sistema: <dir bin kicad>/../share/kicad/template/
- Modelli utente:
 - Unix: ~/kicad/template/
 - Windows: C:\Documents and Settings\username\My Documents\kicad\template or C:\Users\username\Documents\kicad\template
 - Mac: ~/Documents/kicad/template/

Creazione dei modelli

Il nome del modello è il nome della cartella dentro la quale sono memorizzati i file del modello. La cartella dei metadati è una sottocartella di nome **meta**, contenente i file che descrivono il modello.

I metadati consistono in un file necessario e altri file opzionali. Tutti i file devono essere creati dall'utente, usando un editor di testo o usando file di progetto KiCad preesistenti, e devono essere stati sistemati nella necessaria struttura di cartelle.

All files and directories in a template are copied to the new project path when a project is created using a template, except **meta**. Files and directories containing the template name will be renamed with the new project file name.

For example, creating a project called **newproject** from a template named **example**:

Files in template example directory	Files created in project newproject directory
example.kicad_pro example.kicad_sch example.kicad_pcb example-first.kicad_sch second-example.kicad_sch third.kicad_sch third.kicad_pcb	newproject.kicad_pro newproject.kicad_sch newproject.kicad_pcb newproject-first.kicad_sch second-newproject.kicad_sch third.kicad_sch third.kicad_pcb

A template does not need to contain a complete project, if a required project file is missing, KiCad will create it using its default create project behavior:

Files in template example directory	Files created in newproject directory
example.kicad_sch first-example.kicad_sch first-example.kicad_pcb second-example.kicad_sch second-example.kicad_pcb	newproject.kicad_sch first-newproject.kicad_sch first-newproject.kicad_pcb second-newproject.kicad_sch second-newproject.kicad_pcb newproject.kicad_pro (default) newproject.kicad_pcb (default)

As an exception to the template name renaming rule, if one project file (.kicad_pro) exists and its name doesn't match the template name, KiCad will do the renaming based on that project file name instead:

Files in template example directory	Files created in newproject directory
example.kicad_sch example.kicad_pcb first-example.kicad_pro first-example.kicad_sch first-example.kicad_pcb second-example.kicad_sch second-example.kicad_pcb	example.kicad_sch example.kicad_pcb newproject.kicad_pro newproject.kicad_sch newproject.kicad_pcb second-example.kicad_sch second-example.kicad_pcb

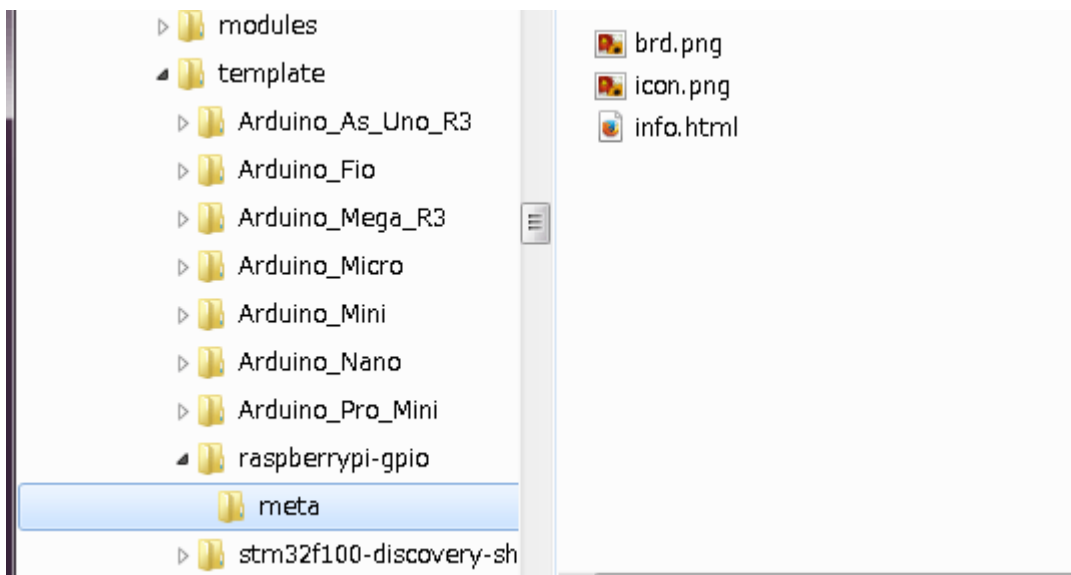
NOTE | It is not recommended to create a template with multiple project files.

Template example

Ecco un esempio che mostra i file di progetto per il modello **raspberrypi-gpio**:



E i file dei metadati:



File richiesti:

meta/info.html	Informazioni di descrizione del modello in formato HTML.
----------------	--

The <title> tag determines the actual name of the template that is exposed to the user for template selection. Note that the project template name will be cut off if it's too long.

Usare l'HTML significa che le immagini possono essere in linea senza doversi inventare un nuovo schema. Solo marcature HTML di base possono essere usate in questo documento.

Ecco un file **info.html** di esempio:

```

<!DOCTYPE HTML PUBLIC "-//W3C//DTD HTML 4.0 Transitional//EN">
<HTML>
<HEAD>
<META HTTP-EQUIV="CONTENT-TYPE" CONTENT="text/html;
charset=windows-1252">
<TITLE>Raspberry Pi - Expansion Board</TITLE>
</HEAD>
<BODY LANG="fr-FR" DIR="LTR">
<P>This project template is the basis of an expansion board for the
<A HREF="http://www.raspberrypi.org/" TARGET="blank">Raspberry Pi $25
ARM board.</A> <BR><BR>This base project includes a PCB edge defined
as the same size as the Raspberry-Pi PCB with the connectors placed
correctly to align the two boards. All IO present on the Raspberry-Pi
board is connected to the project through the 0.1" expansion
headers. <BR><BR>The board outline looks like the following:
</P>
<P><IMG SRC="brd.png" NAME="brd" ALIGN=BOTTOM WIDTH=680 HEIGHT=378
BORDER=0><BR><BR><BR><BR>
</P>
<P>(c)2012 Brian Sidebotham<BR>(c)2012 KiCad Developers</P>
</BODY>
</HTML>

```

File opzionali:

meta/icon.png	Un file icona in formato PNG di 64 x 64 pixel usato come icona cliccabile nella finestra di dialogo di selezione dei modelli.
---------------	---

Qualsiasi altro file immagine usato da **meta/info.html**, come il file immagine della scheda nella finestra di dialogo mostrata sopra, verrà piazzato anche'esso in questa cartella.

Plugin and Content Manager

NOTE

TODO: Write this section