Editor degli schemi elettrici

The KiCad Team

Table of Contents

NOTE	Questo manuale è in fase di revisione per coprire l'ultima versione stabile di KiCad. Esso contiene alcune sezioni non ancora completate. Chiediamo di pazientare mentre il nostro personale tecnico volontario lavora su questo compito e diamo il benvenuto ai nuovi contributori che desiderano aiutare a migliorare la documentazione di KiCad ancora di più.

Copyright

Questo documento è coperto dal Copyright © 2010-2023 dei suoi autori come elencati in seguito. È possibile distribuirlo e/o modificarlo nei termini sia della GNU General Public License (http://www.gnu.org/licenses/gpl.html), versione 3 o successive, che della Creative Commons Attribution License (http://creativecommons.org/licenses/by/3.0/), versione 3.0 o successive.

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

Collaboratori

Jean-Pierre Charras, Fabrizio Tappero, Wayne Stambaugh, Graham Keeth

Traduzione

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

Feedback

Il progetto KiCad accoglie feedback, segnalazioni di bug e suggerimenti relativi al software o alla sua documentazione. Per ulteriori informazioni su come inviare feedback o segnalare un problema, consultare le istruzioni su https://www.kicad.org/help/report-an-issue/

Introduzione all'editor schemi elettrici di KiCad

Descrizione

L'editor degli schemi di KiCad è un software distribuito come parte della suite KiCad, e disponibile per i seguenti sistemi operativi:

- Linux
- Apple macOS
- Windows

Indipendentemente dal sistema operativo, tutti i file KiCad sono 100% compatibili da un sistema all'altro.

L'editor degli schemi è un'applicazione integrata dove tutte le funzioni di disegno, controllo, disposizione, gestione librerie e accesso al software di progettazione di circuiti stampati sono svolte all'interno dell'editor stesso.

L'editor degli schemi elettrici di KiCad è stato concepito per cooperare il programma per la progettazione di circuiti stampati della suite di KiCad. Esso può anche esportare file di netlist, che descrivono le connessioni elettriche dello schema usabili da altri software.

L'editor degli schemi include un editor di librerie di simboli, che può creare e modificare simboli e gestire librerie. Esso integra le seguenti funzioni, aggiuntive ma essenziali, necessarie in ogni moderno software di elaborazione schemi elettrici:

- Controllo regole di progettazione (ERC) per il controllo automatico di connessioni errate o sconnesse
- Esportazione di file del disegno dello schema in molti formati (Postscript, PDF, HPGL e SVG).
- Generazione della distinta materiali (tramite script Python o XSLT, che consentono di modellarla in molti formati).

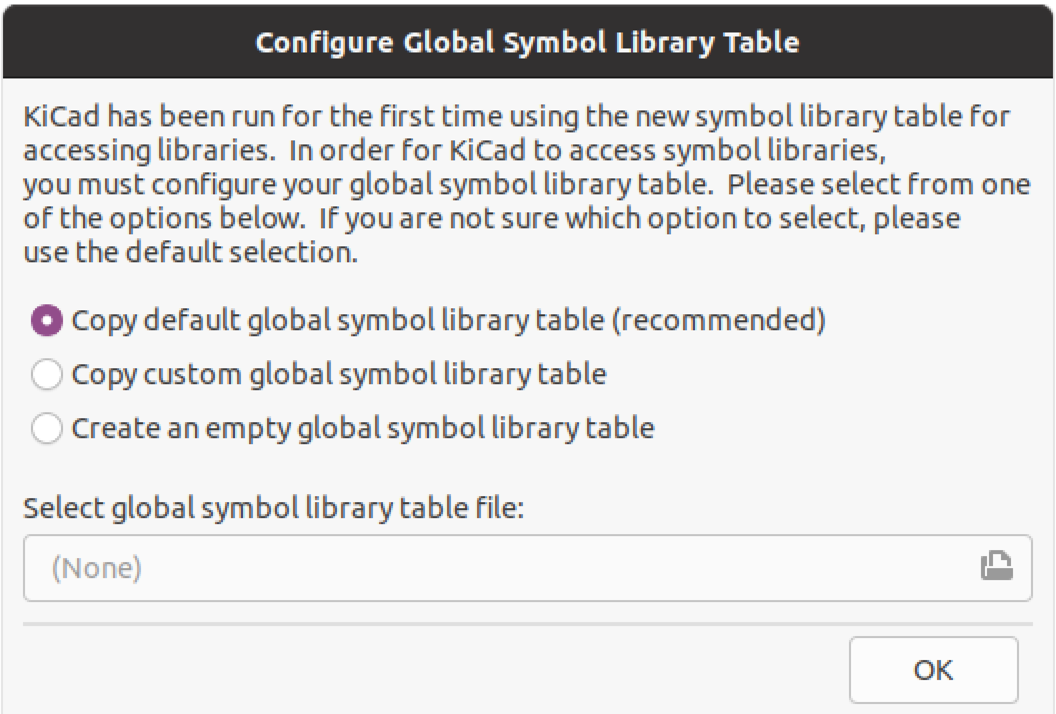
L'editor degli schemi supporta schemi multifoglio in diversi modi:

- Gerarchie piatte (i fogli degli schemi non sono esplicitamente connessi ad uno schema principale).
- Gerarchie semplici (ogni foglio di schema elettrico viene usato solo una volta).
- Gerarchie complesse (alcuni fogli di schemi elettrici sono usati più di una volta).

Gli schemi gerarchici sono descritti in dettaglio più avanti nel manuale.

Configurazione iniziale

Quando l'editor degli schemi viene eseguito per la prima volta, se il file della tabella librerie di simboli globale `sym-lib-table` non si trova nella cartella di configurazione di KiCad, KiCad chiederà come creare questo file:



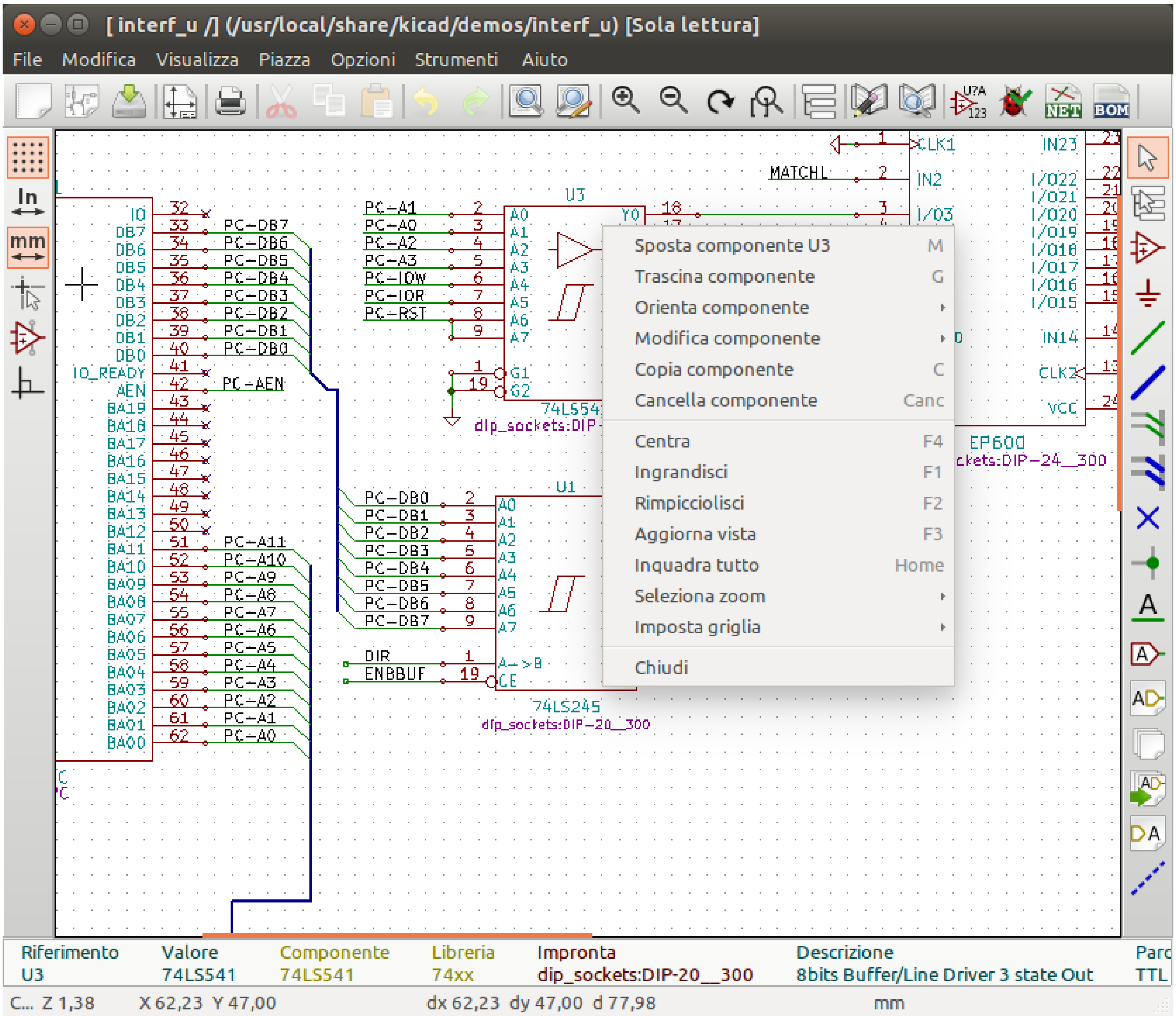
Si consiglia la prima opzione (**Copia tabella libreria simboli globale predefinita (consigliato)**). La tabella librerie impronte predefinita include molte delle librerie impronte standard che sono installate assieme a KiCad.

Se questa opzione è disabilitata, KiCad non è stato in grado di trovare la tabella della libreria di simboli globale predefinita. Questo probabilmente significa che non sono state installate le librerie di simboli standard con KiCad, o che non sono installate dove KiCad si aspetta di trovarle. Su alcuni sistemi le librerie KiCad sono installate come pacchetto separato.

- Se si ha installato le librerie di simboli standard di KiCad e vuole usarle, ma la prima opzione è disabilitata, selezionare la seconda opzione e andare al file `sym-lib-table` nella cartella in cui sono state installate le librerie di KiCad.
- Se si ha già una tabella librerie di simboli personalizzata che si vorrebbe usare, selezionare la seconda opzione e cercare il proprio file `sym-lib-table`.
- Se si vuole costruire una nuova tabella librerie di simboli da zero, selezionare la terza opzione.

La gestione delle librerie di simboli viene descritta in maggior dettaglio più avanti.

Interfaccia utente dell’editor degli schemi elettrici



L’interfaccia utente principale dell’editor degli schemi elettrici è mostrata sopra. Il centro contiene l’area di modifica principale, che è circondata da:

- Barre degli strumenti principali (gestione dei file, strumenti di zoom, strumenti di modifica)
- Barra degli strumenti di sinistra (opzioni di visualizzazione)
- Pannello dei messaggi e barra di stato in basso
- Pannello di destra (strumenti di disegno e progettazione)

Navigazione nell’area di lavoro

L’area di lavoro mostra lo schema in fase di progettazione. È possibile traslare la vista e fare lo zoom su diverse parti dello schema e aprire qualsiasi foglio dello schema nel progetto.

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 every item in the schematic (not including the drawing sheet). For instance, if there are items placed outside of the drawing sheet, they will be visible after zooming to objects.
- 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.

Le coordinate relative possono essere azzerate premendo `Spazio`. È utile per effettuare misure tra due punti o per allineare oggetti.

Comandi da tastiera

The `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 `Cmd` key in place of `Ctrl`, and the `Option` key in place of `Alt`.

Molte azioni non hanno tasti comando assegnati per impostazione predefinita, ma i tasti comando possono essere assegnati o ridefiniti utilizzando l'editor di tasti comando (**Preferenze** → **Preferenze...** → **Tasti comando**).

NOTE	Many of the actions available through hotkeys are also available in context menus. To access the context menu, right-click in the editing canvas. Different actions will be available depending on what is selected or what tool is active.

I comandi da tastiera sono memorizzati nel file `user.hotkeys` nella cartella di configurazione di KiCad. La posizione cambia a seconda della piattaforma:

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

KiCad può importare le impostazioni dei tasti comando da un file `user.hotkeys` utilizzando il pulsante **Importa tasti comando** nell'editor di tasti comando.

Operazioni e selezione col mouse

Selecting items in the editing canvas is done with the left mouse button. Single-clicking on an object will select it. Clicking 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.

The selection action can be modified by holding modifier keys while clicking or dragging. The following modifier keys apply when clicking to select single items:

Modifier Keys (Windows)	**Modifier Keys (Linux)**	**Modifier Keys (macOS)**	**Selection Effect**
Ctrl	Ctrl	Cmd	Toggle selection.
Shift	Shift	Shift	Add the item to the existing selection.
Ctrl + Shift	Ctrl + Shift	Cmd + Shift	Remove the item from the existing selection.
long click	long click or Alt	long click or Option	Clarify selection from a pop-up menu.

The following modifier keys apply when dragging to perform a box selection:

Modifier Keys (Windows)	**Modifier Keys (Linux)**	**Modifier Keys (macOS)**	**Selection Effect**
Ctrl	Ctrl	Cmd	Toggle selection.
Shift	Shift	Shift	Add item(s) to the existing selection.
Ctrl + Shift	Ctrl + Shift	Cmd + Shift	Remove item(s) from the existing selection.

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.

Pressing Esc will always cancel the current tool or operation and return to the selection tool. Pressing Esc while the selection tool is active will clear the current selection.

Controlli di visualizzazione della barra strumenti a sinistra

La barra strumenti a sinistra fornisce le opzioni di visualizzazione degli elementi nell'editor degli schemi elettrici.

	Turns grid display on/off. **Note:** by default, hiding the grid will disable grid snapping. This behavior can be changed in the Display Options section of Preferences.
  	Display/entry of coordinates and dimensions in inches, mils, or millimeters.
	Switches between full-screen and small editing cursor (crosshairs).
	Turns invisible pin display on/off.
  	Switches between free angle, 90 degree mode, and 45 degree mode for placement of new wires, buses, and graphical lines.
	Opens and closes the docked hierarchy navigator pane.

Creazione e modifica di schemi elettrici

Introduzione

Uno schema elettrico progettato con KiCad è più di una semplice rappresentazione grafica di un dispositivo elettronico. Esso normalmente è il punto di ingresso di una catena di sviluppo che permette:

- Il controllo di validità rispetto ad una serie di regole (Controllo Regole Elettriche (ERC)) per il rilevamento di errori e omissioni.
- Generazione automatica della DIBA.
- La generazione di una netlist per software di simulazione tipo SPICE.
- Definizione di un circuito per il trasferimento delle informazioni nella progettazione del circuito stampato.

Uno schema elettrico consiste principalmente di simboli, fili, etichette, giunzioni, bus e simboli di potenza. Per chiarezza, negli schemi elettrici, è possibile inserire elementi puramente grafici come elementi bus, commenti, e polilinee.

I simboli vengono aggiunti allo schema dalle librerie di simboli. Dopo aver creato lo schema, l'insieme di connessioni e impronte viene importato nell'editor dei circuiti stampati per progettare la scheda.

Gli schemi possono essere contenuti in un unico foglio o suddivisi in più fogli. In KiCad, gli schemi a più fogli sono organizzati gerarchicamente, con un foglio principale e uno o più fogli secondari. Ogni foglio è il proprio file `.kicad_sch` ed è esso stesso uno schema completo di KiCad. L'utilizzo degli schemi gerarchici è descritto nel capitolo schemi gerarchici.

Operazioni di modifica degli schemi elettrici

Schematic editing tools are located in the right toolbar. When a tool is activated, it stays active until a different tool is selected or the tool is canceled with the Esc key. The selection tool is always activated when any other tool is canceled.

	Selection tool (the default tool)
	Highlight a net by marking its wires and net labels with a different color. If the PCB Editor is also open then copper corresponding to the selected net will be highlighted as well. Net highlighting can be cleared by clicking with the highlight tool in an empty space, or by using the Clear Net Highlighting hotkey (~).
	Display the symbol selector dialog to place a new symbol.
	Display the power symbol selector dialog to place a new power symbol.
	Draw a wire.
	Draw a bus.

	Draw wire-to-bus entry points. These elements are only graphical and do not create a connection, thus they should not be used to connect wires together.
	Place a "No Connect" flag. These flags should be placed on symbol pins which are meant to be left unconnected. "No connect" flags indicate to the Electrical Rule Checker that the pin is intentionally unconnected and not an error.
	Place a junction. This connects two crossing wires or a wire and a pin, which can sometimes be ambiguous without a junction (i.e. if a wire end or a pin is not directly connected to another wire end).
	Place a local label. Local labels connect items located **in the same sheet**. For connections between two different sheets, use global or hierarchical labels.
	Place a net class directive label.
	Place a global label. All global labels with the same name are connected, even when located on different sheets.
	Place a hierarchical label. Hierarchical labels are used to create a connection between a subsheet and the sheet's parent sheet. See the Hierarchical Schematics section for more information about hierarchical labels, sheets, and pins.
	Place a hierarchical subsheet. You must specify the file name for this subsheet.
	Import a hierarchical pin from a subsheet. This command can be executed only on hierarchical subsheets. It will create hierarchical pins corresponding to hierarchical labels placed in the target subsheet.
	Place a text comment.
	Place a text box.
	Draw a rectangle.
	Draw a circle.
	Draw an arc.
	Draw lines. **Note:** Lines are graphical objects and are not the same as wires placed with the Wire tool. They do not connect anything.
	Place a bitmap image.
	Delete clicked items.

Griglie

Nell'editor degli schemi il puntatore si sposta sempre sopra una griglia. La griglia può essere personalizzata:

I colori possono essere modificati tramite la scheda **Colori** presente nella finestra di dialogo delle **Preferenze** (menu **Preferenze → Opzioni generali**).

- La visibilità può essere accesa/spenta usando il pulsante corrispondente nella barra strumenti di sinistra.

La dimensione predefinita della griglia è 50 mils (0.050") o 1,27 millimetri.

Questa è la griglia raccomandata per piazzare simboli e fili in uno schema elettrico, e per piazzare piedini durante la progettazione di un simbolo nell'editor dei simboli.

NOTE	Wires connect with other wires or pins only if their ends coincide **exactly**. Therefore it is very important to keep symbol pins and wires aligned to the grid. It is recommended to always use a 50 mil grid when placing symbols and drawing wires because the KiCad standard symbol library and all libraries that follow its style also use a 50 mil grid. **Using a grid size other than 50 mil will result in schematics without proper connectivity!**

È possibile usare anche griglie più piccole, ma queste servono solo per il testo e la grafica dei simboli e non è consigliato per posizionarvi pin e fili.

NOTE	Symbols, wires, and other elements that are not aligned to the grid can be snapped back to the grid by selecting them, right clicking, and clicking **Align Elements to Grid**.

Magnetismo

Schematic elements such as symbols, wires, text, and graphic lines are snapped to the grid when moving, dragging, and drawing them. Additionally, the wire tool snaps to pins even when grid snapping is disabled. Both grid and pin snapping can be disabled while moving the mouse by using the modifier keys in the table below.

NOTE	Su tastiere Apple, usare il tasto `Cmd` al posto di `Ctrl`.

Modifier Key	Effect
`Ctrl`	Disable grid snapping.
`Shift`	Disable snapping wires to pins.

Modifica proprietà dell'oggetto

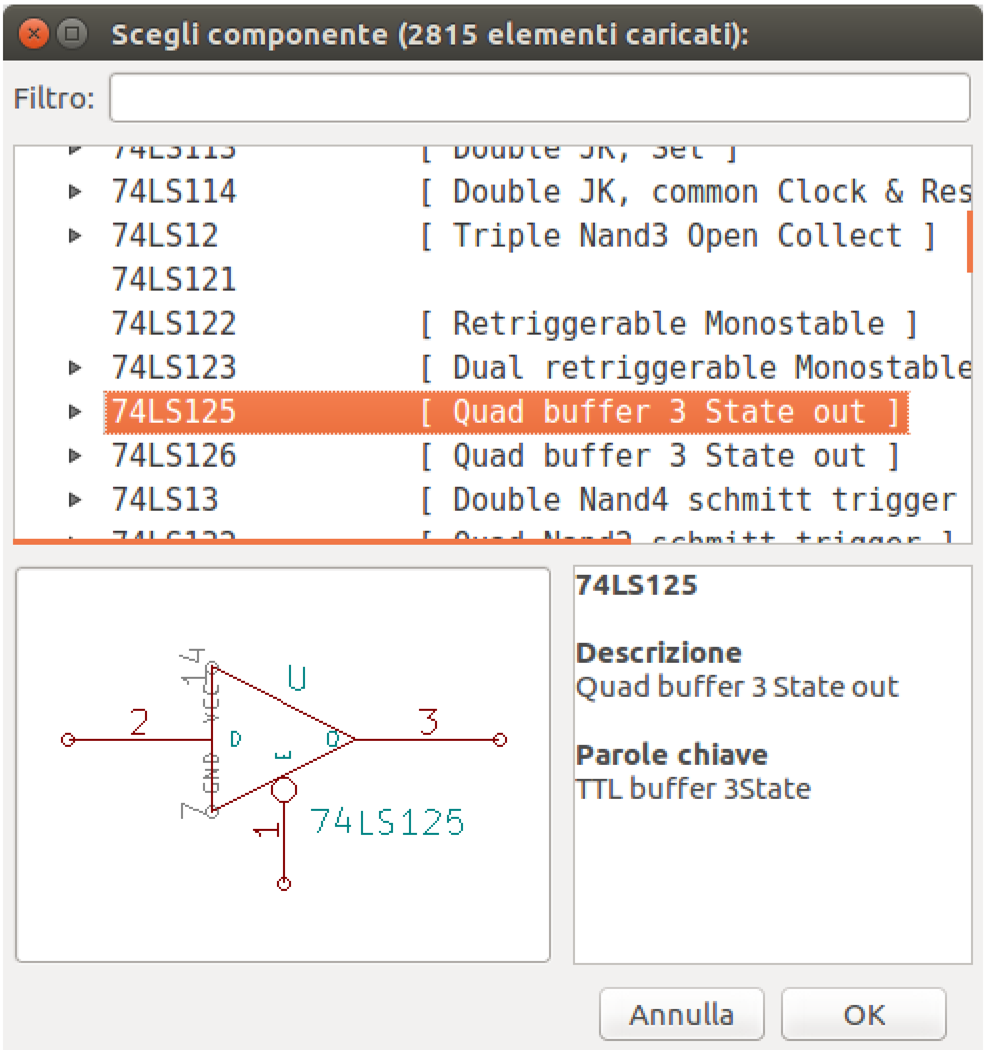
All objects have properties that are editable in a dialog. Use the hotkey `E` or select Properties from the right-click context menu to edit the properties of selected item(s). You can only open the properties dialog if all the items you have selected are of the same type. To edit the properties of different types of items at one time, see the section below on bulk editing tools.

In properties dialogs, any field that contains a numeric value can also accept a basic math expression that results in a numeric value. For example, a dimension may be entered as `2 * 2mm`, resulting in a value of `4mm`. Basic arithmetic operators as well as parentheses for defining order of operations are supported.

Lavorare con i simboli

Piazzamento simboli

Per piazzare un simbolo nello schema elettrico si può usare il pulsante  o il tasto `A`. Appare la finestra di dialogo Scegli simboli che consente di scegliere il simbolo da aggiungere. I simboli sono raggruppati per libreria.



By default, only the symbol/library name and description columns are shown. Additional columns can be added by right-clicking the column header and selecting **Select Columns**.

La finestra di dialogo di scelta del simbolo filtrerà i simboli per nome, parolechiave, descrizione e tutti i campi simbolo aggiuntivi, a seconda di quanto si inserirà nel campo di ricerca.

Alcuni filtri avanzati sono disponibili:

- **Wildcards:** `*` matches any number of any characters, including none, and `?` matches any single character.
- **Paia di chiavi-valore:** se la descrizione di un componente di libreria o parola chiave contiene un marcatore nel formato "chiave:123", si può verificarne la corrispondenza relativa battendo "chiave>123" (maggiore di), "chiave<123" (minore di), ecc. I numeri possono includere uno dei seguenti suffissi indipendenti da maiuscole o minuscole:

p	n	u	m	k	meg	g	t
10^{-12}	10^{-9}	10^{-6}	10^{-3}	10^{3}	10^{6}	10^{9}	10^{12}

ki	mi	gi	ti
2^{10}	2^{20}	2^{30}	2^{40}

- **Espressioni regolari:** se si ha familiarità con le espressioni regolari, si possono usare anch'esse. Il tipo di espressione regolare usato è di stile espressione regolare avanzato dei wxWidgets, che è simile alle espressioni regolari Perl.

Se il simbolo specifica un'impronta predefinita, questa verrà visualizzata in anteprima in basso a destra. Se il simbolo include filtri impronta, è possibile selezionare impronte alternative che soddisfano i filtri impronta nel menu impronta a discesa a destra.

Dopo aver selezionato un simbolo da posizionare, il simbolo verrà attaccato al cursore. Facendo clic con il pulsante sinistro del mouse sulla posizione desiderata nello schema, il simbolo viene inserito nello schema. Prima di posizionare il simbolo nello schema elettrico, è possibile ruotarlo, renderlo speculare e modificarne i campi, usando i tasti comando o il menu contestuale del tasto destro. Queste azioni possono essere eseguite anche dopo il posizionamento.

Se l'opzione **Posiziona copie ripetute** è selezionata, dopo aver posizionato un simbolo, KiCad inizierà a posizionare un'altra copia del simbolo. Questo processo continua finché l'utente non preme `Esc`.

Per i simboli con più unità, se l'opzione **Posiziona tutte le unità** è selezionata, dopo aver posizionato il simbolo KiCad inizierà a posizionare l'unità successiva nel simbolo. Ciò continuerà fino a quando l'ultima unità sarà stata posizionata o sino a quando l'utente premerà `Esc`.

Piazzamento simboli di potenza

Un simbolo di potenza è un simbolo che rappresenta una connessione a un collegamento di potenza (N.d.T. come un'alimentazione). I simboli sono raggruppati nella libreria `power`, quindi possono essere posizionati utilizzando il selettore simboli. Tuttavia, poiché i posizionamenti di simboli potenza sono frequenti, è disponibile lo strumento . Questo strumento è simile, tranne per il fatto che la ricerca viene eseguita direttamente nella libreria `power` e in qualsiasi altra libreria che contenga simboli di potenza.

Spostamento simboli

I simboli possono essere spostati utilizzando gli strumenti Sposta (`M`) o Trascina (`G`). Questi strumenti agiscono sul simbolo selezionato, oppure se nessun simbolo fosse stato selezionato, agiscono sul simbolo sotto il cursore.

Lo strumento **Sposta** sposta il simbolo stesso senza mantenere connessioni cablate ai piedini del simbolo.

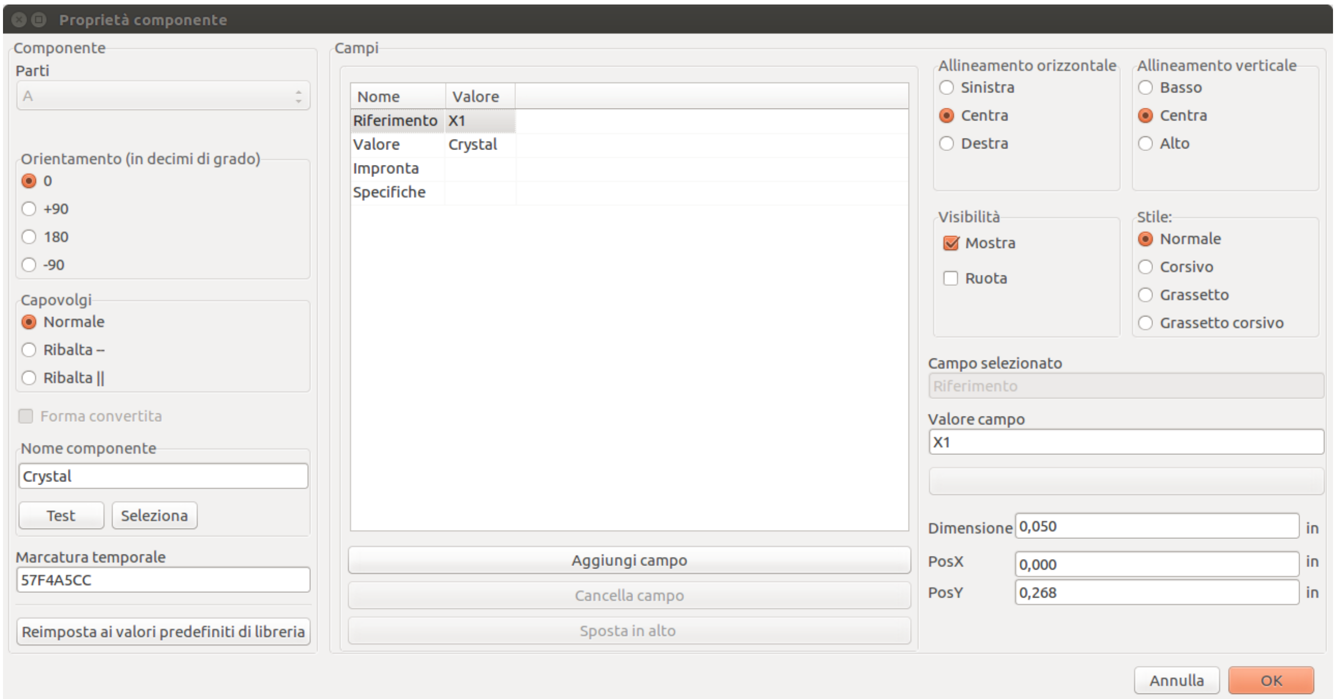
Lo strumento **Trascina** sposta il simbolo senza interrompere le connessioni cablate ai suoi piedini, e quindi sposta anche i fili collegati.

You can also Drag symbols by clicking and dragging them with the mouse, depending on the **Left button drag gesture** setting in the **Mouse and Touchpad** section of Preferences.< Si può anche trascinare i simboli facendo clic e trascinandoli con il mouse, a seconda dell'impostazione **Gesture trascinamento del pulsante sinistro** nella sezione **Mouse e touchpad** delle Preferenze.

Symbols can also be rotated (R) or mirrored in the X (X) or Y (Y) directions.

Modifica delle proprietà del simbolo

A symbol's fields can be edited in the symbol's Properties window. Open the Symbol Properties window for a symbol with the E hotkey or by double-clicking on the symbol.



The Symbol Properties window displays all the fields of a symbol in a table. New fields can be added, and existing fields can be deleted, edited, reordered, moved, or resized.

Il nome e il valore di ciascun campo possono essere visibili o nascosti e sono disponibili diverse opzioni di formattazione: allineamento orizzontale e verticale, orientamento, posizione, carattere, colore del testo, dimensione del testo ed enfasi in grassetto/corsivo. Il posizionamento automatico del campo può anche essere abilitato in base al campo. La posizione visualizzata è sempre indicata per un simbolo visualizzato normalmente (nessuna rotazione o specularità) ed è relativa al punto di ancoraggio del simbolo.

NOTE	Le opzioni di formattazione per i campi del simbolo possono essere visualizzate o nascoste facendo clic con il pulsante destro del mouse sulla riga dell'intestazione della tabella campi simbolo abilitando o disabilitando le colonne desiderate. Non tutte le colonne sono visibili per impostazione predefinita.

The **Update Symbol from Library…** button is used to update the schematic's copy of the symbol to match the copy in the library. The **Change Symbol…** button is used to swap the current symbol to a different symbol in the library.

Edit Symbol… opens the Symbol Editor to edit the copy of the symbol in the schematic. Note that the original symbol in the library will not be modified. The **Edit Library Symbol…** button opens the Symbol Editor to edit the original symbol in the library. In this case, the symbol in the schematic will not be modified until the user clicks the **Update Symbol from Library…** button.

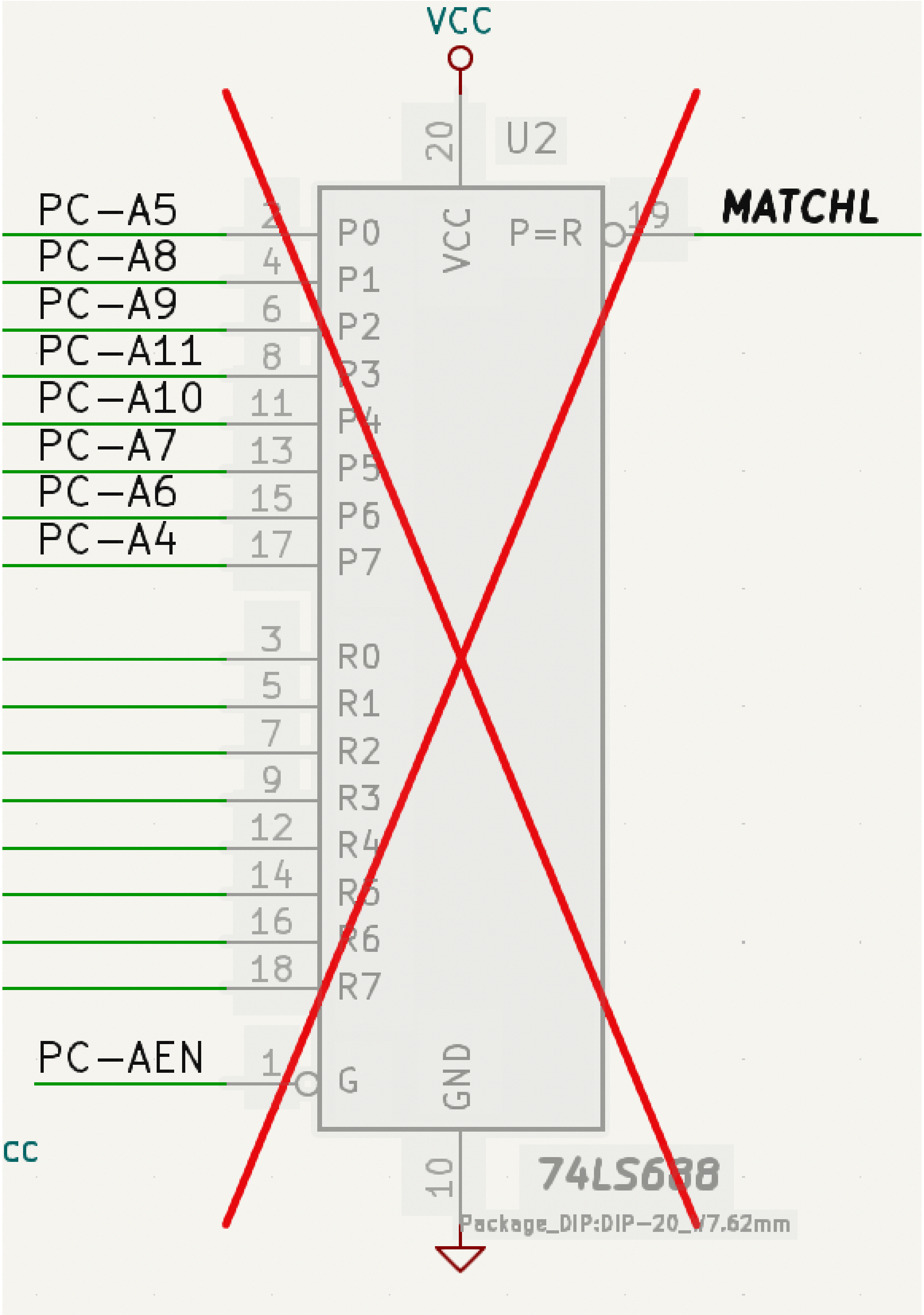
Symbols have several attributes that affect how the symbols are treated by other parts of KiCad.

Exclude from simulation prevents the symbol from being included in SPICE simulations.

Exclude from bill of materials prevents the component from being included in BOM exports.

Exclude from board means that the symbol is schematic-only, and a corresponding footprint will not be added to the PCB.

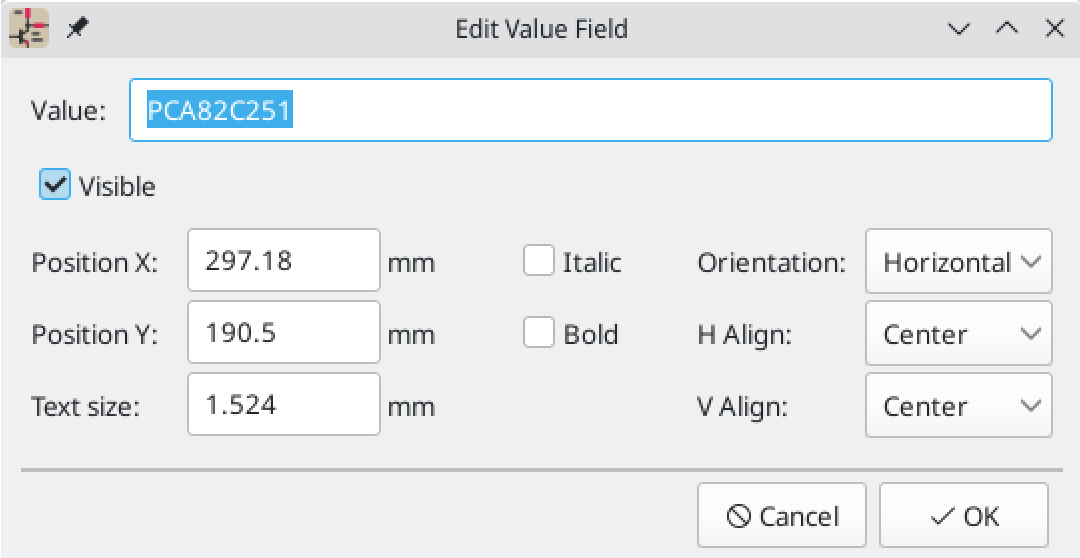
Do not populate means that the component should not be attached to the PCB, although a corresponding footprint should still be added to the board. DNP symbols appear desaturated and with a red "X" over them in the schematic, as shown below.



Modifica individuale dei campi del simbolo

An individual symbol text field can be edited directly with the E hotkey (with a field selected instead of a symbol) or by double-clicking on the field.

Some symbol fields have their own hotkey to edit them directly. With the symbol selected, the Reference, Value, and Footprint fields can be edited with the U, V, or F hotkeys, respectively.

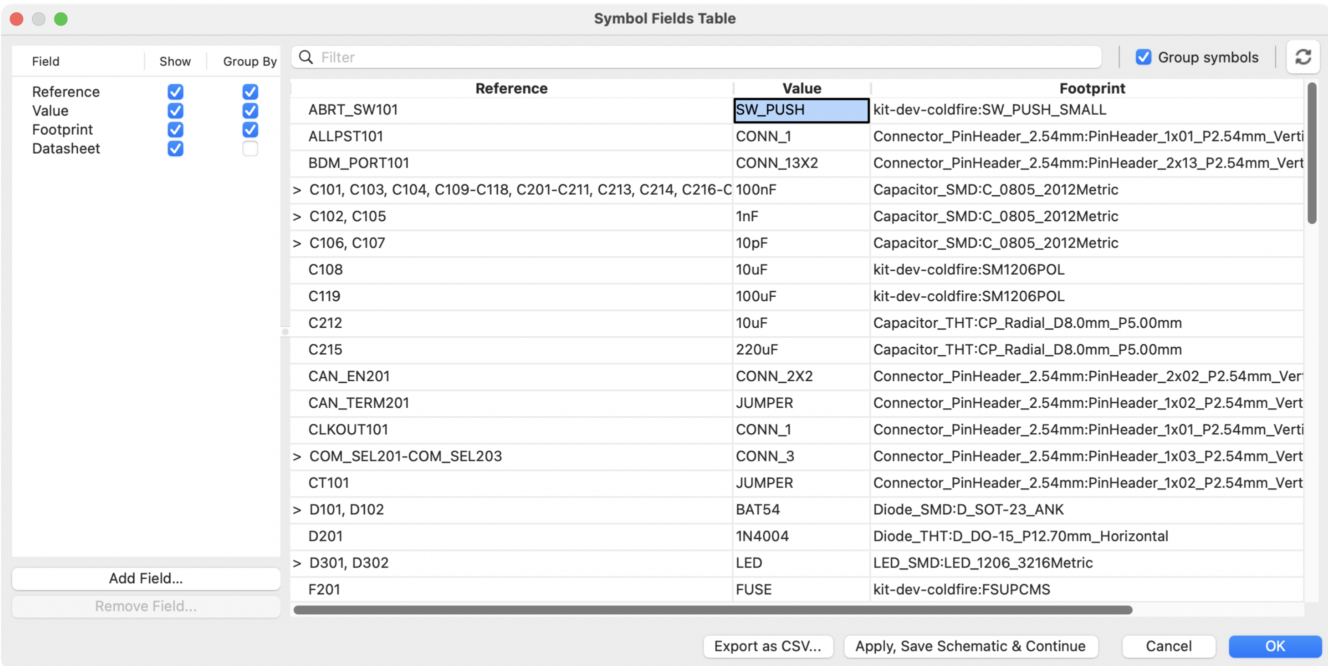


The options in this dialog are the same as those in the full Symbol Properties dialog, but are specific to a single field.

Symbol fields can be automatically moved to an appropriate location with the Autoplace Fields action (select a symbol and press `O`). Field autoplacement is configurable in the Schematic Editor's Editing Options, including a setting to always autoplace fields.

Tabella campi del simbolo

The Symbol Fields Table allows you to view and modify field values for all symbols in a spreadsheet interface. You can open the Symbol Fields Table with the button.



Cells are navigated with the arrow keys, or with `Tab` / `Shift`+`Tab` to move right / left and `Enter` / `Shift`+`Enter` to move down / up, respectively.

A range of cells can be selected by clicking and dragging. The whole range of selected cells will be copied (`Ctrl`+`C`) or pasted into (`Ctrl`+`V`) on a copy or paste action. Copying a range of cells from the table can be useful for creating a BOM. More details of copying and pasting cells are described below.

Any symbol field can be shown or hidden using the **Show** checkboxes on the left, or by right-clicking on the header of the table. New symbol fields can be added using the **Add Field...** button.

Similar symbols can optionally be grouped by any symbol field using the **Group By** checkboxes. Grouped symbols are shown in a single row in the table. The grouped row can be expanded to show the individual symbols by clicking the arrow at the left of the row. The **Group Symbols** checkbox enables or disables symbol grouping, and the button recalculates groupings.

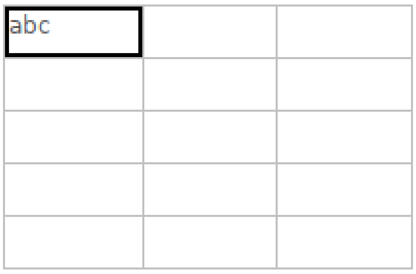
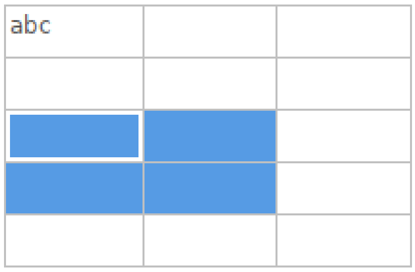
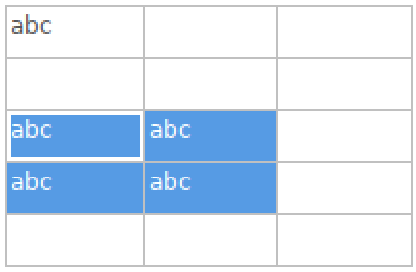
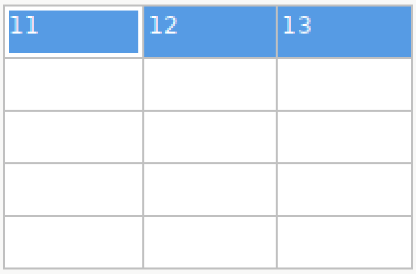
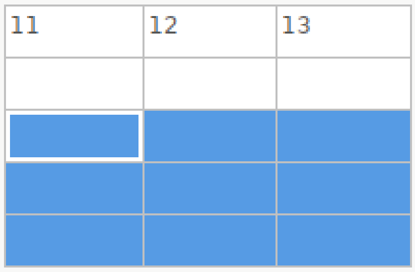
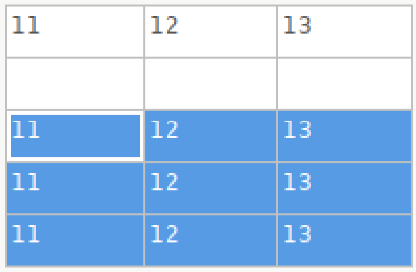
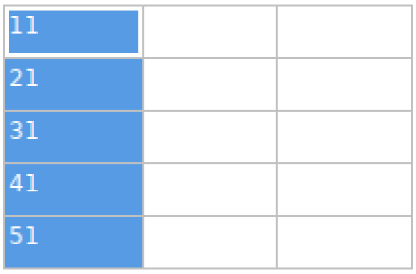
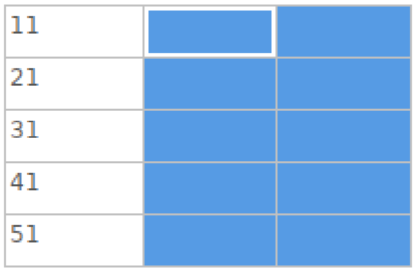
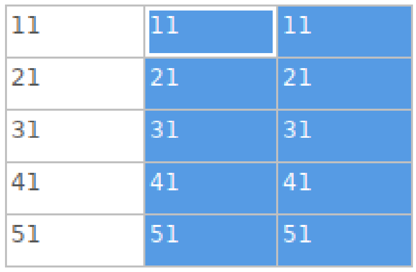
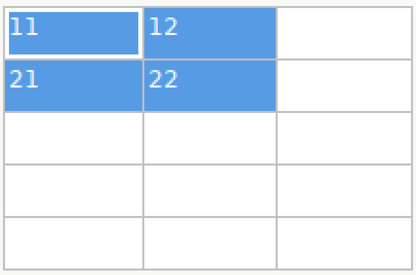
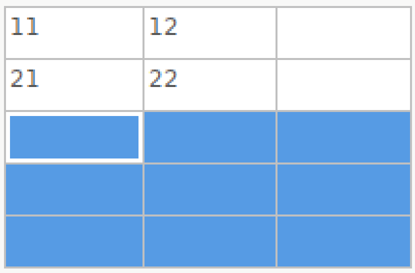
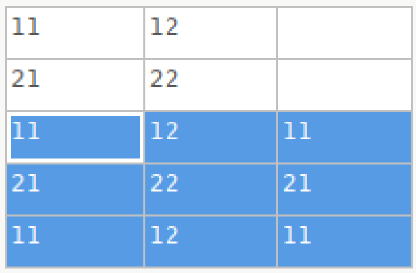
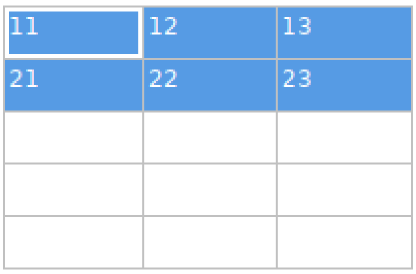
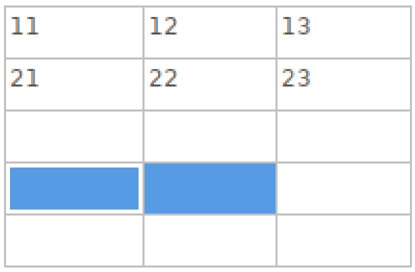
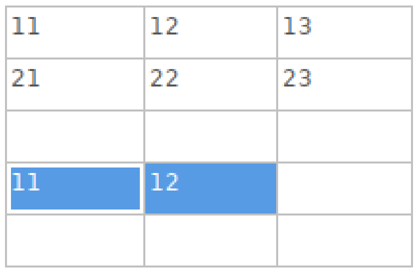
Symbols can be filtered using the **Filter** textbox at the top. The filter supports wildcards: `*` matches any number of any characters, including none, and `?` matches any single character.

You can use the **Export as CSV...** button to save the symbol fields to an external file. This can be used as a simple BOM generation tool, although the BOM tool provides better control over the generated output.

Trucchi per semplificare lo riempimento dei campi

Esistono diversi metodi speciali di copia/incolla nel foglio di calcolo per incollare i valori in regioni più grandi, comprese le celle incollate con incremento automatico. Queste funzionalità possono essere utili quando si incollano valori condivisi in più simboli.

Questi metodi sono illustrati sotto.

1. Copia (Ctrl + C)	2. Selezione celle obiettivo	3. Incolla (Ctrl + V)
		
		
		
		
		

NOTE Queste tecniche sono disponibili anche in altre finestre di dialogo con elementi di controllo a griglia.

Reference Designators and Symbol Annotation

Reference designators are unique identifiers for components in a design. They are often printed on a PCB and in assembly diagrams, and allow you to match symbols in a schematic to the corresponding components on a board.

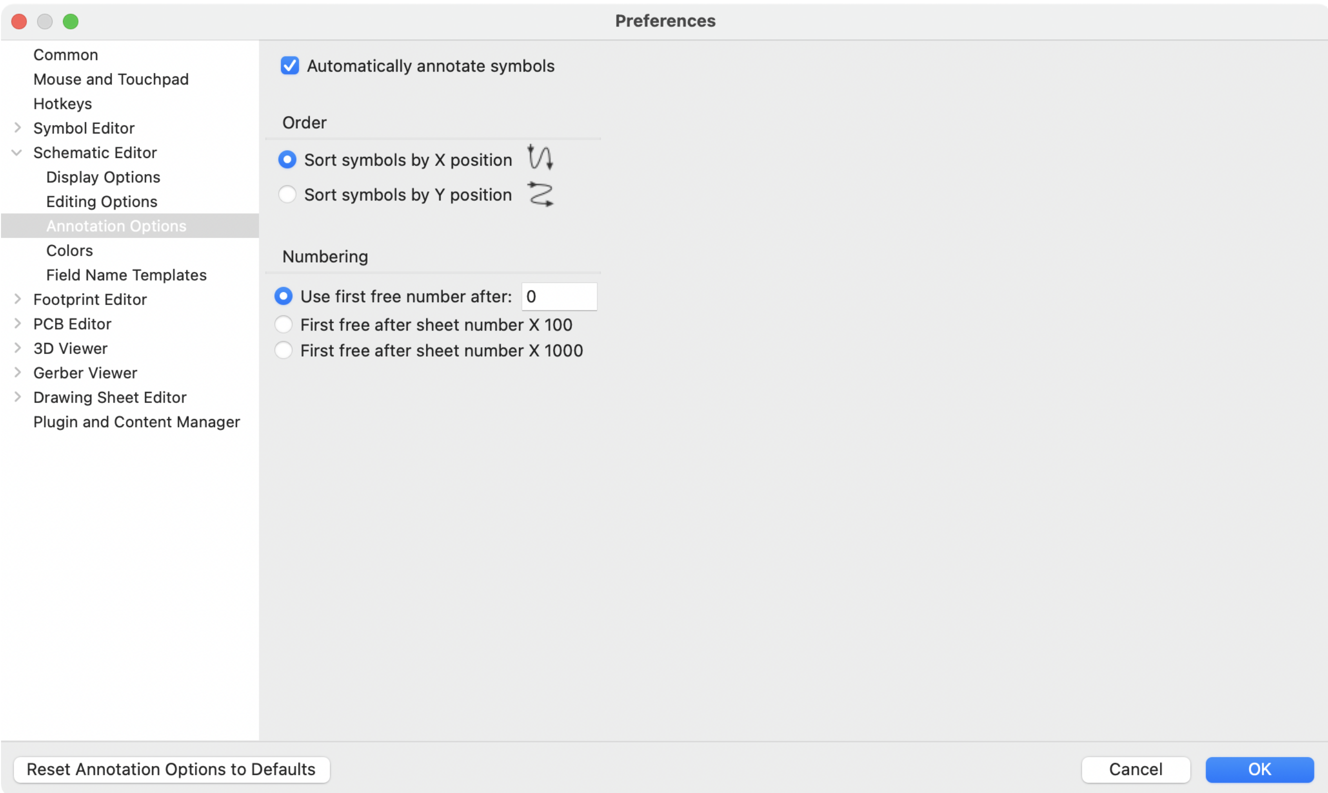
In KiCad, reference designators consist of a letter indicating the type of component (`R` for resistor, `C` for capacitor, `U` for IC, etc.) followed by a number. If the symbol has multiple units then the reference designator will also have a trailing letter indicating the unit. Symbols that don't have a reference designator set have a `?` character instead of the number. Reference designators must be unique.

Reference designators can be automatically set when symbols are added to the schematic, and you can set or reset reference designators yourself by manually editing an individual symbol's reference designator field or in bulk using the Annotation tool.

> **NOTE** The process of setting a symbol's reference designator is called **annotation**.

Auto annotazione

When auto-annotation is enabled, symbols will be automatically annotated when they are added to the schematic. You can enable auto-annotation by checking the **Automatically annotate symbols** checkbox in the **Schematic Editor → Annotation Options** pane in **Preferences**. Auto-annotation can also be toggled using the button in the left toolbar.



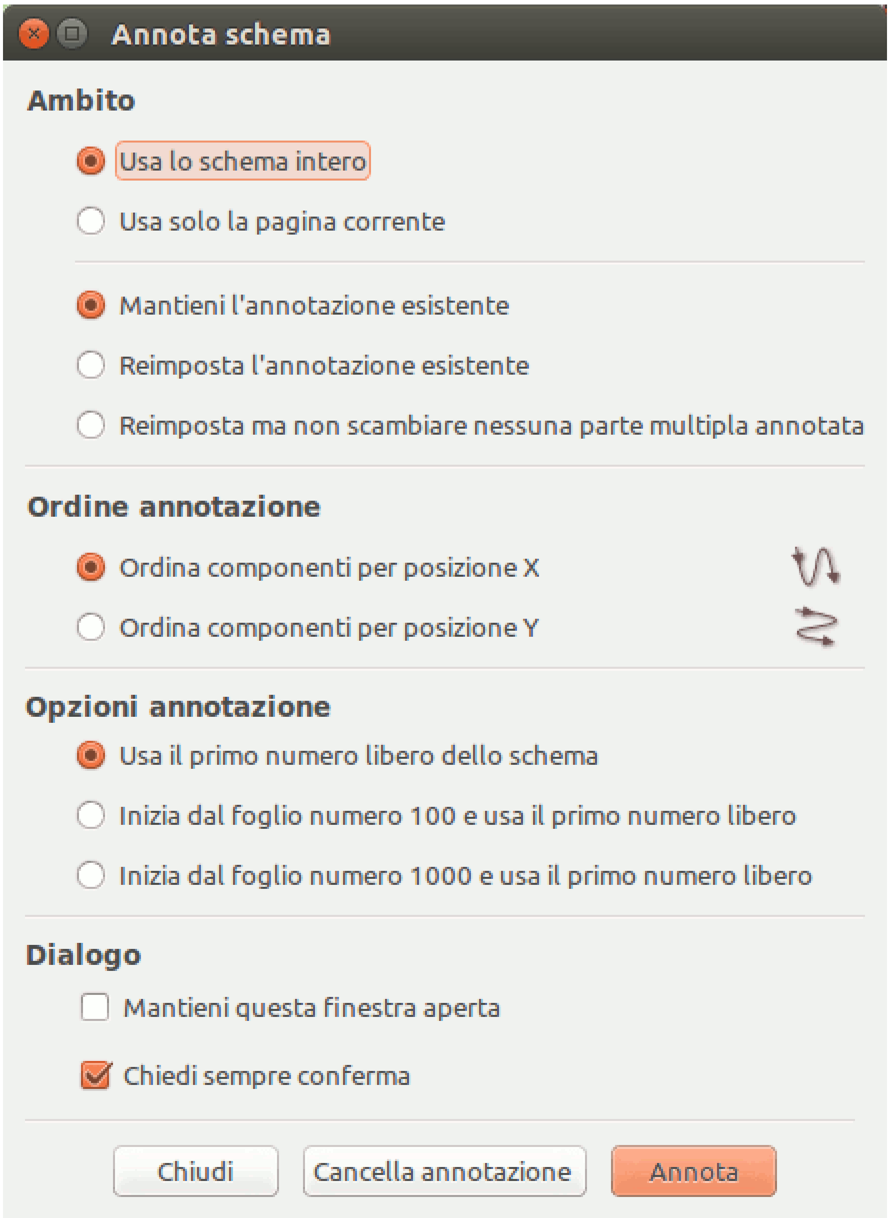
When multiple symbols are added simultaneously, they are annotated according to the **Order** setting, sorted by either X or Y position.

The **Numbering** option sets the starting number for new reference designators. This can be the lowest available number, or a number based on the sheet number.

For more information about annotation options, see the documentation for the Annotation tool.

Strumento di annotazione

Lo strumento di annotazione assegna automaticamente i riferimenti a simboli nello schema. Per eseguire lo strumento di annotazione fare clic sull'icona nella barra strumenti in alto.



The tool provides several options to control how symbols are annotated.

Scope: Selects whether annotation is applied to the entire schematic, to only the current sheet, or to only the selected symbols. If the **Recurse into subsheets** option is selected, symbols in subsheets of the selected scope will be reannotated; otherwise symbols in subsheets will not be reannotated. For example, if **Recurse into subsheets** and **Selection only** selected, symbols in any selected subsheets will be reannotated.

Options: Selects whether annotation should apply to all symbols and reset *existing reference designators, or apply only to unannotated symbols.

Order: Chooses the direction of numbering. If symbols are sorted by X position, all symbols on the left side of a schematic sheet will be lower numbered than symbols on the right side of the sheet. If symbols are sorted by Y position, all symbols on the top of a sheet will be lower numbered than symbols at the bottom of the sheet.

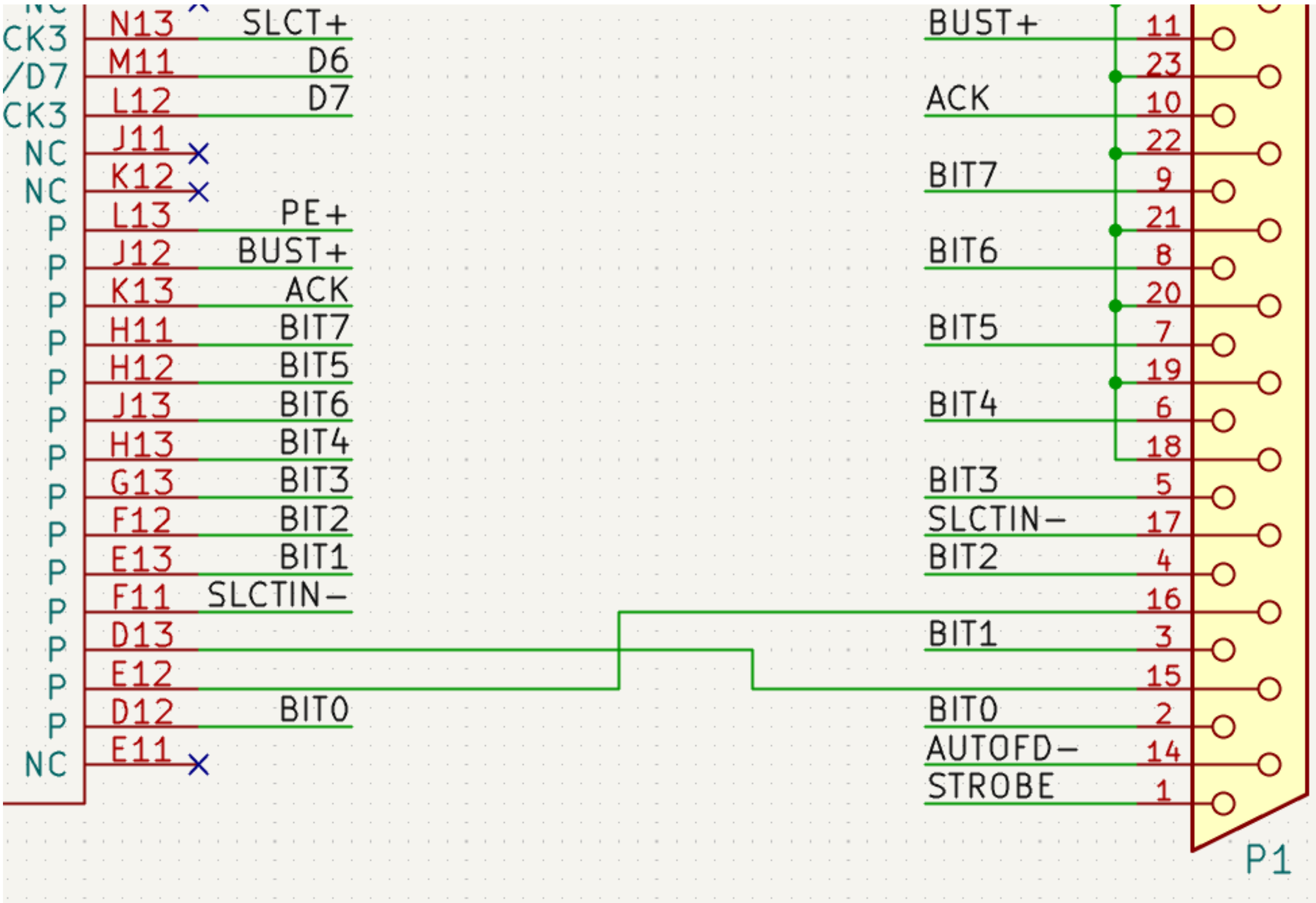
Numbering: Selects the starting point for numbering reference designators. The lowest unused number above the starting point is picked for each reference designator. The starting point can be an arbitrary number (typically zero), or it can be the sheet number multiplied by 100 or 1000 so that each part's reference designator corresponds to the schematic page it is on.

The **Clear Annotation** button clears all reference designators in the selected scope.

Annotation messages can be filtered with the checkboxes at the bottom or saved to a report using the **Save...** button.

Connessioni elettriche

There are two primary ways to establish connections: wires and labels. Wires make direct connections, while labels connect to other labels with the same name. Both wires and labels are shown in the schematic below.



Connections can also be made with buses and with implicit connections via hidden power pins.

This section will also discuss two special types of symbols that can be added with the "Power symbol" button on the right toolbar:

- **Simboli di potenza**: simboli per connettere una linea di potenza o di massa.
- **PWR_FLAG**: a specific symbol for indicating that a net is powered when it is not connected to a power output pin (for example, a power net that is supplied by an off-board connector).

Fili

I fili vengono utilizzati per stabilire direttamente collegamenti elettrici tra due punti. Per stabilire una connessione, un segmento di filo deve essere collegato per la sua estremità a un altro segmento o ad un pin. Solo le estremità dei fili creano connessioni; se un filo attraversa il centro di un altro filo, non verrà effettuata una connessione.

Unconnected wire ends have a small square that indicates the connection point. The square disappears when a connection is made to the wire end. Unconnected pins have a circle, which also disappears when a connection is made.

NOTE	Wires connect with other wires or pins only if their ends coincide exactly. Therefore it is important to keep symbol pins and wires aligned to the grid. It is recommended to always use a 50 mil grid when placing symbols and drawing wires because the KiCad standard symbol library and all libraries that follow its style also use a 50 mil grid.

NOTE	Symbols, wires, and other elements that are not aligned to the grid can be snapped back to the grid by selecting them, right clicking, and selecting **Align Elements to Grid**.

Disegno e modifica fili

To begin connecting elements with wire, use the Wire tool in the right toolbar (`W`). Wires can also be automatically started by clicking on an unconnected symbol pin or wire end.

You can restrict wires to 90 degree angles using the button in the left toolbar, or to 45 degree angles with the button. The button allows you to place wires at any angle. You can cycle through these modes using `Shift`+`Space`, or select the desired mode in **Preferences** → **Schematic Editor** → **Editing Options**. These modes affect graphic lines in addition to wires.

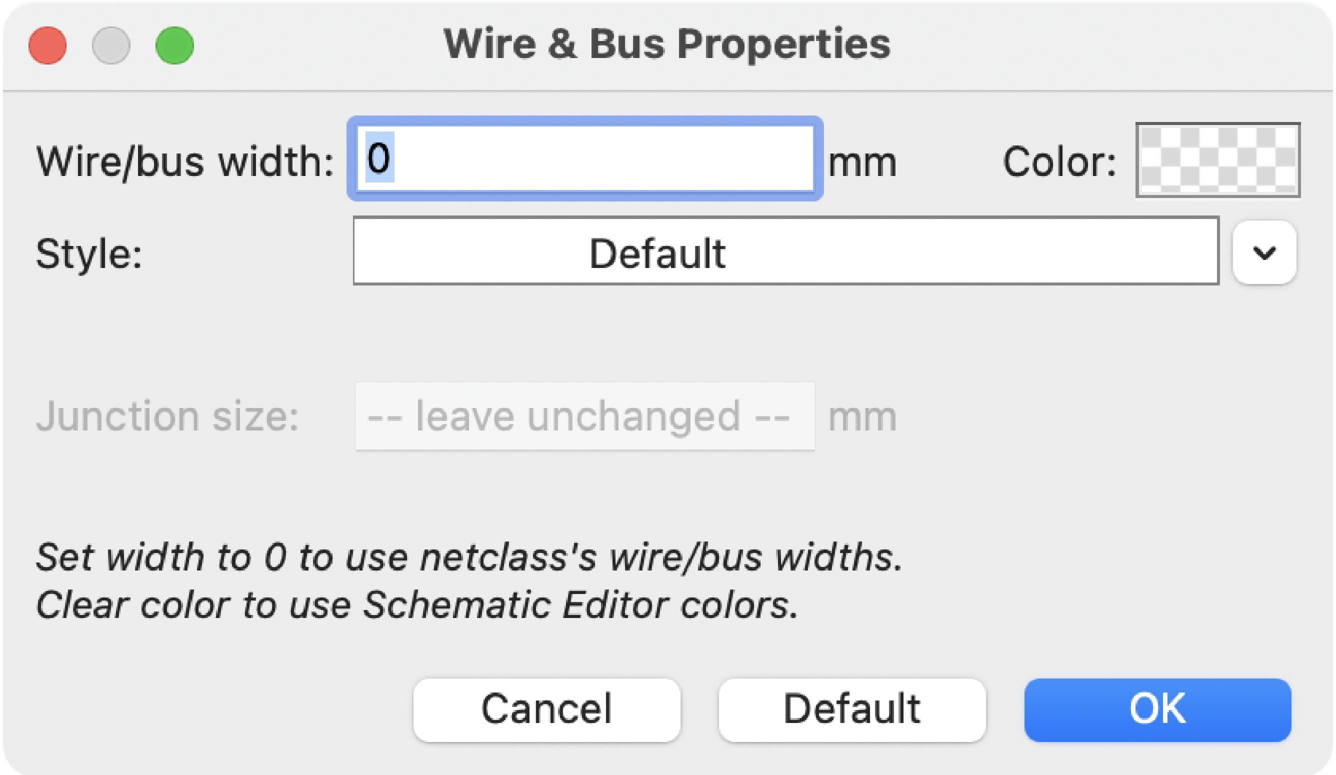
As in the PCB editor, the `/` hotkey switches wire posture.

Wires can be moved and edited using the Move (`M`) or Drag (`G`) tools. As with symbols, the **Move** tool moves only the selected segment, without maintaining existing connections to other segments. The **Drag** tool maintains existing connections.

You can select connected wires using the **Select Connection** tool (`Alt`+`4`). This tool selects all connected wire segments until it reaches a junction, starting with the selected segment or the segment under the cursor. Using the tool again expands the existing selection to the next junction.

You can break a wire segment into two pieces by right-clicking a wire and selecting **Slice**. The segment will be separated at the current mouse position. You can also separate a wire segment from the adjacent segments by right-clicking the segment and selecting **Break**.

Normally the line style of a wire follows the net's netclass settings (nets are in the `Default` netclass if no other netclass is specified). However, the line style for the selected wire segments can be overridden in the wire's properties dialog (`E` when a wire segment is selected). The wire's width, color, and line style (solid, dashed, dotted, etc.) can be set. Setting the width to `0`, clearing the color, and using the `Default` line style uses the default width, color, and style, respectively, from the netclass settings. If a wire junction is included in the selection, the junction size can also be edited here.

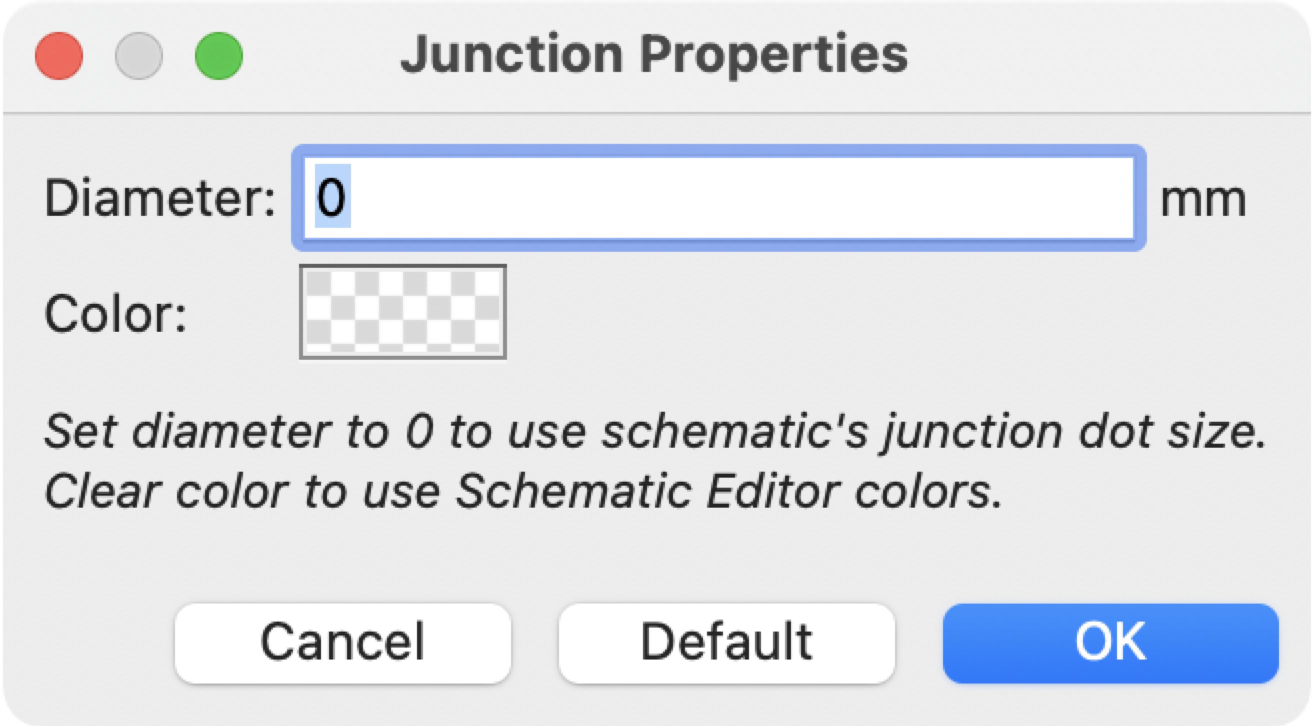


Giunzioni di fili

I fili che si incrociano non sono implicitamente connessi. È necessario collegarli assieme esplicitamente con un punto di giunzione se si vuole stabilire una connessione (pulsante nella barra strumenti a destra). I punti di giunzione verranno aggiunti automaticamente ai fili che cominciano o finiscono su altri fili esistenti.

I punti di giunzione sono utilizzati nella figura dello schema precedente sui fili collegati ai pin 18, 19, 20, 21, 22 e 23 di `P1`.

Junction size automatically follows the schematic's **Junction dot size** setting in **Schematic Setup** → **General** → **Formatting**. Color follows the netclass setting. The automatic size and color can be overridden in each junction dot's properties; a size of `0` is equivalent to the schematic default size, and clearing the color uses the netclass color.



Etichette

Labels are used to assign net names to wires and pins. Wires with the same net name are considered to be connected, so labels can be used to make connections without drawing direct wire connections.

Una net può avere un solo nome. Se due etichette diverse vengono posizionate sulla stessa rete, verrà generata una violazione ERC. Nella netlist verrà utilizzato solo uno dei nomi di net. Il nome della net finale è determinato in base alle regole descritte di seguito.

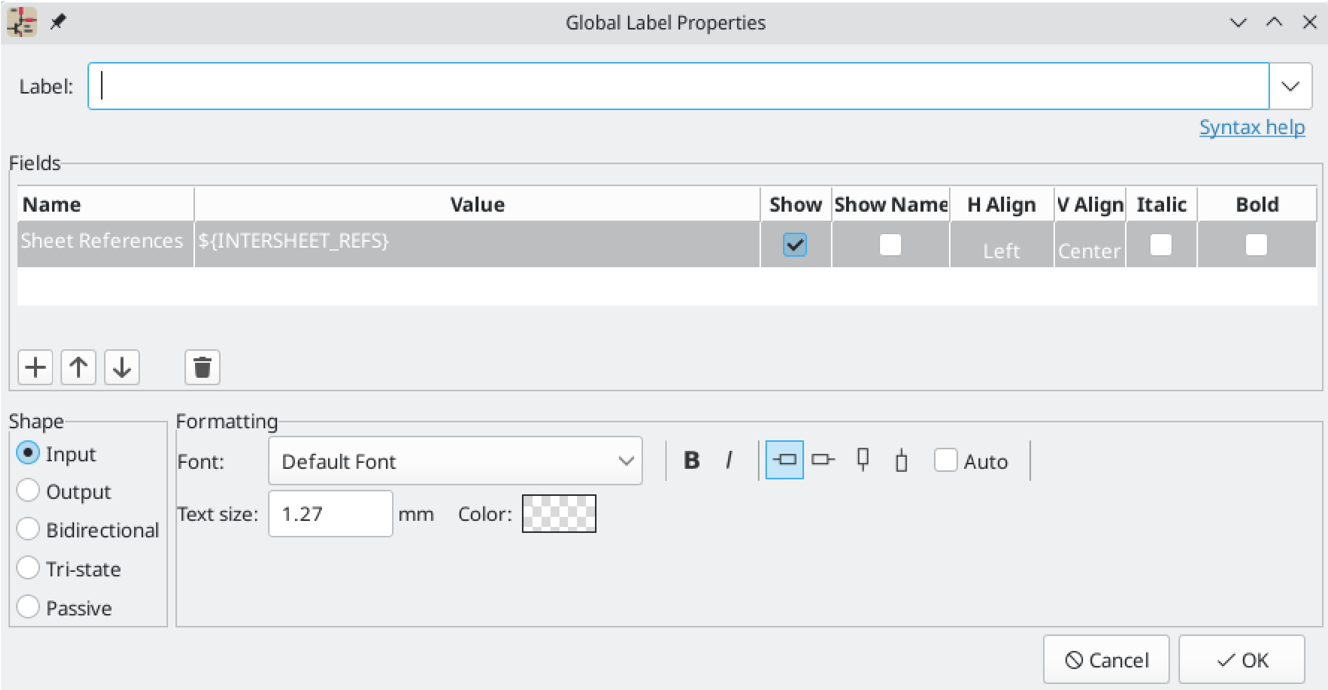
There are three types of labels, each with a different connection scope.

- **Local labels**, also referred to simply as labels, only make connections within a sheet. Add a local label with the button in the right toolbar.
- **Global labels** make connections anywhere in a schematic, regardless of sheet. Add a global label with the button in the right toolbar.
- **Hierarchical labels** connect to hierarchical sheet pins and are used in hierarchical schematics for connecting child sheets to their parent sheet. Add a hierarchical label with the button in the right toolbar.

NOTE	Labels that have the same name will connect, regardless of the label type, if they are in the same sheet.

Aggiunta e modifica etichette

After using the appropriate button or hotkey to create a label, the Label Properties dialog appears.



The **Label** field sets the label's text, which determines the net that the label assigns to its attached wire. Label text supports markup for overbars, subscripts, etc., as well as variable substitution. Use the **Syntax help** link in the dialog for a summary.

There are several options to control the label's appearance. You can change the font, size, and color of the text, and set bold and italic emphasis. You can also set the orientation of the text relative to the label's connection point. Hierarchical and global labels have several additional options: the **Auto** option automatically sets the label orientation based on the connected schematic elements, and **Shape** option controls the shape of the label outline (**Input**, **Output**, **Bidirectional**, **Tri-state**, or **Passive**). The outline shape is purely visual and has no electrical consequence.

NOTE The default text size can be set for a schematic in Schematic Setup, and the default font can be set in Preferences.

NOTE Global labels have additional settings to control margins around the label text in the Schematic Setup dialog.

Labels can also have fields added to them. Two fields have special meaning (`Net Class` and `Sheet References`, described below), but arbitrary fields can also be added. Label fields behave like symbol fields: you can show or hide their name and value and adjust the alignment, orientation, position, size, font, color, and emphasis.

NOTE Le opzioni di formattazione per i campi etichetta possono essere visualizzate o nascoste facendo clic con il pulsante destro del mouse sulla riga dell'intestazione della tabella campi etichetta e abilitando o disabilitando le colonne desiderate. Non tutte le colonne sono visibili per impostazione predefinita.

Like symbol fields, label fields can be edited individually by opening the properties of a specific label field from the schematic (double click the label field, or use `E`).

After accepting the label properties, the label is attached to the cursor for placement. The connection point for a label is the small square in the corner of the label. The square disappears when the label is connected to a wire or the end of a pin.

The connection point's position relative to the label text can be changed by choosing a different label orientation in the label's properties, or by mirroring/rotating the label.

The Label Properties dialog can be accessed at any time by selecting a label and using the `E` hotkey, double-clicking on the label, or with **Properties…** in the right-click context menu.

Assigning net classes with labels

In addition to assigning net names, labels can be used to assign net classes. A label field named `Net Class` assigns the specified netclass to the net associated with the label. To make it easier to assign net classes in this way, `Net Class` is the default name for new label fields, and `Net Class` fields present a dropdown list of all the net classes in the design. Net classes must be created in the Schematic Setup or Board Setup windows before they can be assigned with a label field.

For more information about assigning netclasses, see the netclass documentation.

Inter-sheet references

Global labels can display inter-sheet references, which are a list of page numbers for other places in the schematic where the same global label appears. Clicking an inter-sheet reference travels to the listed page. If multiple references are listed, clicking the reference list brings up a menu to select the desired page.

Inter-sheet references are globally controlled in the Schematic Setup window's Formatting page. References can be enabled or disabled, and the displayed format for the list can be adjusted, including with optional prefix or suffix characters.

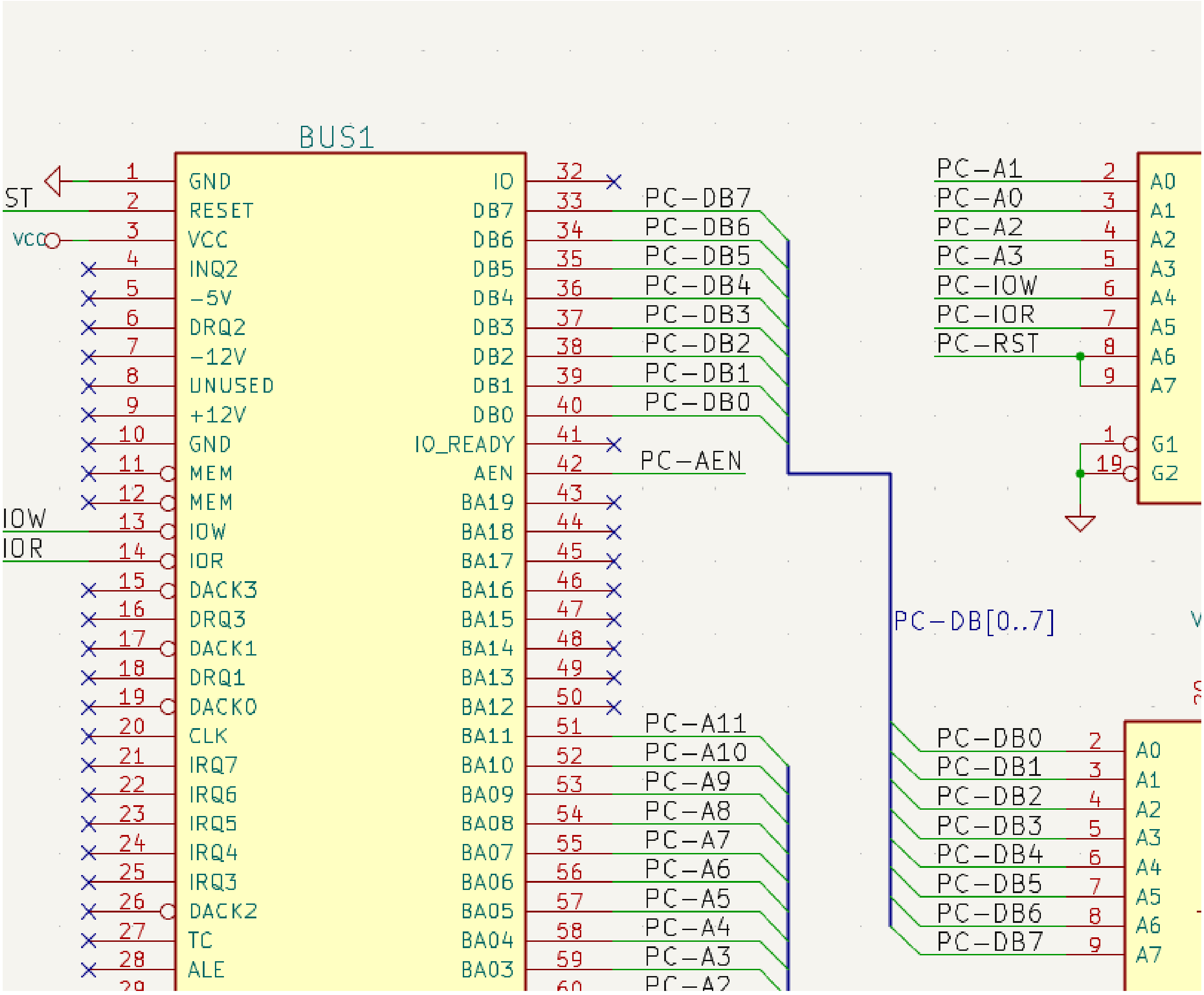
The image below shows a global label with inter-sheet references to two other schematic pages. A prefix and suffix of `[` and `]`, respectively, were added in Schematic Setup.

A `Sheet References` field with value `${INTERSHEET_REFS}` is automatically added to global labels, and is used to control the appearance of inter-sheet references for that label. The `${INTERSHEET_REFS}` text variable gets expanded to the full list of inter-sheet references for the global label, as configured in Schematic Setup. Visibility of inter-sheet references is globally controlled in Schematic Setup rather than with the `Sheet References` field visibility control. The `Sheet References` field has no meaning for other types of labels.

Buses

I bus sono un modo per raggruppare segnali in relazione tra loro in uno schema elettrico, in modo da semplificare i progetti complessi. I bus possono essere disegnati come i fili usando lo strumento bus , e i loro nomi vengono assegnati usando le etichette allo stesso modo di come si fa con i fili dei segnali.

Nello schema elettrico seguente, molti piedini sono connessi a dei bus, cioè le linee spesse blu al centro.



Membri di bus

There are two types of bus in KiCad 6.0 and later: vector buses and group buses.

Un **bus vettoriale** è un insieme di segnali che cominciano con un prefisso comune e finiscono con un numero. I bus vettoriali hanno nome nella forma `<PREFISSO>[M..N]` dove `PREFISSO` è un qualsiasi nome di segnale valido, `M` è il primo numero del suffisso, e `N` è l'ultimo numero del suffisso. Per esempio, il bus `DATA[0..7]` contiene i segnali `DATA0`, `DATA1`, e così via fino a `DATA7`. Non importa in quale ordine `M` ed `N` vengono specificati, ma entrambi devono essere positivi.

Un **bus di gruppo** è un insieme di uno o più segnali e/o bus vettoriali. I bus di gruppo possono essere usati per tenere assieme segnali correlati anche quando questi hanno nomi diversi. I bus di gruppo usano una sintassi etichetta speciale:

```
<NOME_OPZIONALE>{SEGNALE1 SEGNALE2 SEGNALE3}
```

I membri del gruppo sono elencati dentro parentesi graffe (`{}`) separati da spazi. Il nome opzionale del gruppo va prima della prima parentesi. Se il bus di gruppo è anonimo, i collegamenti risultanti sul C.S. saranno semplicemente i nomi dei segnali dentro il gruppo. Se il bus di gruppo possiede un nome, i collegamenti risultanti avranno il nome come prefisso, con un punto (`.`) di separazione tra il prefisso e il nome del segnale.

Per esempio, il bus `{SCL SDA}` ha due segnali membri, e nella netlist questi segnali saranno `SCL` e `SDA`. Il bus `USB1{DP DM}` genererà collegamenti chiamati `USB1.DP` e `USB1.DM`. Per progetti con bus grandi, ripetuti tra diversi circuiti simili, l'uso di questa tecnica può far risparmiare tempo.

I bus di gruppo possono contenere anche bus vettoriali. Per esempio, il bus `MEMORY{A[7..0] D[7..0] OE WE}` contiene sia bus vettoriali che segnali normali, il ché porterà a del collegamenti del tipo `MEMORY.A7` e `MEMORY.OE` sul C. S. .

I fili di bus si possono disegnare e collegare allo stesso modo dei fili dei segnali, compreso l'uso di giunzioni per creare connessioni tra fili che si incrociano. Come per i segnali, i bus non possono avere più di un nome — se due etichette in conflitto sono associate allo stesso bus, verrà generato un errore di controllo regole elettriche (ERC).

Connessioni tra membri di bus

Piedini connessi tra gli stessi membri di un bus devono essere connessi da etichette. Non è possibile connettere un piedino direttamente ad un bus; questo tipo di connessione sarà ignorata da KiCad.

Nell'esempio sopra, le connessioni vengono effettuate dalle etichette piazzate sui fili connessi ai piedini. Le voci di bus (segmenti di filo a 45 gradi) sono solo elementi grafici, e non sono necessarie per formare connessioni logiche.

In effetti, usando il comando di ripetizione (`Ins`), le connessioni possono essere eseguite molto velocemente nel modo seguente, se i piedini del componente sono allineati in ordine incrementale (una pratica comune in componenti come memorie, microprocessori, ecc.):

- Piazzare la prima etichetta (per esempio `PCA0`)
- Usare il tasto di ripetizione quanto serve per piazzare membri. KiCad creerà automaticamente le etichette successive (`PCA1`, `PCA2`…) allineate verticalmente, teoricamente nella esatta posizione degli altri piedini.
- Disegnare il filo sotto la prima etichetta. Usare poi il comando di ripetizione per piazzare gli altri fili sotto le etichette.
- Se necessario, piazzare le voci di bus allo stesso modo (piazzare la prima voce, poi usare il tasto di ripetizione).

NOTE

Nella sezione **Editor schemi elettrici → Opzioni di modifica** del menu delle preferenze, è possibile impostare i parametri di ripetizione:

- Passo orizzontale
- Passo verticale
- Incremento etichetta (che perciò potrà essere incrementata o decrementata di 2, 3, ecc.)

Dispiegamento bus

Lo strumento di dispiegamento permette di estrarre velocemente i segnali da un bus. Per dispiegare un segnale, fare clic con il tasto destro del mouse su un oggetto di tipo bus (un filo di bus, ecc) e scegliere

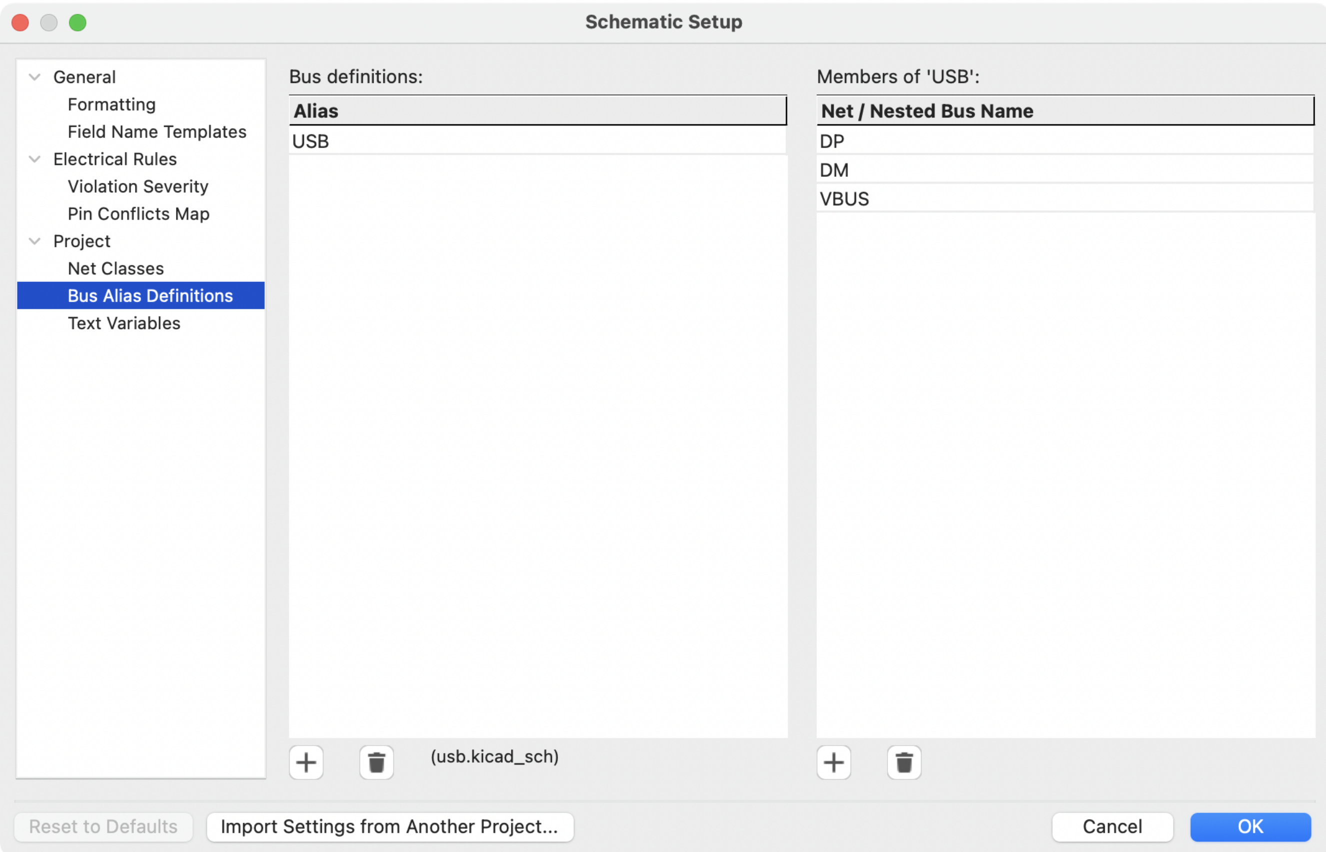
Dispiega bus. In alternativa, usare il tasto comando **Dispiega bus** (predeinito: `C`) quando il puntatore è posizionato sopra un oggetto di tipo bus. Il menu permette di selezionare quale membro del bus dispiegare.

Dopo aver selezionato il membro del bus, il successivo clic posizionerà l'etichetta del membro del bus alla posizione desiderata. Lo strumento genera automaticamente una voce bus ed un filo in direzione della posizione dell'etichetta. Dopo il posizionamento dell'etichetta, si può continuare a posizionare altri segmenti di filo (per esempio, per connetterli ai pin di un componente) e completare il collegamento in uno qualsiasi dei normali metodi.

Alias di bus

Gli alias di bus sono scorciatoie che permettono di lavorare con grandi insiemi di bus in modo più efficiente. Essi permettono di definire un gruppo di bus e dare ad esso un nome corto che può essere usato poi al posto del nome completo in tutto lo schema elettrico.

To create bus aliases, open the **Bus Alias Definitions** pane in Schematic Setup.



Ad un alias si può dare come nome un qualsiasi nome di segnale valido. Usando la finestra di dialogo, si possono aggiungere segnali o bus vettoriali all'alias. Come scorciatoia, si può battere o incollare dentro un'elenco di segnali e/o buse separati da spazi, e questi verranno aggiunti alla definizione di alias. In questo esempio, definiamo un alias chiamato `USB` con membri `DP`, `DM`, e `VBUS`.

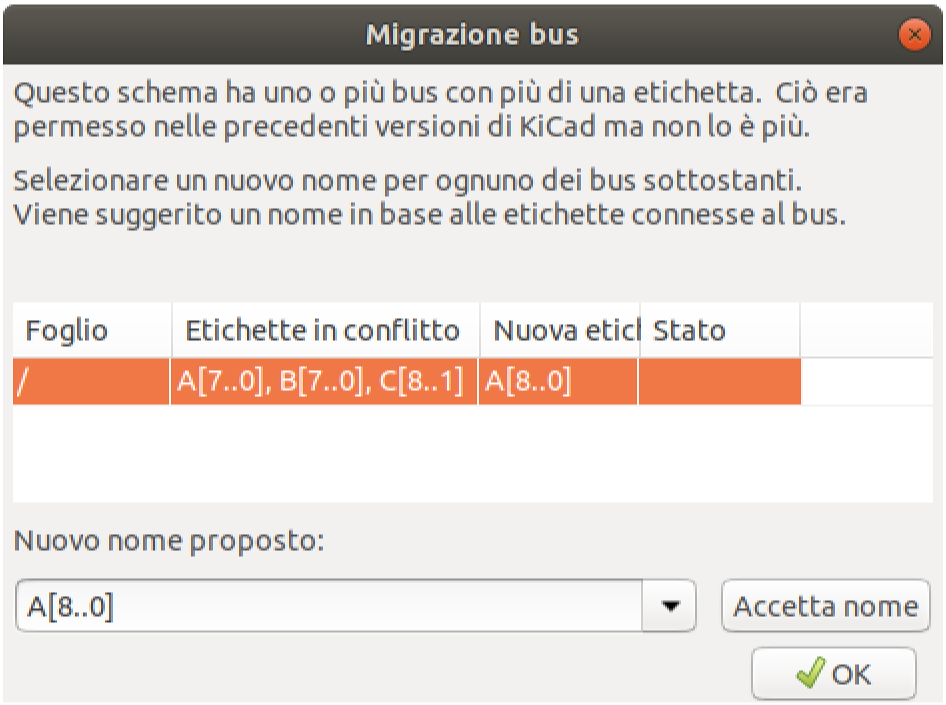
Dopo la definizione di un alias, esso può essere usato in una etichetta di bus di gruppo mettendo il nome dell'alias dentro le parentesi graffe del bus di gruppo: `{USB}`. Ciò ha lo stesso effetto dell'etichettare il bus `{DP DM VBUS}`. Si può anche aggiungere un nome prefisso al gruppo, come `USB1{USB}`, il che da come risultati collegamenti come `USB1.DP` come descritto sopra. Per bus complessi, l'uso di alias può rendere l'etichettatura dello schema elettrico molto più corta. Si faccia attenzione al fatto che gli alias sono solo scorciatoie, e che il nome dell'alias non viene incluso nella netlist.

Bus aliases are saved in the schematic file that is opened when the alias is created. The **Bus Alias Definitions** window shows the schematic file associated with the selected alias at the bottom of the alias list. Any aliases created in a given schematic sheet are available to use in any other schematic sheet that is in the same hierarchical design. If multiple sheets in a hierarchical design contain identically-named bus aliases, the aliases must all have the same members. ERC will report a violation if multiple bus aliases with the same name do not have consistent members.

Bus con più di una etichetta

KiCad 5.0 e versioni precedenti permettevano la connessione di bus con diverse etichette assieme, e collegavano assieme i membri di detti bus durante la creazione della netlist. Questo comportamento è stato eliminato in KiCad 6.0 perché è incompatibile con i bus di gruppo, e anche perché tendeva a creare confusione nelle netlist perché il nome che un dato segnale avrebbe ricevuto non era facilmente predicibile.

Se si apre un progetto che faceva uso di questa caratteristica in una versione moderna di KiCad, si potrà osservare la finestra di dialogo di ``Migrazione bus'' che vi guiderà attraverso la procedura di aggiornamento dello schema in modo tale da garantire l'esistenza di una sola etichetta per un dato insieme di collegamenti bus.



Per ogni insieme di fili di bus che ha più di una etichetta, è necessario scegliere l'etichetta da tenere. Il menu a discesa permette di scegliere tra le etichette che esistono nel progetto, oppure è anche possibile scegliere un nome ancora diverso inserendolo manualmente nel campo del nuovo nome.

Hidden Power Pins

When the power pins of a symbol are visible, they must be connected, as with any other signal. However, symbols such as gates and flip-flops are sometimes drawn with hidden power input pins which are connected implicitly.

KiCad automatically connects invisible pins with type "power input" to a global net with the same name as the pin. For example, if a symbol has a hidden power input pin named `VCC`, this pin will be globally connected to the `VCC` net on all sheets.

NOTE Hidden pins can be shown in the schematic by checking the **Show hidden pins** option in the **Schematic Editor** → **Display Options** section of the preferences, or by selecting **View** → **Show hidden pins**. There is also a toggle icon on the left toolbar.

It may be necessary to join power nets of different names (for example, `GND` in TTL components and `VSS` in MOS components). To accomplish this, add a power symbol for each net and connect them with a wire.

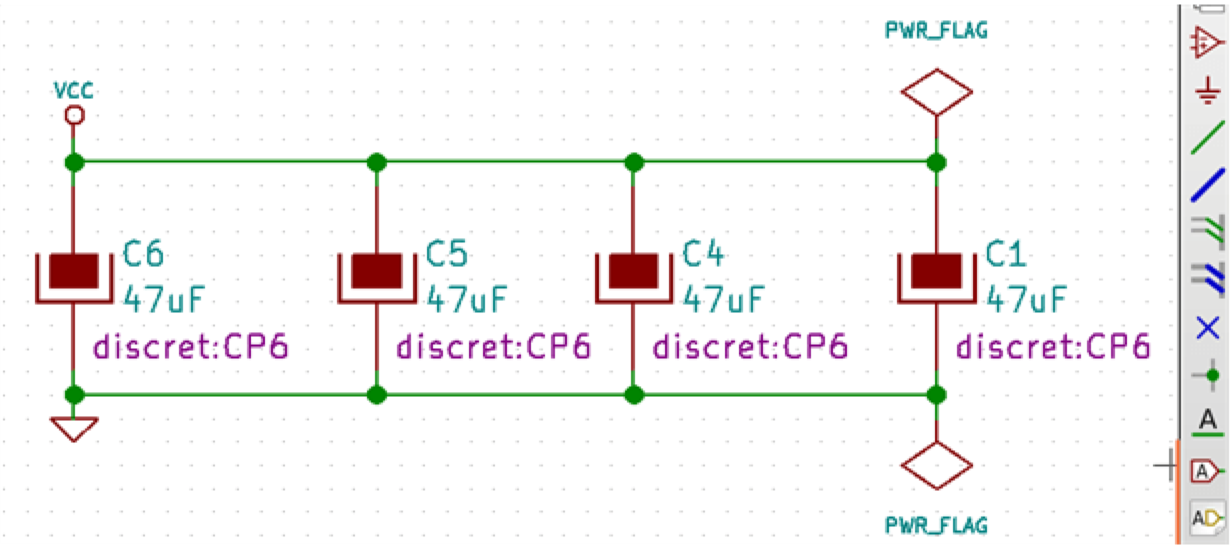
Se vengono usati pin di potenza nascosti, non è raccomandabile usare etichette locali per le connessioni di potenza, dato che queste ultime non connetterebbero i piedini di potenza nascosti su altri fogli.

NOTE Care must be taken with hidden power input pins because they can create unintentional connections. By nature, hidden pins are invisible and do not display their pin name. This makes it easy to accidentally connect two power pins to the same net. For this reason, **using invisible power pins in symbols is not recommended** outside of power symbols, and is only supported for compatibility with legacy designs and symbols.

Power Symbols

Power symbols are symbols that are conventionally used to represent a connection to a power net, such as `VCC` or `GND`. In addition to being a visual indicator that the attached net is a power rail, power symbols make global connections: two power symbols with the same pin name connect to each other anywhere in the schematic, regardless of sheet.

In the figure below, power symbols are used to connect the positive and negative terminals of the capacitors to the `VCC` and `GND` nets, respectively.



In the KiCad standard library, power symbols are found in the `power` library, but power symbols can be created in any library. To create a custom power symbol, make a new symbol with a power input pin that is set to be invisible. Name the pin according to the desired power net. In addition, set the "Define as power symbol" symbol property. As described in the hidden power pins section, invisible power input pins make global connections based on the hidden power pin's name. The process of creating a power symbol is described in more detail in the Symbol Editor section.

NOTE The connected net name is determined by the power symbol's **pin name**, not the name or value of the symbol. This means that power symbol net names can only be changed in the symbol editor, not in the schematic.

Net name assignment rules

Every net in the schematic is assigned a name, whether that name is specified by the user or automatically generated by KiCad.

When multiple labels are attached to the same net, the final net name is determined in the following order, from highest priority to lowest:

1. Etichette globali
2. Power symbols
3. Etichette locali
4. Etichette gerarchiche
5. Pin fogli gerarchici

If there are multiple labels of one type attached to a net, the names are sorted alphabetically and the first is used.

If a net travels through multiple sheets of a hierarchy, it will take its name from the highest level of the hierarchy where it has a hierarchical label or local label. As usual, local labels take priority over hierarchical labels.

If none of the label types above are attached to a net, the net's name is automatically generated based on the connected symbol pins.

PWR_FLAG

Due simboli `PWR_FLAG` sono visibili nella schermata soprastante. Essi indicano all'ERC che le due net di potenza `VCC` e `GND` sono effettivamente connesse ad una sorgente di potenza, dato che non c'è una sorgente di potenza esplicita come l'uscita di un regolatore di tensione collegata a nessuno dei due collegamenti.

Without these two flags, the ERC tool would diagnose: *Error: Input Power pin not driven by any Output Power pins.*

The `PWR_FLAG` symbol is found in the `power` symbol library. The same effect can be achieved by connecting any power output pin to the net.

Indicatore di Non-connesso

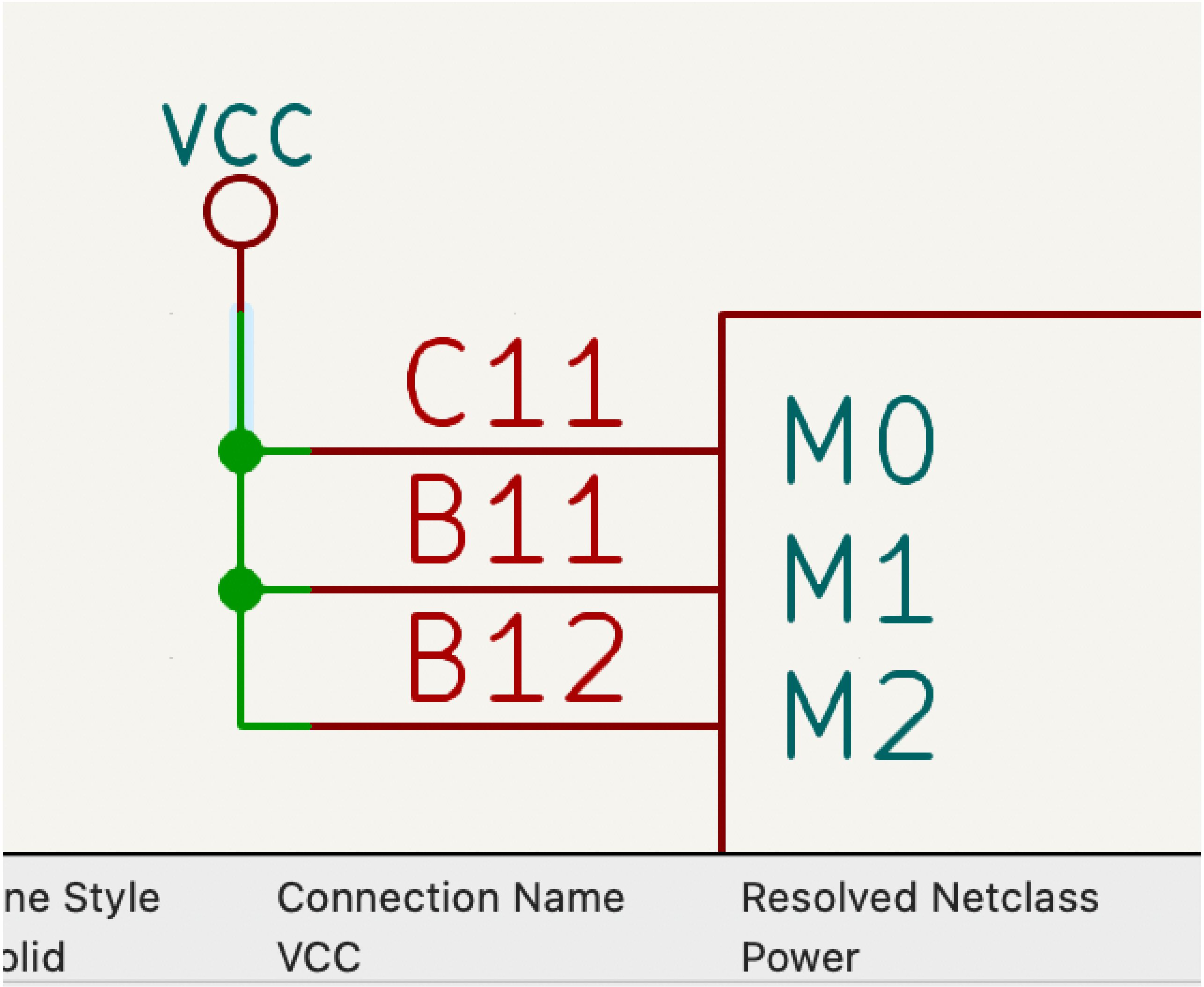
No-connection flags ( ) are used to indicate that a pin is intentionally unconnected. These flags do not have any effect on the schematic's connectivity, but they prevent "unconnected pin" ERC warnings for pins that are intentionally unconnected.

Netclasses

Netclasses are groups of nets that can be assigned design rules (for the PCB) and graphical properties (for the schematic). In KiCad, each net is part of exactly one net class. If you do not add a net to a specific class, it will be part of the Default class, which always exists.

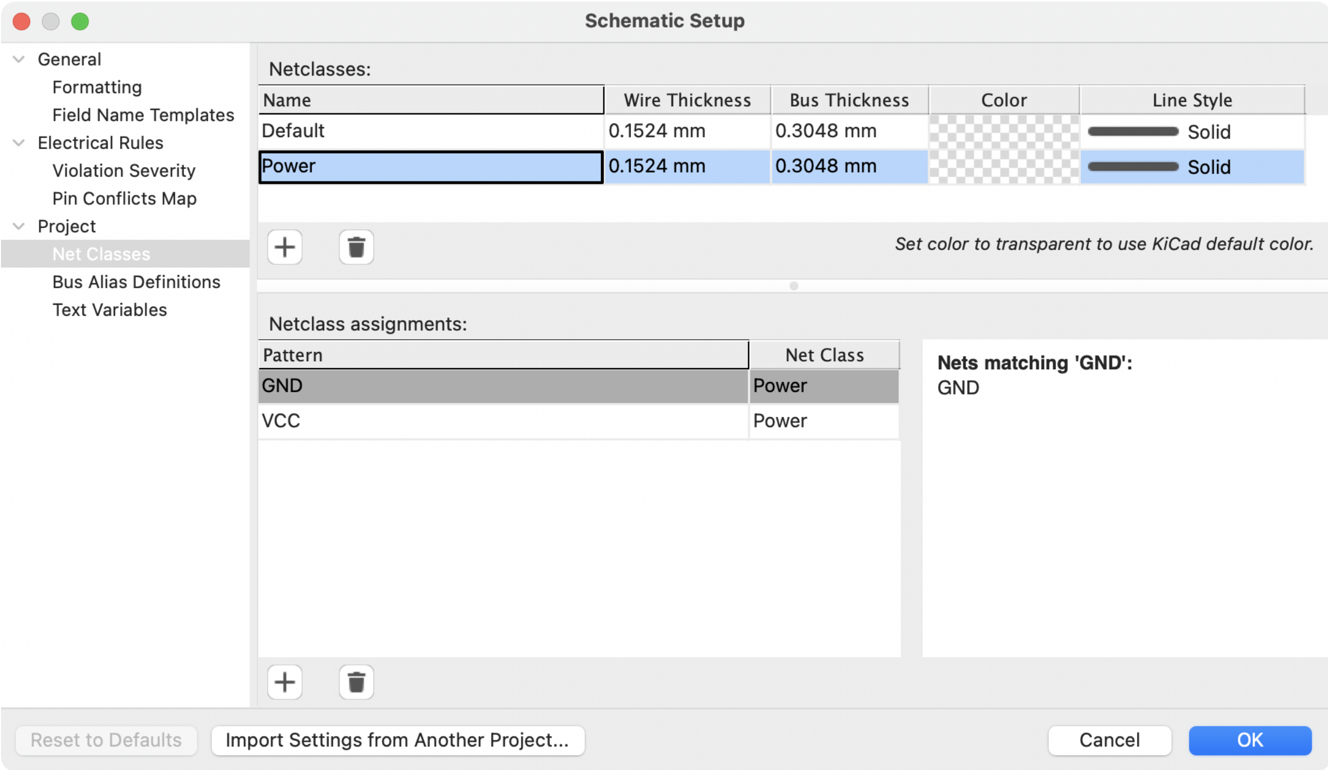
Net classes may be created and edited in either the Schematic or Board Setup dialogs. Nets can be added to netclasses in either the schematic or board using pattern-based assignments described below. Nets can also be assigned to netclasses in the schematic using graphical assignments with net class directives or net labels.

Selecting a wire or label displays the net's netclass in the message panel at the bottom of the window.



Managing netclasses in Schematic Setup

Netclasses are managed in the **Net Classes** panel of the **Schematic Setup** dialog.



The top pane lists the netclasses that exist in the design. The `Default` netclass always exists, and you can add additional netclasses with the + button or remove the selected netclass with the button.

Each netclass can have unique graphic properties that determine how wires of that netclass are displayed in the schematic. Wire and bus thicknesses, color, and line style (solid, dashed, dotted, etc.) can all be adjusted. Setting the color to transparent will use the theme's default wire/bus color for the netclass, which is configurable in Preferences.

You can also set board design rules for each netclass, although the DRC fields are hidden by default. Right click the header row to show or hide additional columns. For more information about setting netclass design rules, see the PCB editor documentation.

The bottom pane lists pattern-based netclass assignments. Each row has a net name pattern and a netclass; nets with names that match the pattern are assigned to the specified netclass. If a net matches multiple patterns, the first match is used. Pattern-based netclass assignments are dynamic: when a new net is added that matches an existing pattern, it will be assigned to the associated netclass automatically. Net patterns can use both wildcards (`*` to match any number of any characters, including none, and `?` to match any character) and regular expressions. The nets that match the selected pattern are displayed to the right of the pattern list.

For example, the `net*` pattern matches nets named `net`, `net1`, `network`, and any other net name beginning with `net`. Because `*` has a slightly different meaning in a regular expression (`*` matches zero or more of the preceding character), the `net*` pattern would also match a net named `ne`.

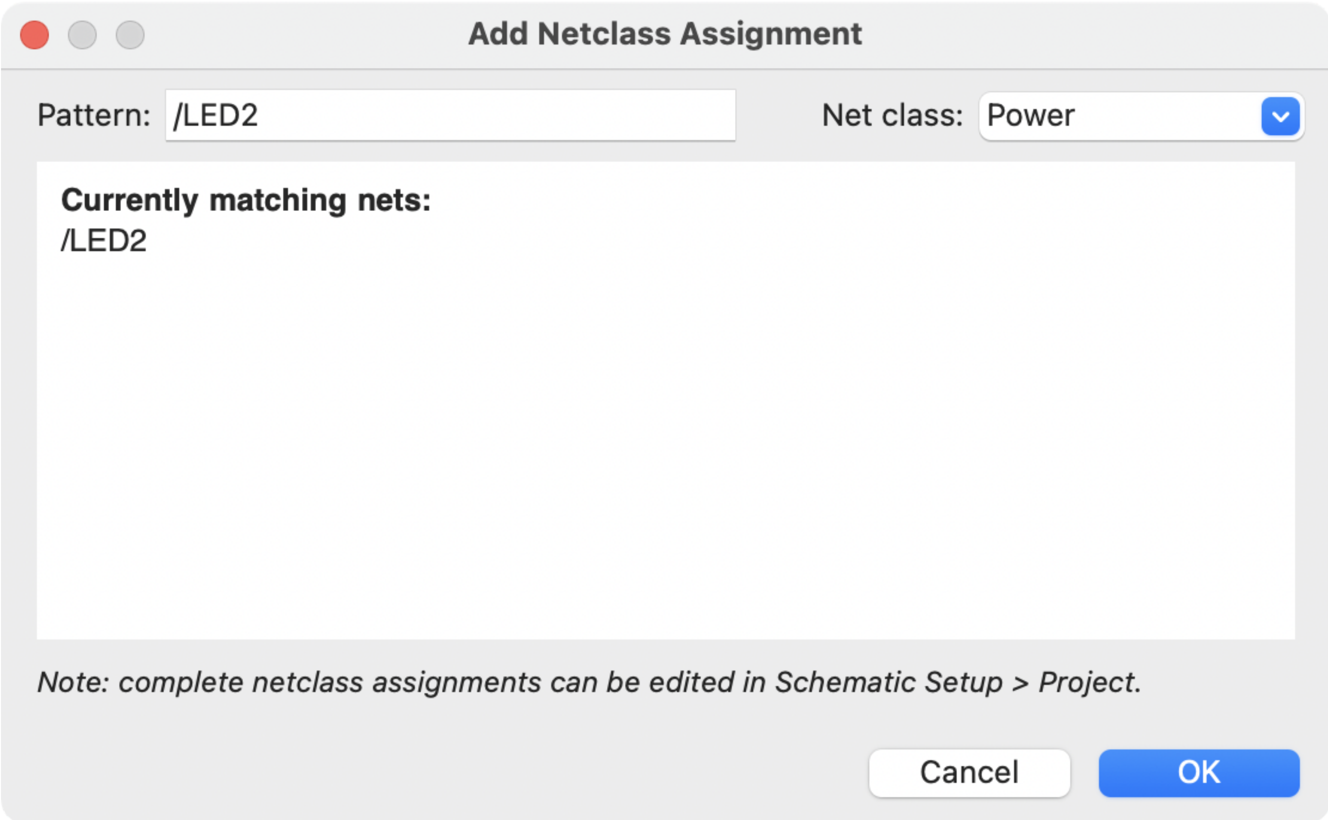
NOTE	Remember that net names must include the full sheet path. For example, a locally labeled net in the root sheet has a name prefixed with `/`.

Use the + button to add a net class assignment pattern or the button to remove a pattern.

NOTE

A netclass pattern containing only the `*` wildcard will match all explicitly named nets, but will not match unlabeled nets. To match unlabeled nets, you can include more of the net name before the wildcard character. All unlabeled nets have names that begin with `Net-`, so the pattern `Net-*` will match all unlabeled nets. You can also assign a netclass to an unlabeled net using a net class directive.

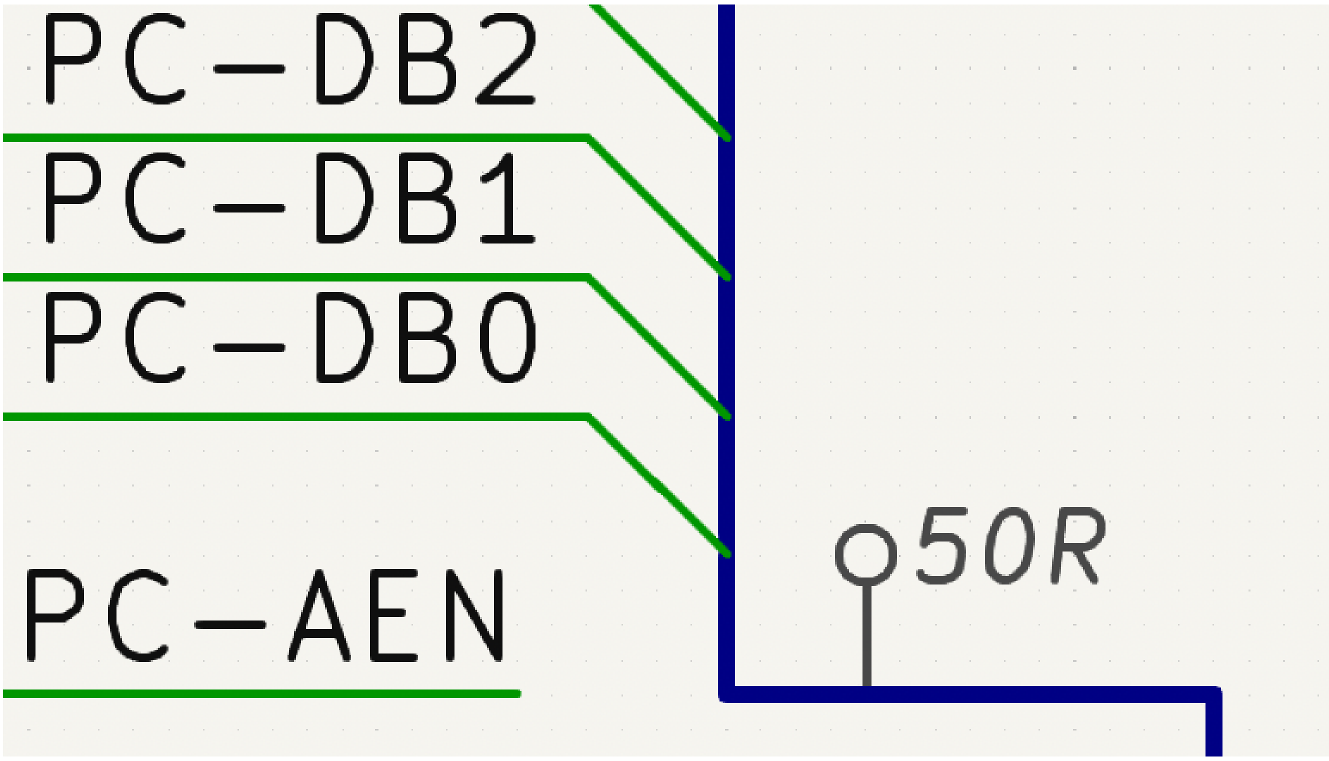
Instead of adding netclass patterns in the Schematic Setup dialog, you can directly create netclass patterns from the schematic canvas. Right click a net and select **Assign Netclass...** to bring up the **Add Netclass Assignment** dialog. The netclass pattern is pre-filled with the name of the selected net, but the pattern can be changed if desired. All nets matching the pattern are displayed in the dialog.



Graphically assigning netclasses in the schematic

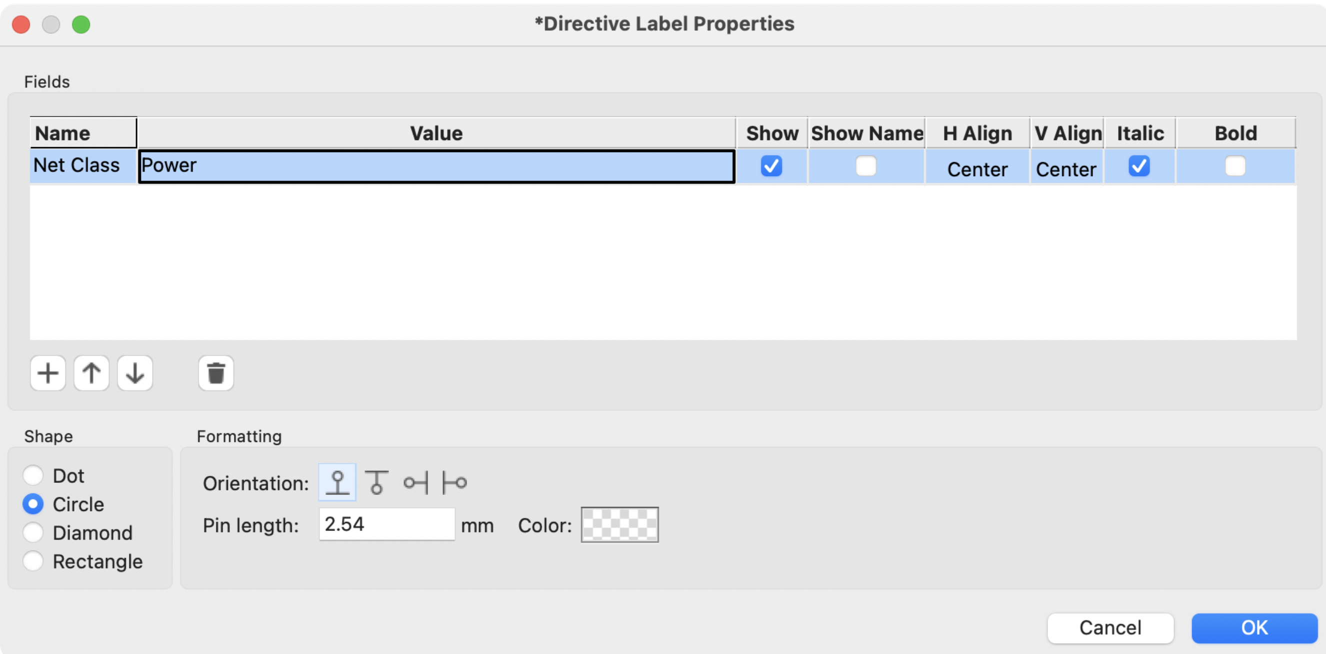
As an alternative to pattern-based netclass assignment, netclasses can be graphically assigned to nets in the schematic using either **net class directives** or **labels**. Netclasses must be created in Schematic Setup before they can be assigned graphically.

In the image below, a net class directive is used to assign signals to the `50R` netclass.



Net class directives are added with the button in the right toolbar. They behave like labels, except that they cannot be used to name a net. The attached net is assigned a netclass according to the value of the directive's `Net Class` field. The `Net Class` field presents a dropdown list of all the net classes in the design.

If a directive is attached to a bus, all members of the bus are assigned to the specified net class.



In addition to the associated netclass, you can edit the directive's **shape** (dot, circle, diamond, or rectangle), **orientation**, **pin length**, and **color** in the directive's properties.

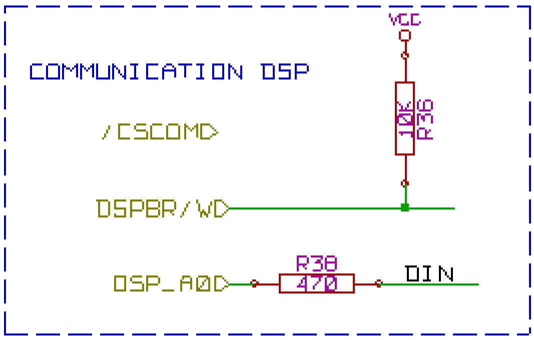
Net labels can also be used to assign netclasses to nets by adding a `Net Class` field to the label.

If more than one different netclass is graphically assigned to a single net, ERC will report an issue. Graphical netclass assignments override pattern-based assignments: if a net matches a netclass pattern assignment and also has a netclass assigned graphically, the graphically assigned netclass will be used.

Elementi grafici

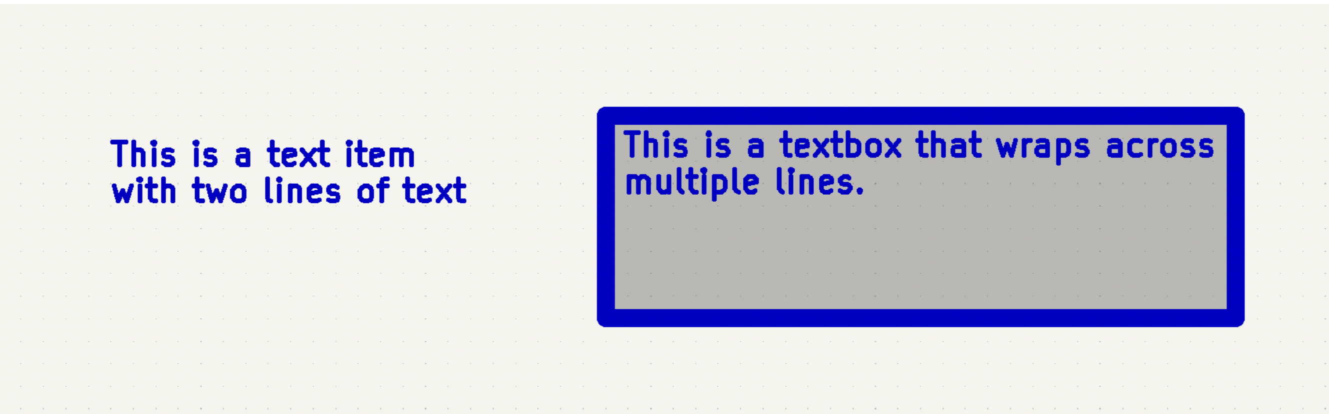
Text, graphic shapes, and images can be added to schematics for documentation purposes. These items do not have any electrical effect on the schematic.

The image below shows graphic lines and text ("COMMUNICATION DSP") in addition to symbols and several types of labels.



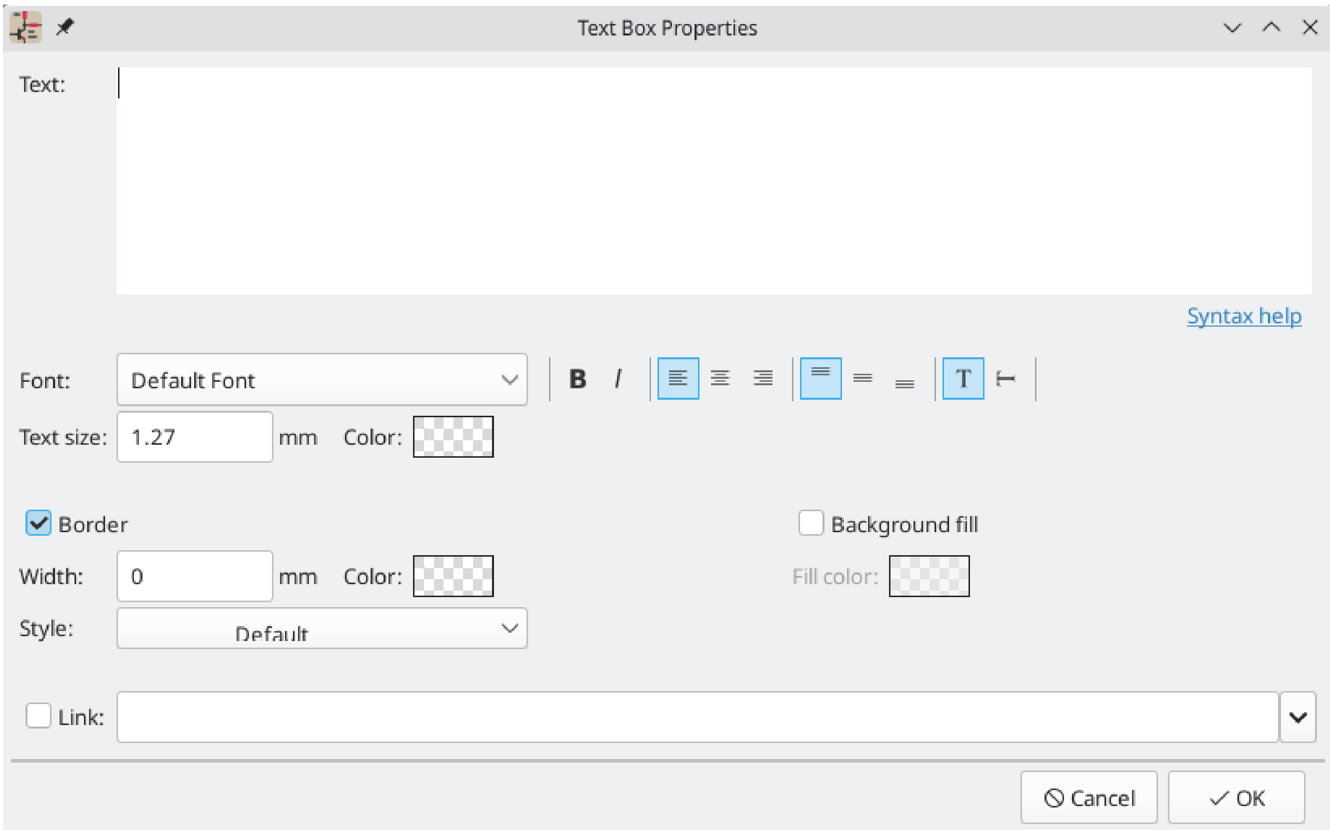
Text and Text Boxes

Two kinds of text can be added to schematics, which are referred to as text (T) and text boxes ( ). Both are added using their respective buttons in the right toolbar.



Both kinds of text item support multiline text and basic formatting features, but text boxes wrap text to fit in the outline and have additional formatting options. All text has adjustable fonts, color, size, bold and italic emphasis, left and right alignment, and vertical and horizontal orientation. Text boxes additionally support horizontal centering, vertical alignment options, and colored borders and fill.

NOTE	The default text size can be set for a schematic in Schematic Setup, and the default font can be set in Preferences.



Links

Text and text boxes can be made into a link by entering a target in the **Link** box in the text properties. The link target can be a local file (using the `file://` protocol prefix followed by the file's path), to a website (using `http://` or `https://` followed by the rest of the URL), or to another page in the same schematic (using `#` followed by the page number). These can also be autofilled using the dropdown menu in the link target box.

Fonts

Text and text boxes support custom fonts, which are selectable with the **Font** dropdown in the properties dialog for the text. In addition to the KiCad font, you can use any TTF font installed on your computer.

NOTE	User fonts are not embedded in the project. If the project is opened on another computer that does not have the selected font installed, a different font will be substituted. For maximum compatibility, use the KiCad font.

Text Markup

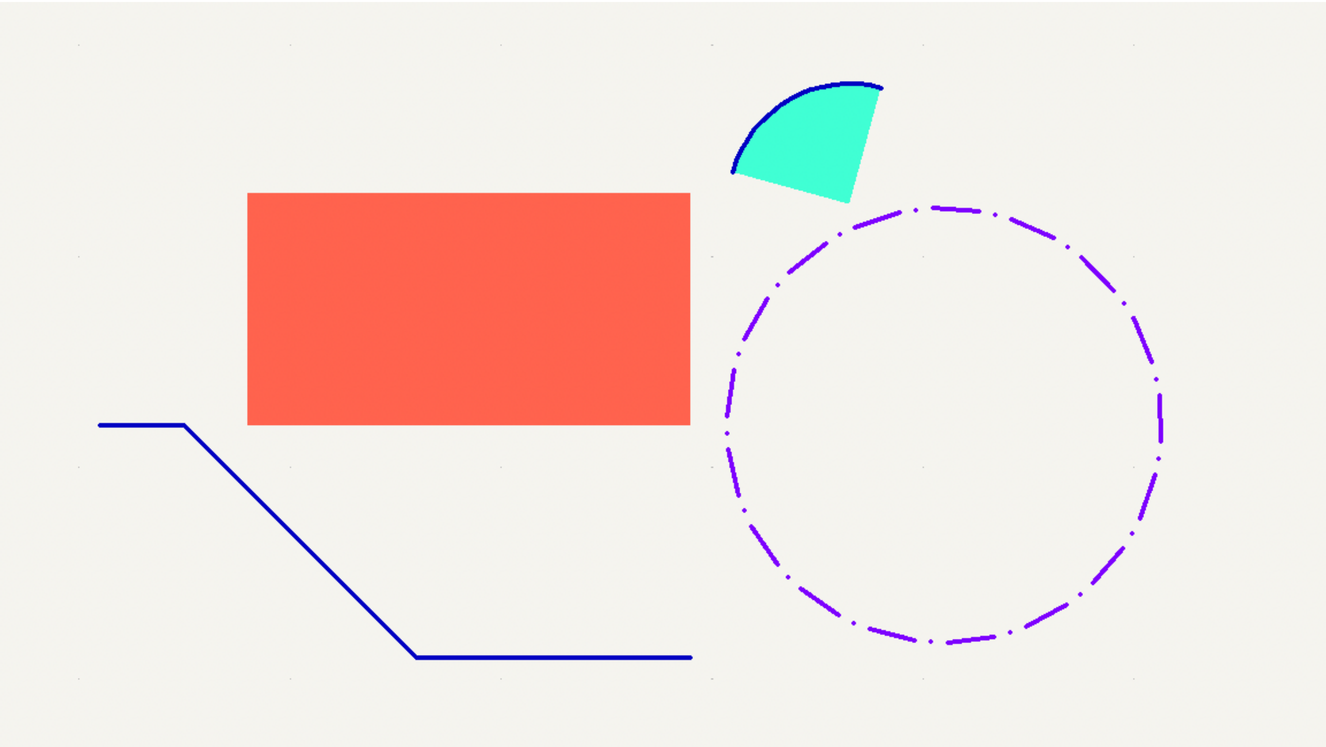
Text supports markup for superscripts, subscripts, overbars, evaluating project variables, and accessing symbol field values.

Feature	Markup Syntax	Result
Superscript	`text^{superscript}`	$\text{text}^{\text{superscript}}$
Subscript	`text_{subscript}`	$\text{text}_{\text{subscript}}$
Overbar	`~{text}`	$\overline{\text{text}}$
Variables	`${variable}`	*variable_value*
Symbol Fields	`${refdes:field}`	*field_value* of symbol *refdes*

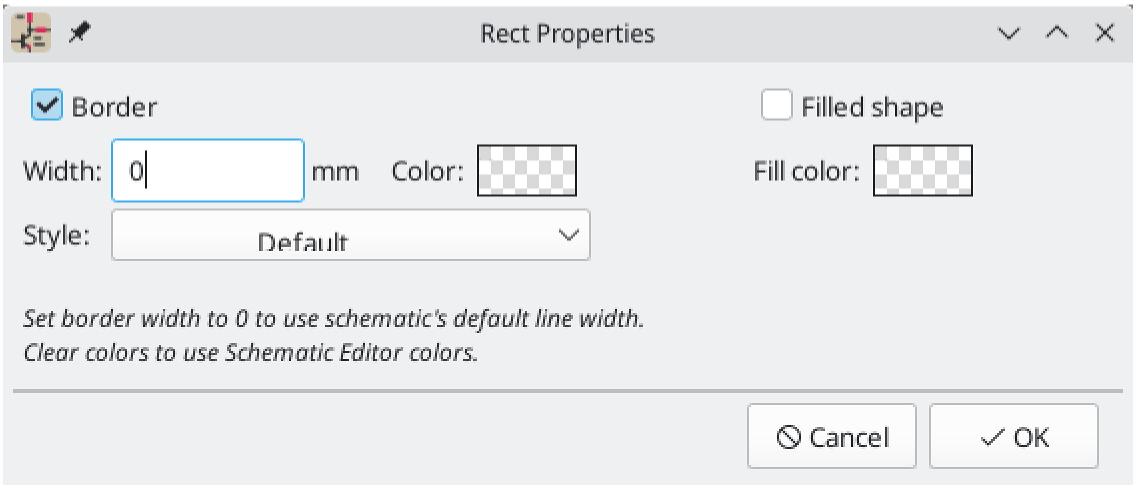
NOTE Variables must be defined in Schematic Setup before they can be used. There are also a number of built-in system text variables.

Graphic Shapes

Graphic rectangles ( ), circles ( ), arcs ( ), and lines ( ) can all be added using their respective buttons in the right toolbar.



Line width, color, and style (solid, dashed, or dotted) can be configured in the properties dialog for each shape (E). Rectangles, circles, and arcs can also have a fill color set and have their outlines removed.



Setting a shape's line width to 0 uses the schematic default line width, which is configurable in Schematic Setup. Spacing for line dashes is also configurable there. Removing a line or fill color uses the color theme's graphics color, which is configurable in Preferences.

Like wires, graphic lines obey the line drawing mode setting (90 degree, 45 degree, or free angle), which you can set using the toggle buttons on the left toolbar (, , and , respectively). Shift + Space cycles through the modes.

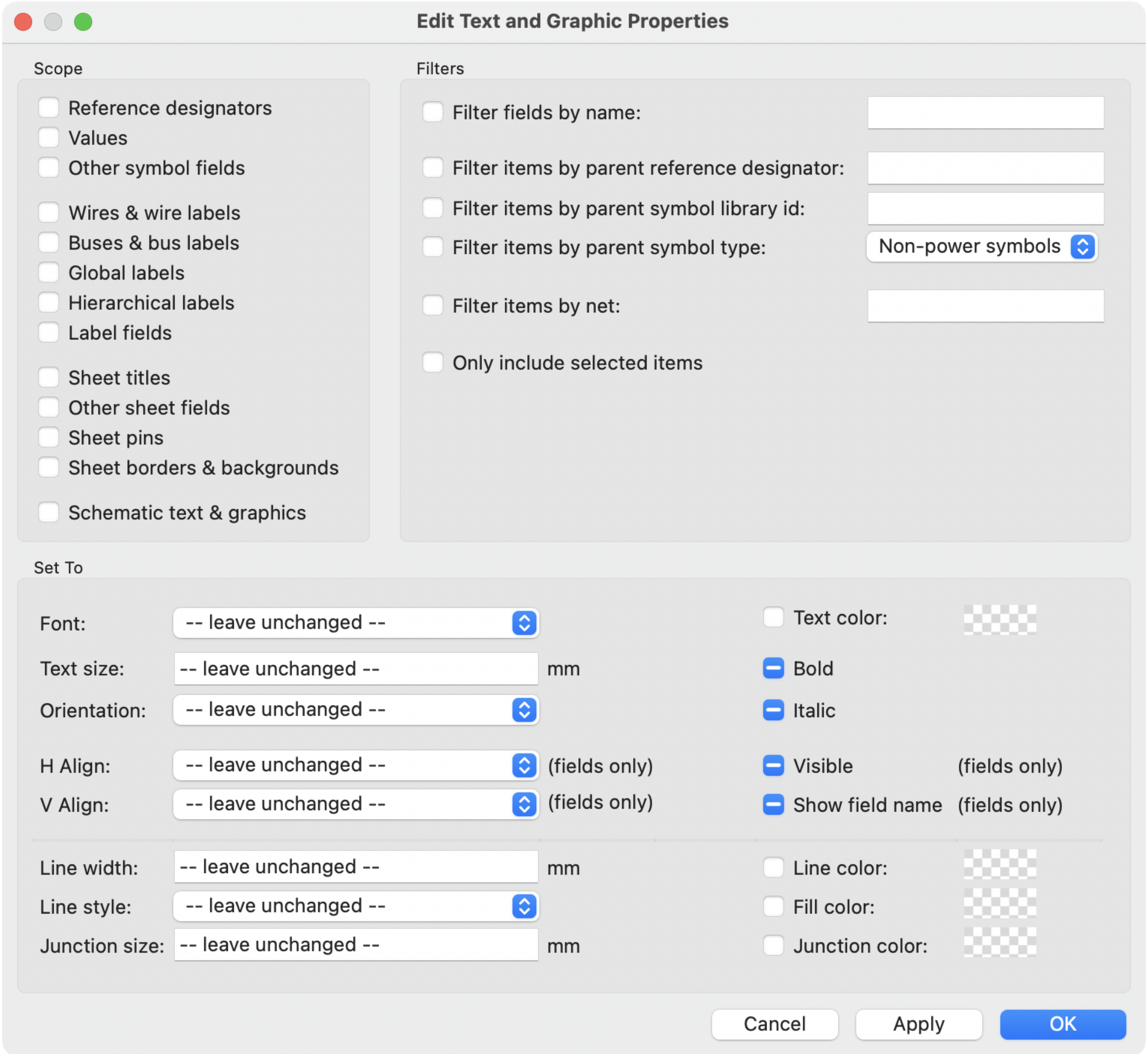
As with PCB tracks, the / hotkey switches line posture.

Bitmap Images

Bitmap images can be added to the schematic with the button. Images in the schematic can be moved and scaled. The properties dialog allows setting a location and scale as well as converting the image to greyscale.

Bulk editing text and graphics

Properties of text and graphics can be edited in bulk using the **Edit Text and Graphic Properties** dialog (**Tools** → **Edit Text and Graphic Properties...**). The tool can also modify visual properties of wires and buses.



Scope and Filters

Scope settings restrict the tool to editing only certain types of objects. If no scopes are selected, nothing will be edited.

Filters restrict the tool to editing particular objects in the selected scope. Objects will only be modified if they match all enabled and relevant filters (some filters do not apply to certain types of objects. For example, symbol field filters do not apply to wires and are ignored for the purpose of changing wire properties). If no filters are enabled, all objects in the selected scope will be modified. For filters with a text box, wildcards are supported: `*` matches any number of any characters, including none, and `?` matches any single character.

Filter fields by name filters to the specified symbol, label, or sheet field.

Filter items by parent reference designator filters to fields in the symbol with the specified reference designator. **Filter items by parent symbol library id** filters to fields in symbols with the specified library identifier. **Filter items by parent symbol type** filters to fields in symbols of the selected type (power or non-power).

Filter items by net filters to wires and labels on the specified net.

Only include selected items filters to the current selection.

Editable Properties

Properties for filtered objects can be set to new values in the bottom part of the dialog.

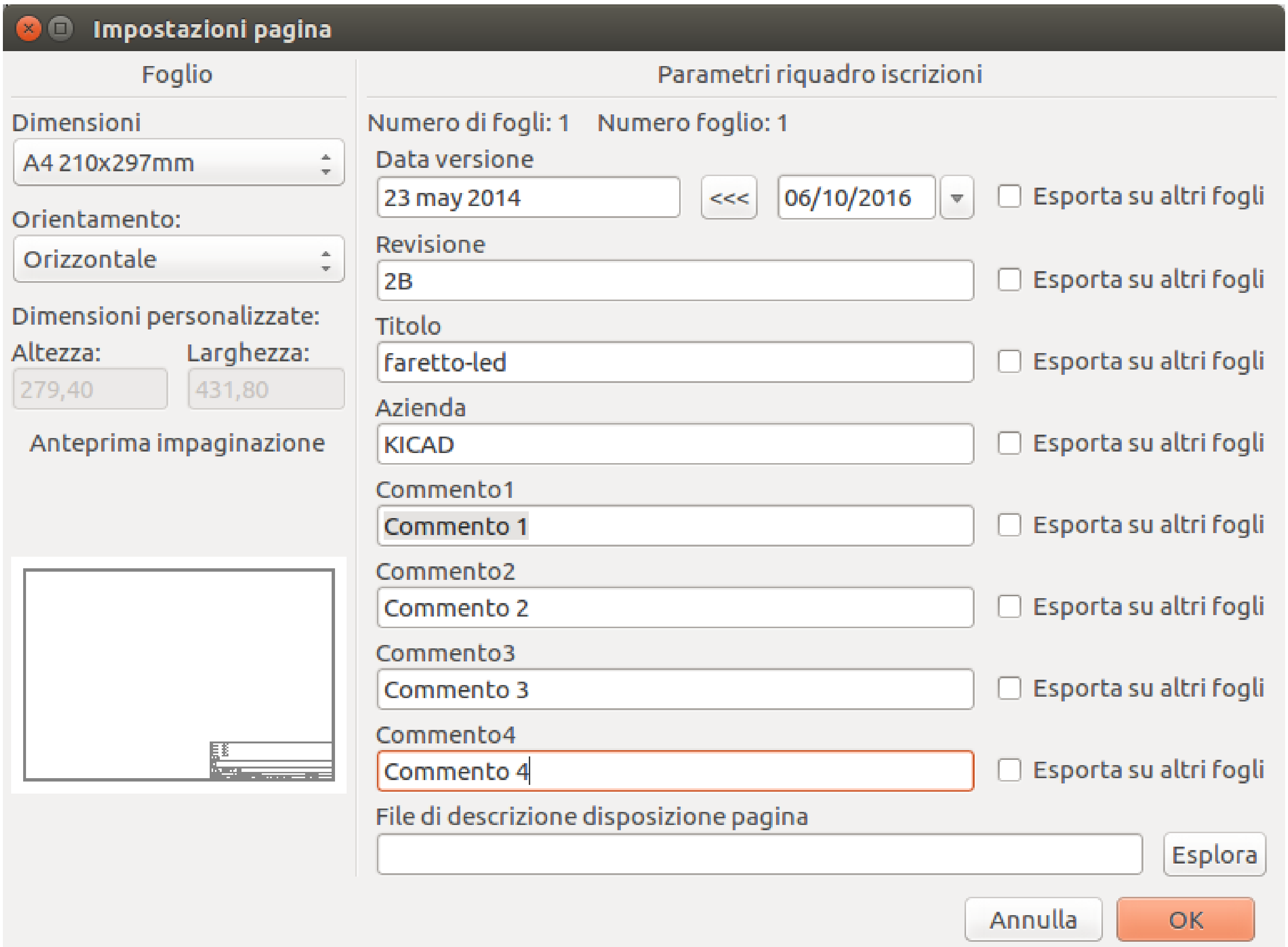
Drop-down lists and text boxes can be set to `-- leave unchanged --` to preserve existing values. Checkboxes can be checked or unchecked to enable or disable a change, but can also be toggled to a third "leave unchanged" state. Color properties must be checked to change the value; a checkerboard swatch indicates that the color will be inherited from the default value from the the schematic settings or netclass properties.

Text properties that can be modified are **font**, **text size**, **text orientation** (right/up/leftdown), **horizontal** and **vertical alignment**, **text color**, emphasis (**bold** and **italic**), and **visibility** of fields and field names.

Graphic and wire properties that can be modified are **line width**, **line style** (solid, dashed, and dotted lines), **line color**, **fill color** for shapes, and **junction size** and **junction color** for wire junctions.

Blocco del titolo del foglio

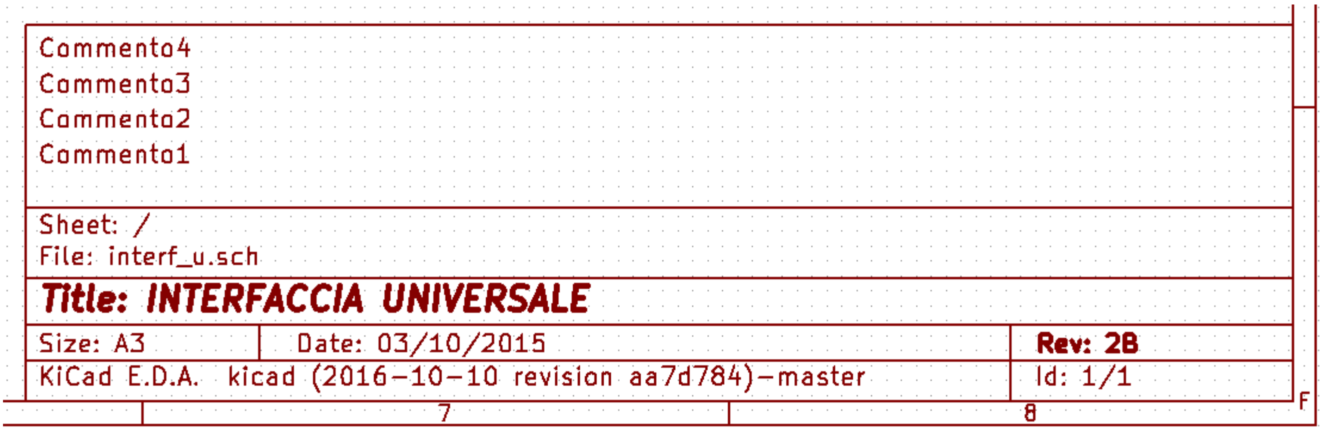
Il riquadro iscrizioni viene modificato con lo strumento per le inpostazioni pagina ( ).



Each field in the title block can be edited, as well as the paper size and orientation. If the **Export to other sheets** option is checked for a field, that field will be updated in the title block of all sheets, rather than only the current sheet.

You can set the date to today's or any other date by pressing the left arrow button next to **Issue Date**. Note that the date in the schematic will not be automatically updated.

A drawing sheet template file can also be selected.



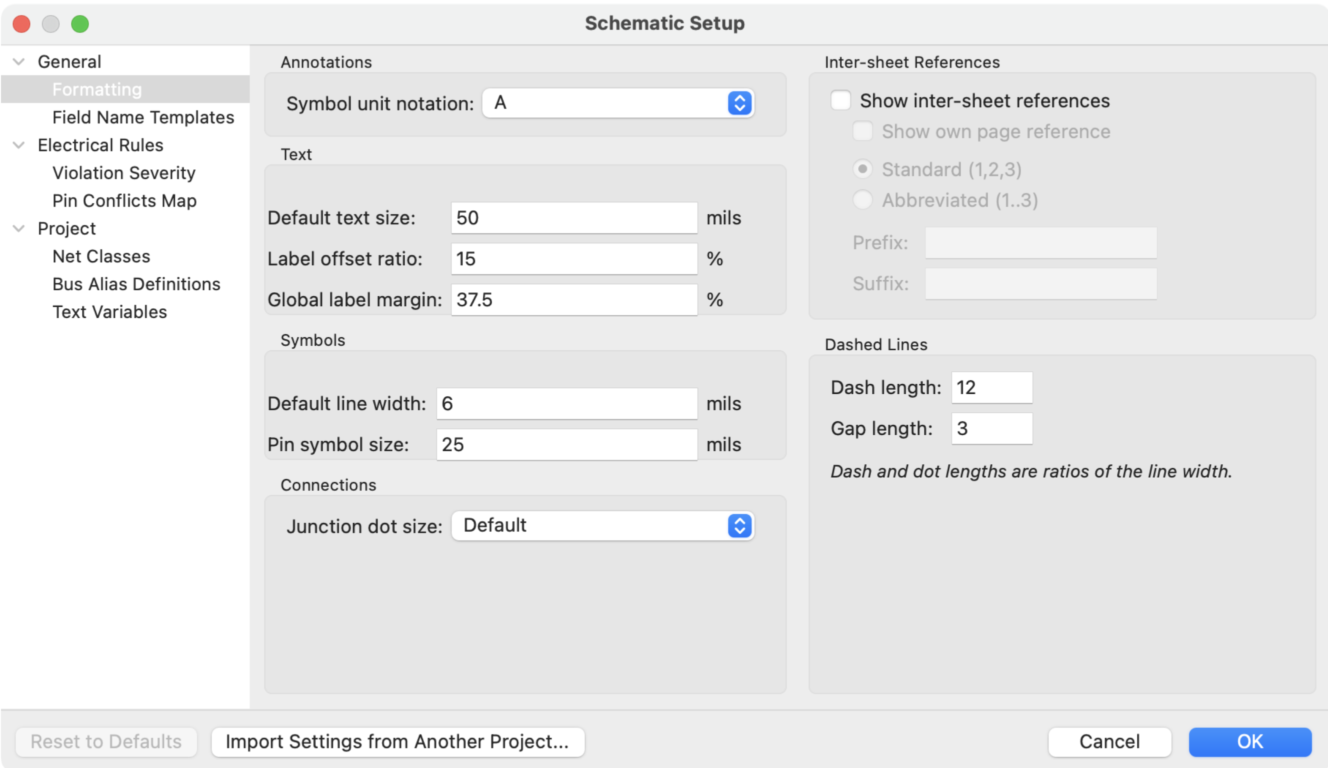
The sheet number (Sheet X/Y) is automatically updated, but sheet page numbers can also be manually set using **Edit → Edit Sheet Page Number…**.

Impostazioni schema

La finestra Impostazioni schema viene utilizzata per impostare le opzioni dello schema che sono specifiche dello schema attualmente attivo. Ad esempio, la finestra contiene le opzioni di formattazione, la configurazione delle regole elettriche, l'impostazione della netclass e l'impostazione delle variabili di testo dello schema.

You can import schematic settings from an existing project using the **Import Settings from Another Project…** button. This allows you to choose a project to use as a template and select which settings to import (formatting preferences, field name templates, pin conflict map, violation severities, and net classes).

Schematic formatting



The formatting panel contains settings for the appearance of symbols, text, labels, graphics, and wires.

Symbol unit notation sets how each unit of a multi-unit symbol is referred to in its reference designator. By default, a different letter for each unit is appended to the reference designator with no separator, for example `U1B` for the second unit of symbol `U1`, but this can be changed. Numbers can be used instead of letters, and various separators can be used between the symbol designator and the unit identifier (`.`, `-`, `_`, or none).

Default text size sets the default text height used by the text, text box, and label tools. **Label offset ratio** controls the vertical spacing between a local label's text and the attached wire, relative to the label's text size. This also affects the spacing between symbol pins and their pin number. **Global label margin** defines the size of the box around a global label, relative to the global label's text size. Increasing the margin may be useful to avoid overlapping text with overbars (`~{}`) or letters with descenders, but this may cause closely packed global labels to overlap with each other.

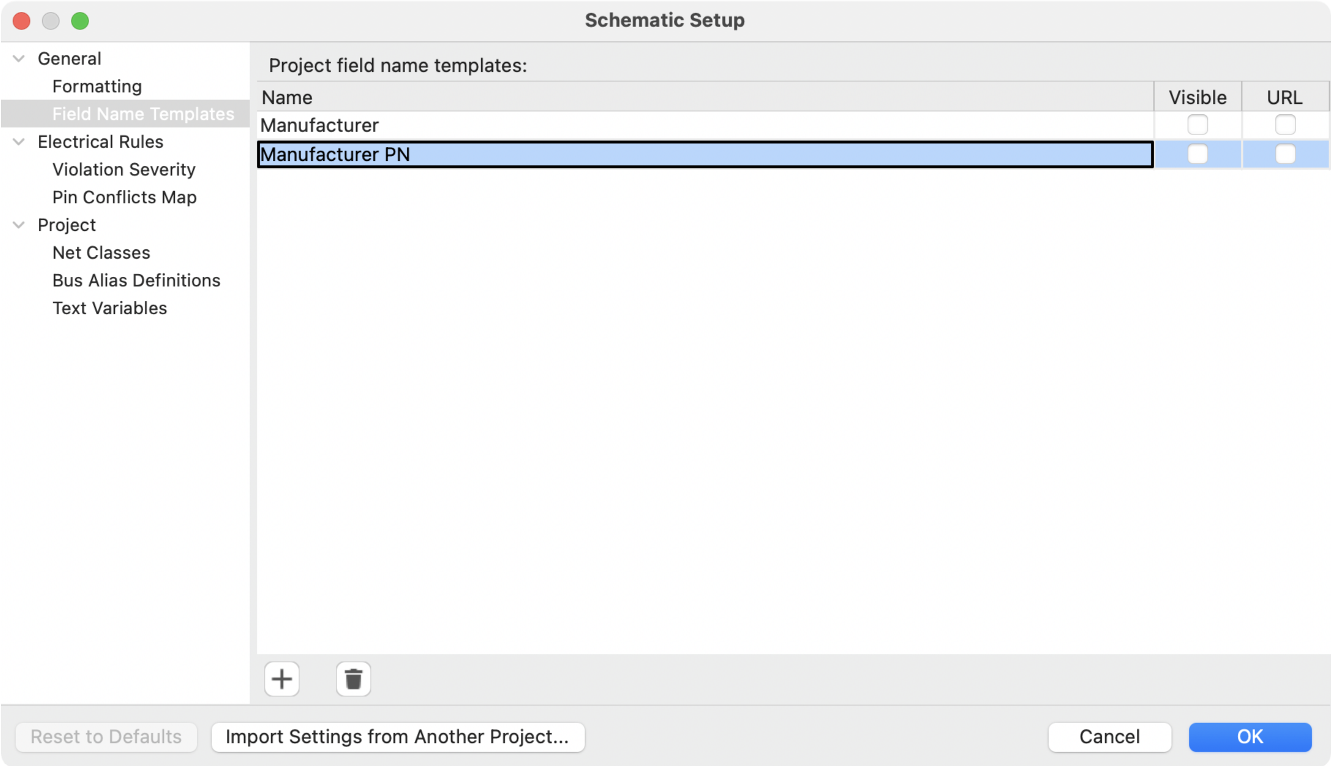
Default line width sets the default line width for symbol graphics, if the symbol does not override the default line width. **Pin symbol size** scales symbol pin graphic style annotations, such as the bubble on an inverted pin.

Junction dot size sets the schematic's default wire junction dot size. The default size can be overridden by editing an individual junction dot's properties.

Show inter-sheet references enables or disables the display of inter-sheet references, which are a list of page numbers next to a global labels that link to other places in the schematic where the same global label appears. **Show own page reference** controls whether the current page is included in the list of page numbers. **Standard** and **abbreviated** determine whether to display the complete list of page numbers or only the first and last page numbers. The **prefix** and **suffix** fields add optional characters before and after the list of page numbers. In the image of an inter-sheet reference below, a prefix and suffix of `[` and `]`, respectively, have been added.

Dashed line appearance is controlled in the Formatting section. **Dash length** controls the length of dashes, while **Gap length** controls the spacing between dashes and dots. The dash and gap lengths are relative to the line width: a gap length of `2` means twice the width of the line.

Field name templates



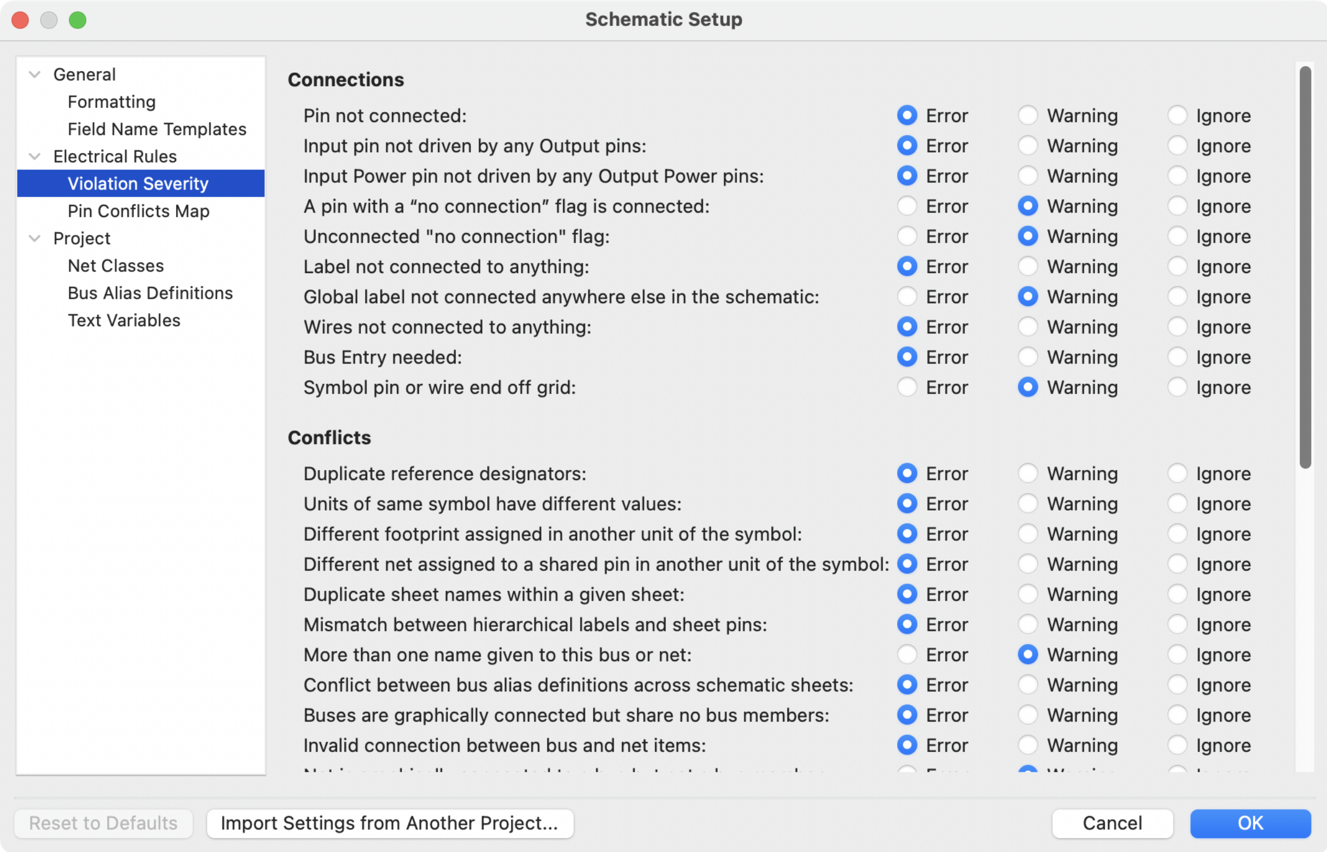
Field name templates are empty symbol fields that are automatically added to all symbols in the schematic. These can be useful when every symbol in the schematic needs additional fields beyond the fields that are defined in the library symbols, for example a field for the manufacturer's part number.

Template fields can be set as visible or invisible, and can also be set as URL fields.

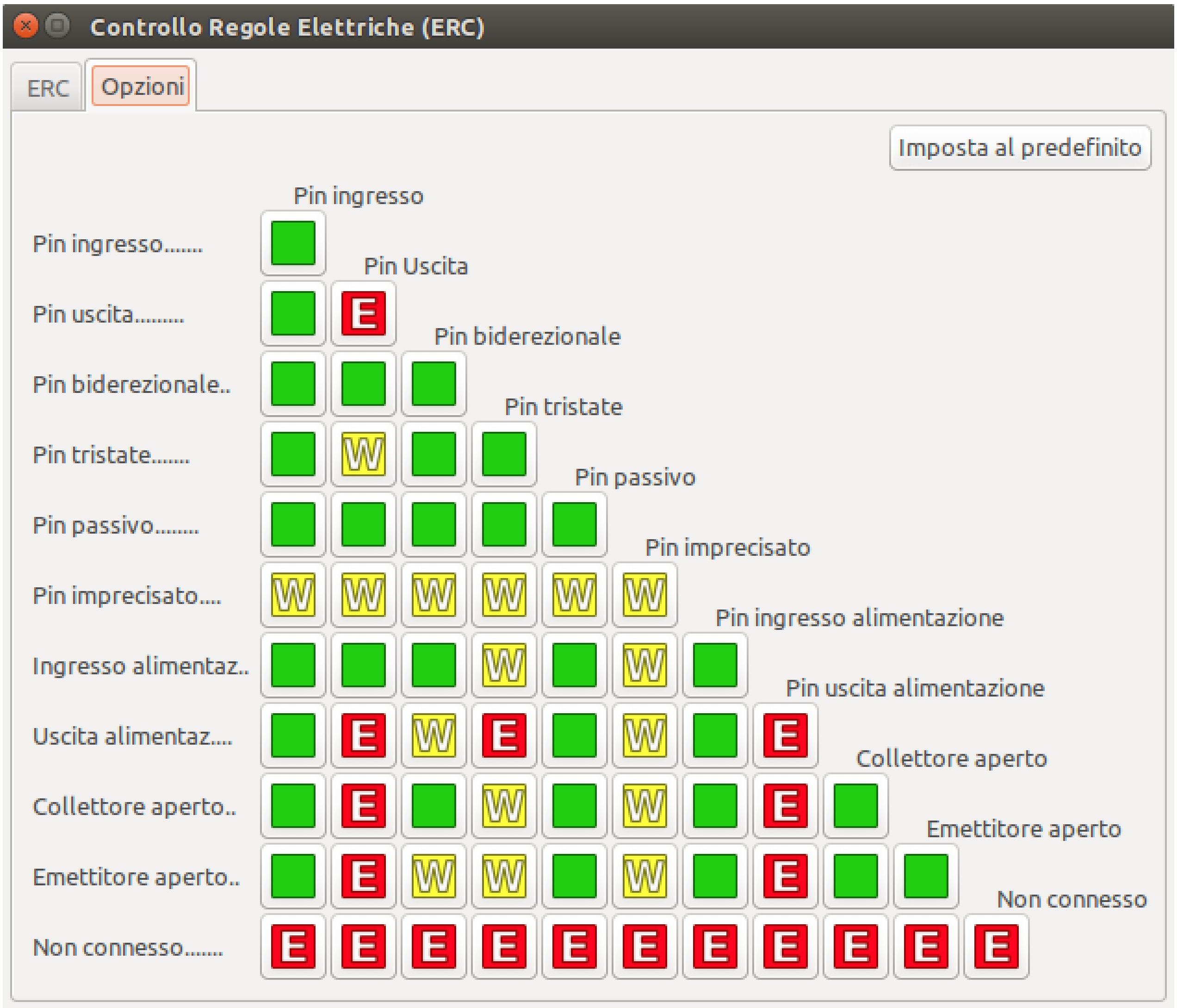
Field name templates that are defined in schematic setup apply only to the current project. Field name templates can also be defined in Preferences, which apply to all projects edited on your computer.

ERC violation severity and pin conflicts map

The **Violation Severity** panel lets you configure what types of ERC messages should be reported as Errors, Warnings, or ignored.

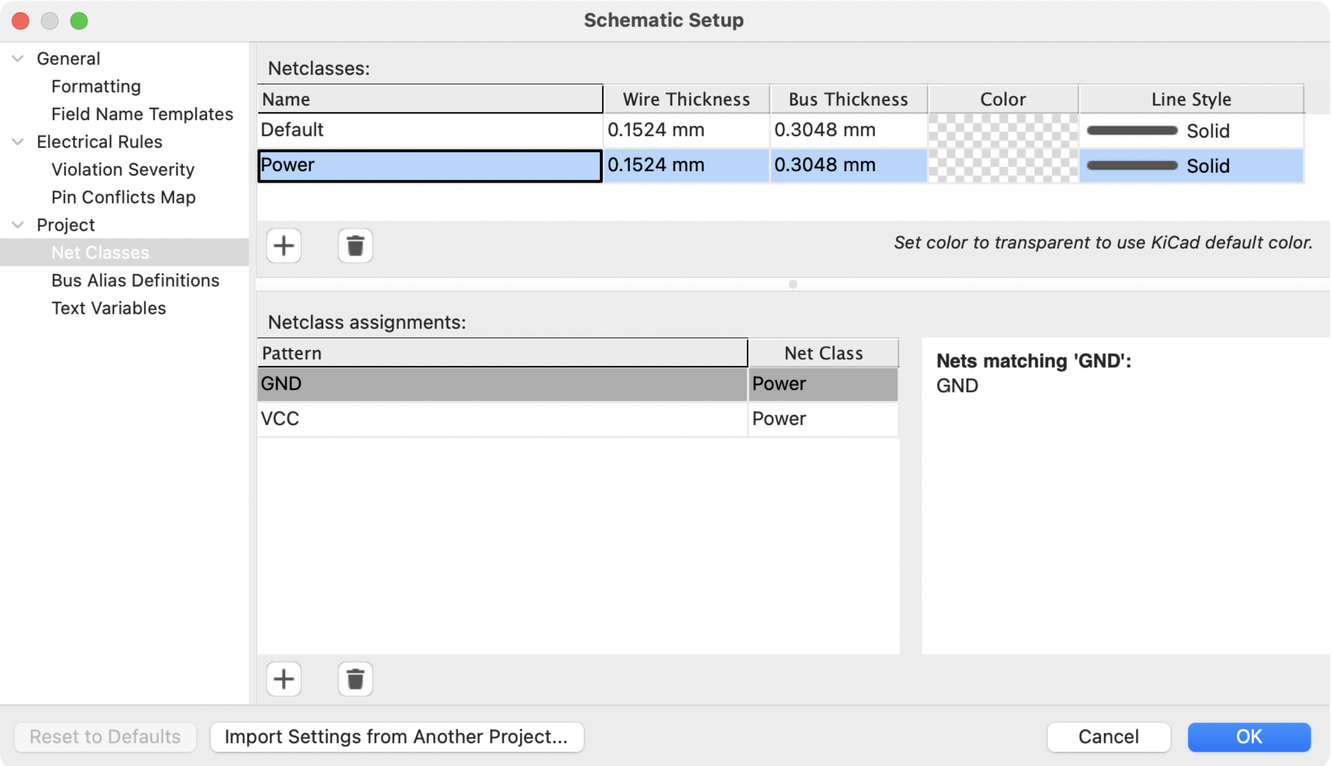


The **Pin Conflicts Map** allows you to configure connectivity rules to define electrical conditions for errors and warnings based on what types of pins are connected to each other. For example, by default an error is produced when an output pin is connected to another output pin.



These panels are explained in more detail in the ERC section.

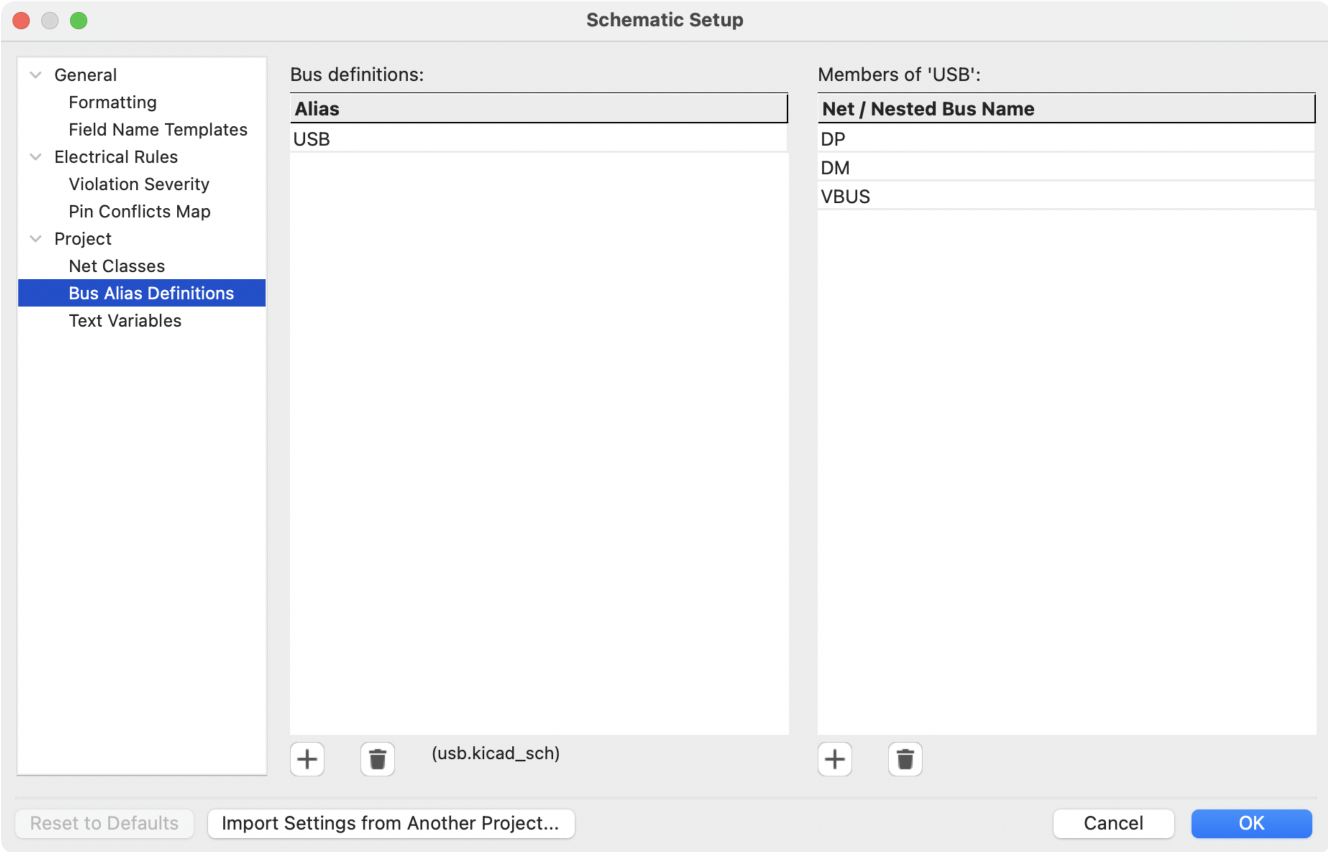
Net classes



The **Net Classes** panel allows you to manage netclasses for the project and assign nets to netclasses with patterns. Managing netclasses in this panel is equivalent to managing them in the Board Setup dialog. Nets can also be assigned to netclasses in the schematic using graphical assignments with net class directives or net labels.

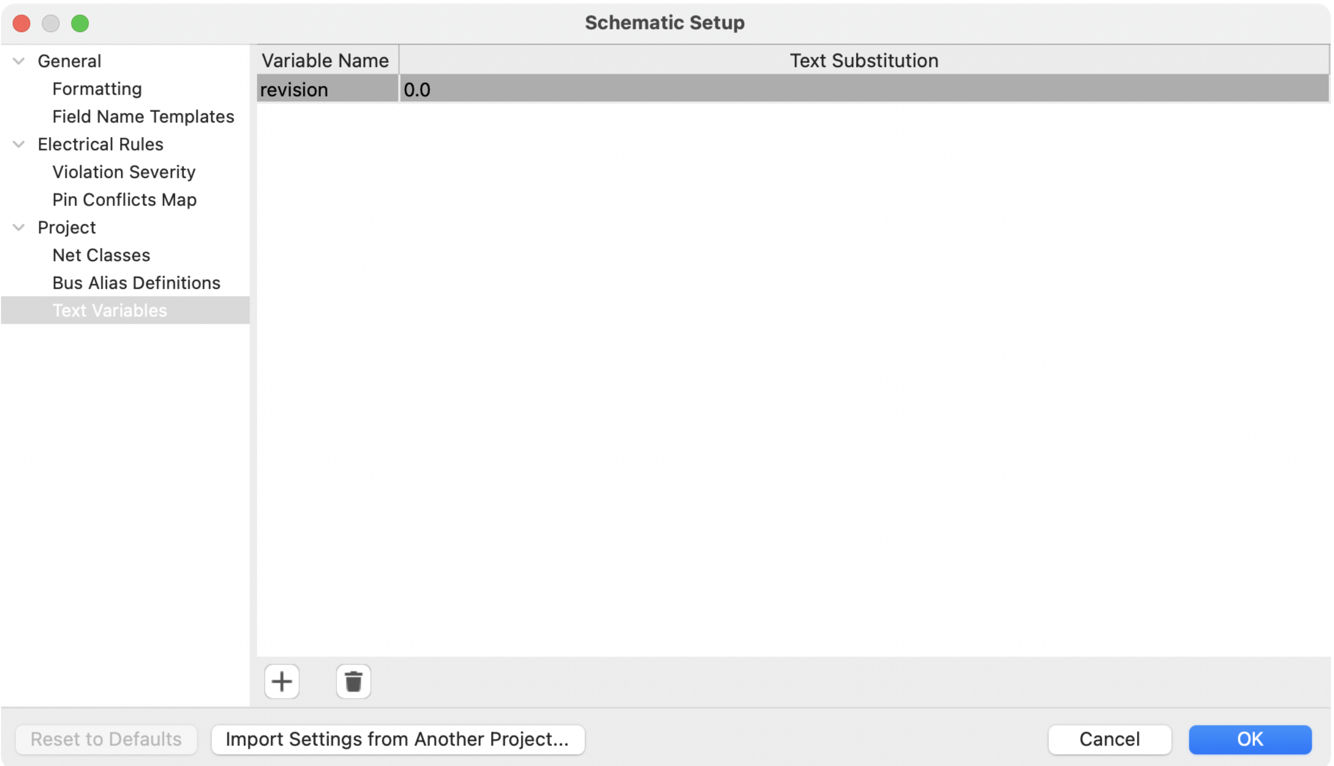
Pattern-based netclass assigment is explained in more detail in the net classes section.

Bus alias definitions



The **Bus Alias Definitions** panel allows you to create bus aliases, which are names for groups of signals in a bus. For more information about bus aliases, see the bus alias documentation.

Variabili di testo



Text replacement variables can be created in the Text Variables section. These variables allow you to substitute the variable name for any text string. This substitution happens anywhere the variable name is used inside the variable replacement syntax of `${VARIABLENAME}`.

For example, you could create a variable named `VERSION` and set the text substitution to `1.0`. Now, in any text object on the PCB, you can enter `${VERSION}` and KiCad will substitute `1.0`. If you change the substitution to `2.0`, every text object that includes `${VERSION}` will be updated automatically. You can also mix regular text and variables. For example, you can create a text object with the text `Version: ${VERSION}` which will be substituted as `Version: 1.0`.

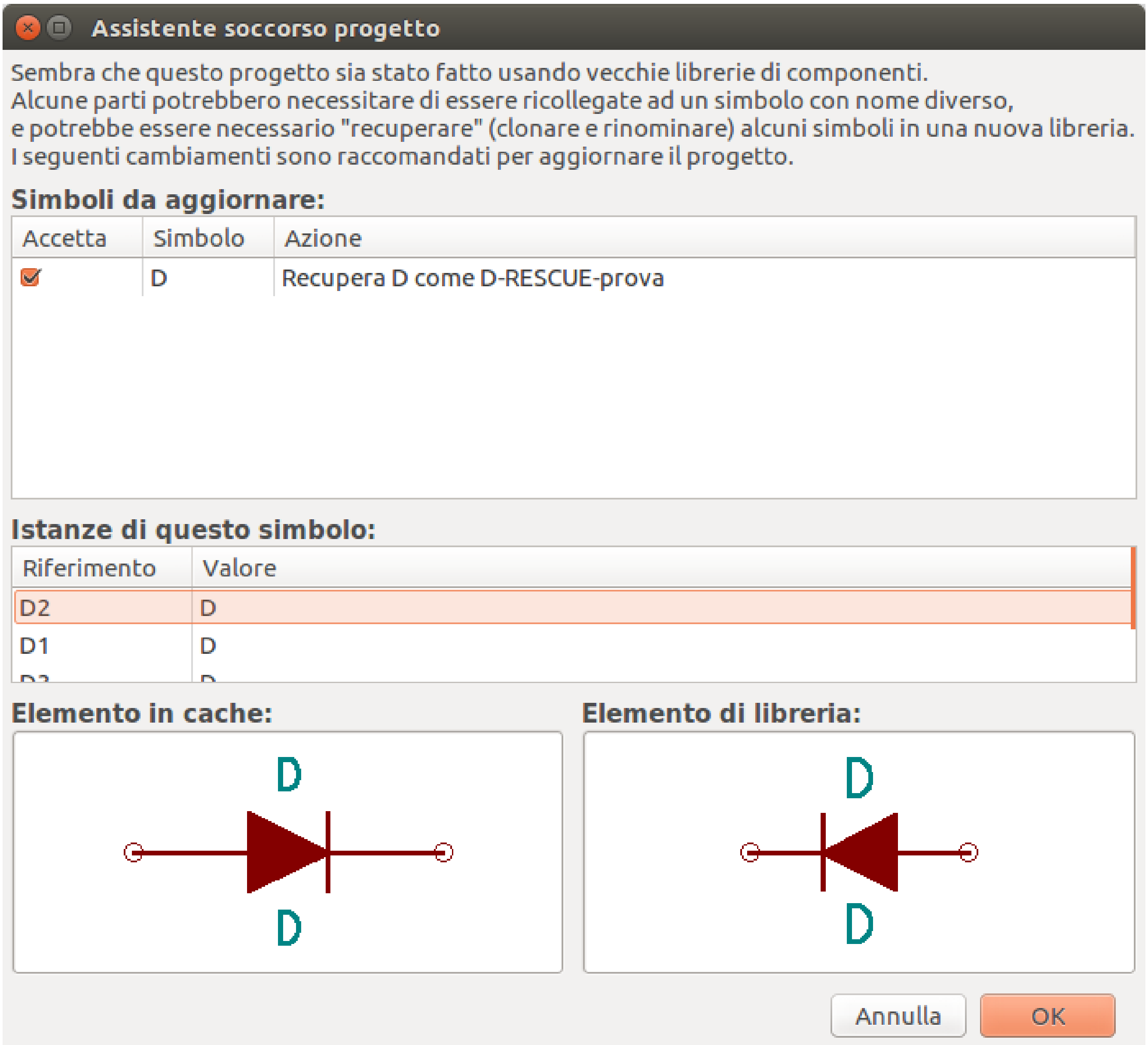
Text variables can also be created in Board Setup. Text variables are project-wide; variables created in the schematic editor are also available in the board editor, and vice versa.

There are also a number of built-in system text variables.

Recupero di simboli dalla cache

Come impostazione predefinita, KiCad carica i simboli dalle librerie del progetto, secondo le impostazioni dei percorsi e l'ordine delle librerie. Ciò può provocare dei problemi durante il caricamento di progetti molto vecchi: se i simboli nella libreria sono cambiati o sono stati rimossi o la libreria non esiste più da quando erano stati usati nel progetto, i simboli presenti nel progetto vengono automaticamente rimpiazzati dalle corrispondenti nuove versioni. Le nuove versioni potrebbero allinearsi correttamente o potrebbero essere orientati diversamente, generando così uno schema errato.

Quando un progetto viene salvato, viene salvata anche una libreria archivio (cache), contenente i simboli usati nel progetto, con esso. Ciò consente di distribuire il progetto senza tutte le librerie. Se si carica un progetto i cui simboli sono presenti sia nella libreria archivio (cache) che nelle le librerie di sistema, KiCad scansionerà le librerie per rilevare eventuali conflitti. Se vengono rilevati conflitti questi verranno elencati nella finestra di dialogo seguente:



Si può vedere in questo esempio che il progetto in origine aveva usato un diodo con il catodo verso l'alto, ma ora la libreria ne contiene uno con il catodo verso il basso. Questo cambiamento può rovinare il progetto! Premendo OK qui farà in modo di salvare il vecchio simbolo in una speciale libreria di `recupero`, e tutti i componenti che usano quel simbolo verranno rinominati per evitare conflitti di nome.

Se si preme Annulla, nessun recupero verrà effettuato, perciò KiCad caricherà i nuovi componenti come impostazione predefinita. Se si salva lo schema a questo punto, la cache verrà sovrascritta e i vecchi simboli non saranno più recuperabili. Se si è salvato lo schema, è ancora possibile tornare indietro ed eseguire ancora la funzione di recupero: scegliere "Recupera vecchi componenti" nel menu strumenti per richiamare nuovamente la finestra di dialogo.

Se si preferisce non visualizzare questa finestra di dialogo, è possibile premere ``Non mostrare più''. L'impostazione predefinita non farà nulla e permetterà di caricare i nuovi componenti. Questa opzione può essere ripristinata nelle preferenze delle librerie.

Schemi elettrici gerarchici

Introduzione

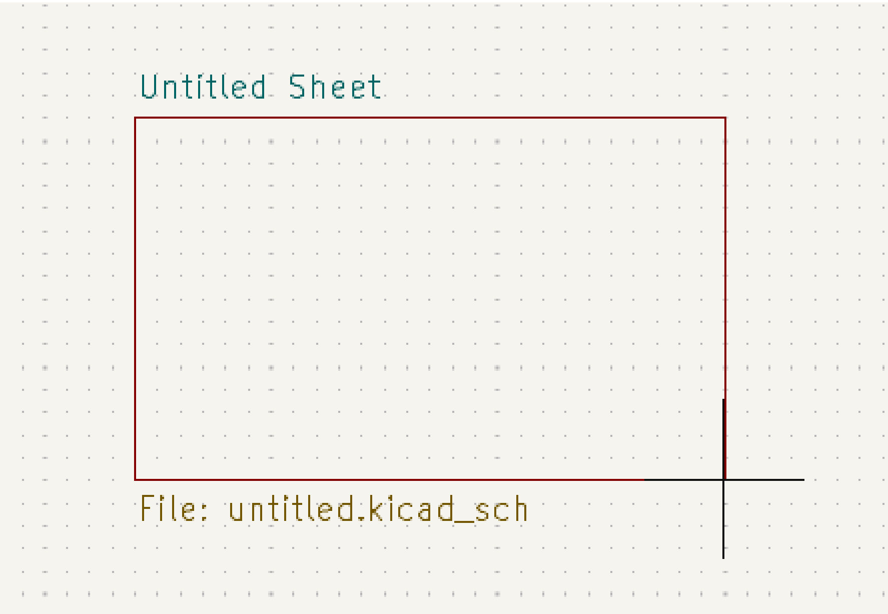
In KiCad, multi-sheet schematics are hierarchical: there is a single root sheet, and additional sheets are created as subsheets of either the root sheet or another subsheet. Sheets can be included in a hierarchy multiple times, if desired.

Carefully drawing a schematic as a hierarchical design improves schematic legibility and reduces repetitive drawing.

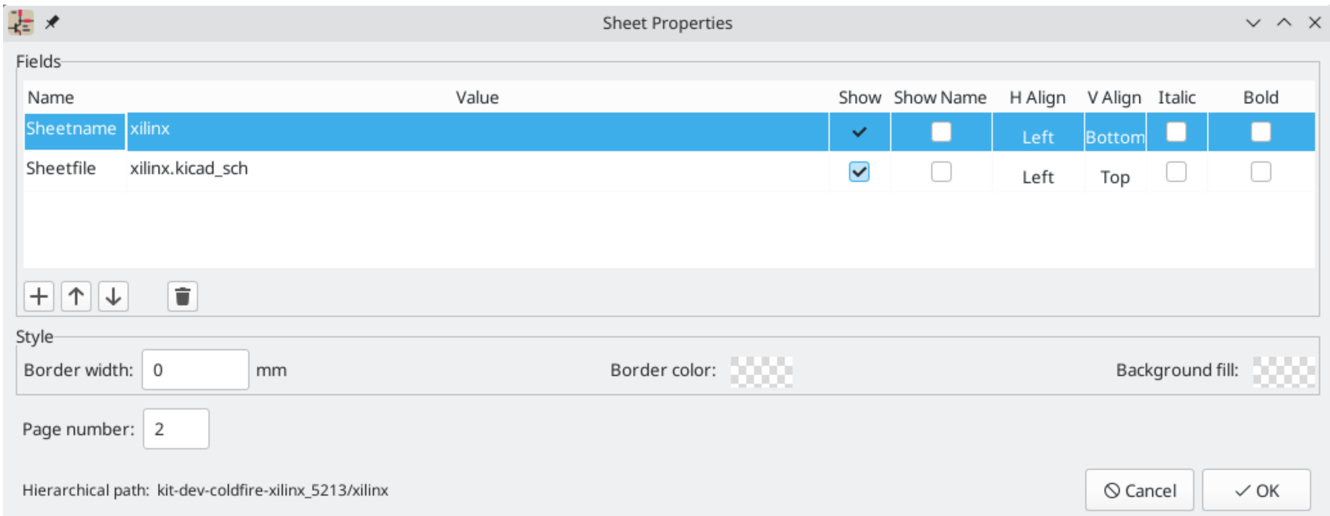
Creating a hierarchical schematic starts from the root sheet. The process is to create a subsheet, then draw the circuit in the subsheet and make the necessary electrical connections between sheets. Connections can be made between nets in a subsheet and nets in the parent sheet using hierarchical pins and labels, or between any two nets in the hierarchy using global labels.

Adding sheets to a design

You can add a subsheet to a design with the Add Hierarchical Sheet tool (S hotkey, or the  button in the right toolbar). Launch the tool, then click twice in the canvas to draw the upper left and lower right corners of the subsheet symbol. Make the sheet outline large enough to fit the hierarchical pins you will add later.



The Sheet Properties dialog will appear and prompt you for a sheet name and filename.



The **sheet name** must be unique, as it is used in the full net name for any nets in the subsheet. For example, a net with the local label `net1` in the sheet `sheet1` would have a full net name of `/sheet1/net1`. The sheet name is also used to refer to the sheet in various places in the GUI, including the title block and the hierarchy navigator.

The **sheet file** specifies the file that the new sheet will be saved to or loaded from. The path to the sheet file can be relative or absolute. It is usually preferable to save subsheet files in the project directory and use a relative path so that the project is portable.

A single sheet file can be used more than once in a project by specifying the same filename for each repeated sheet; the circuit drawn in the sheet will be instantiated once per usage, and any edits in once instance will be reflected in the other instances.

NOTE	Sheet files can be shared between multiple projects to allow design reuse between projects. However, this is not recommended due to path portability concerns and the risk of unintentionally changing other projects while editing a shared sheet.

The sheet's **page number** is configurable here. The page number is displayed in the sheet title block and the hierarchy navigator, and sheets are sorted by page number in the hierarchy navigator and when printing or plotting.

Several graphical options are also available. **Border width** sets the width of the border around the sheet shape. **Border color** and **Background fill** set the color for the border and fill of the sheet shape, respectively. If no color is set, a checkerboard swatch is shown and the default values from the color theme are used.

Sheets support arbitrary custom fields, which can be added and removed with the + and 🗑 buttons, respectively. Sheet fields can be optionally displayed on the schematic by checking their **Show** box, and they can be accessed from inside the sheet or in other sheet fields using text variables.

The Sheet Properties dialog can be accessed at any time by selecting a sheet symbol and using the `E` hotkey, or by right-clicking on a sheet symbol and selecting **Properties...**.

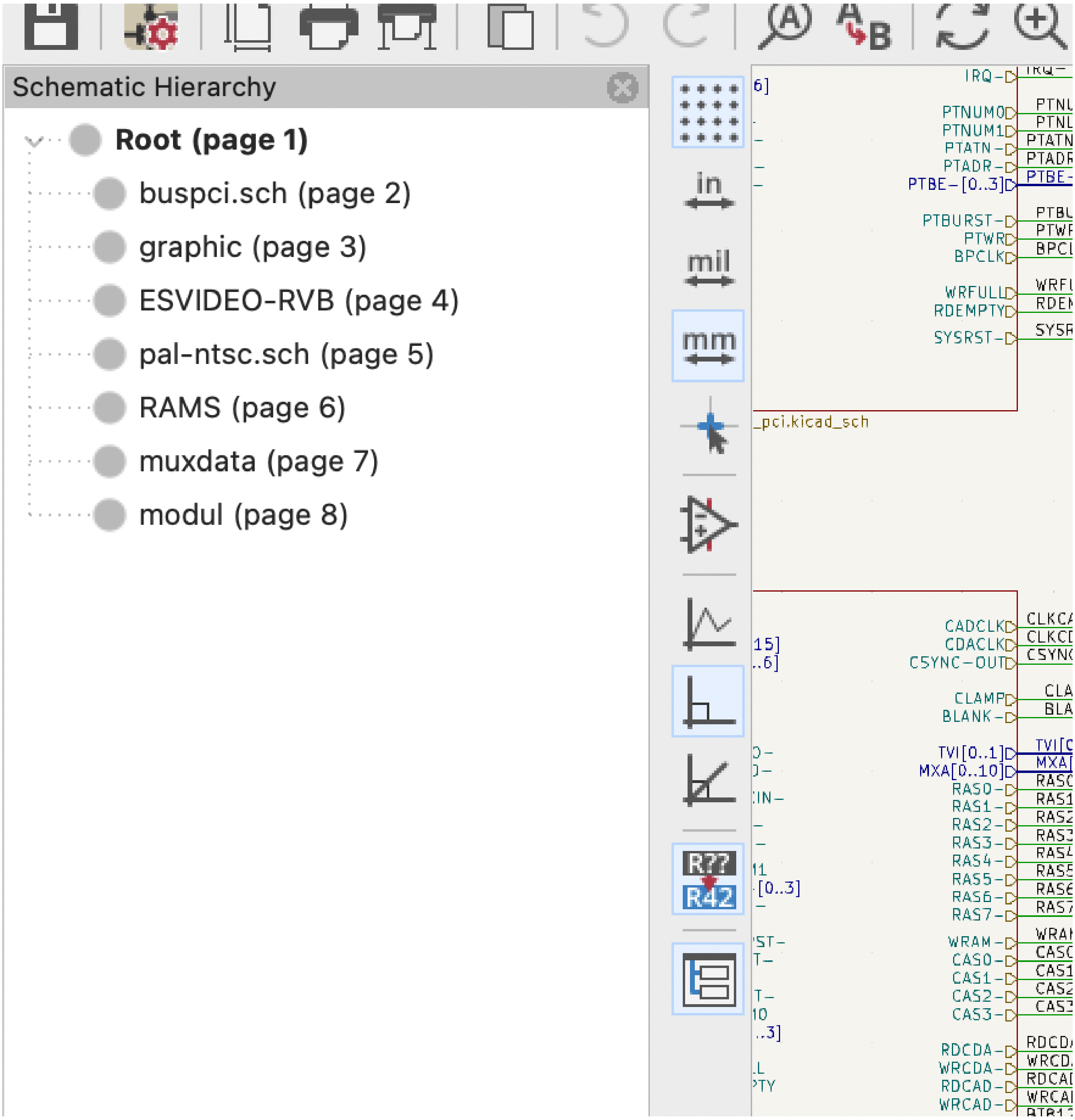
Navigating between sheets

You can enter a hierarchical sheet from the parent sheet by double-clicking the child sheet's shape, or right-clicking the child sheet and selecting **Enter Sheet**.

Return to the parent sheet by using the button in the top toolbar, or by right-clicking in an empty part of the schematic and clicking **Leave Sheet**.

You can jump to the next sheet with the button, or to the previous sheet with the button.

Alternatively, you can jump to any sheet with the hierarchy navigator. To open the hierarchy navigator, click the button in the left toolbar. The hierarchy navigator docks at the left of the screen. Each sheet in the design is displayed as an item in the tree. Clicking a sheet name opens that sheet in the editing canvas.



Connessioni elettriche tra fogli

Panoramica etichette

Electrical connections between sheets are made with labels. There are several kinds of labels in KiCad, each with a different connection scope.

- **Local labels** only make connections within a sheet. Therefore local labels cannot be used to connect between sheets. Local labels are added with the button.

- **Global labels** make connections anywhere in a schematic, regardless of sheet. Global labels are added with the button.

- Le **etichette gerarchiche** si connettono ai **pin del foglio gerarchico** accessibili nel foglio principale. I progetti gerarchici si basano su etichette e pin gerarchici per creare connessioni tra fogli padre e fogli figli; si può pensare ai pin gerarchici come mezzi per definire l'interfaccia per un foglio. Le etichette gerarchiche vengono aggiunte con il pulsante .

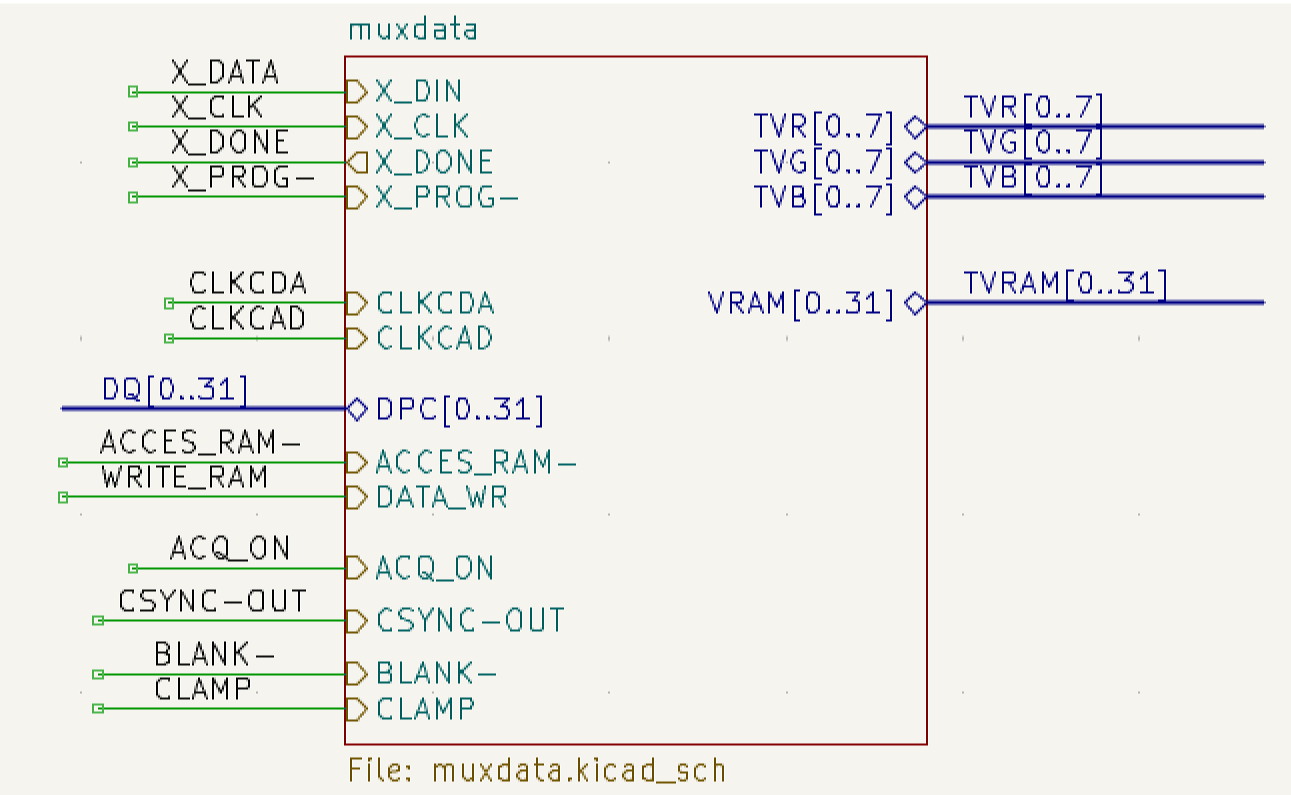
NOTE	Labels that have the same name will connect, regardless of the label type, if they are in the same sheet.

NOTE	Hidden power pins can also be considered global labels, because they connect anywhere in the schematic hierarchy.

Pin fogli gerarchici

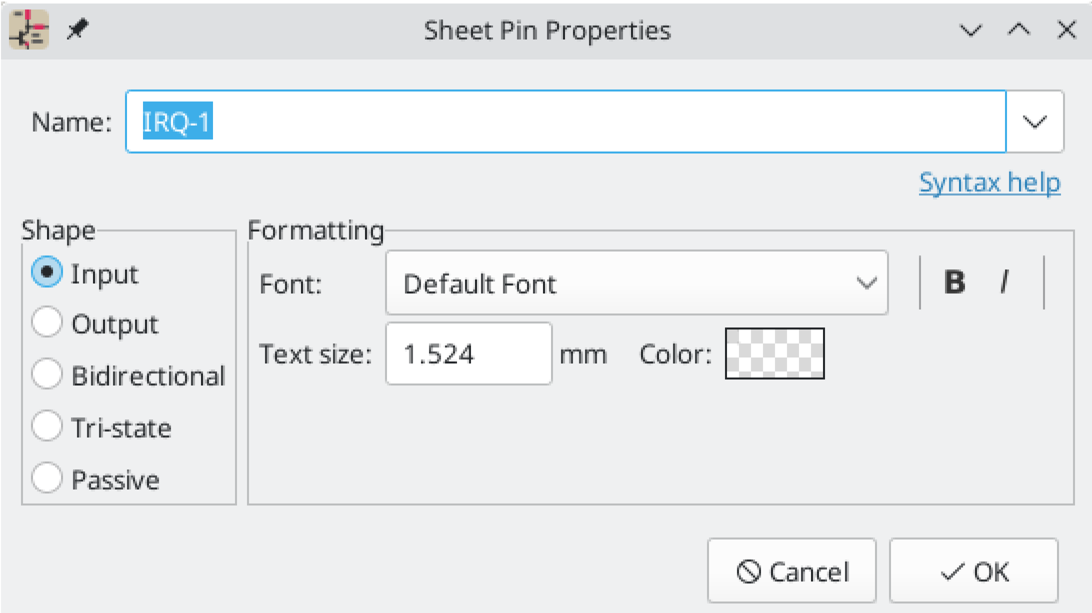
After placing hierarchical labels within the subsheet, matching **hierarchical pins** can be added to the subsheet symbol in the parent sheet. You can then make connections to the hierarchical pins with wires, labels, and buses. Hierarchical pins in a subsheet symbol are connected to the matching hierarchical labels in the subsheet itself.

NOTE	Hierarchical labels must be defined in the subsheet before the corresponding hierarchical sheet pin can be imported in the sheet symbol.



Per ogni etichetta gerarchica nel sottofoglio, importare il pin gerarchico corrispondente nel simbolo del foglio facendo clic sul pulsante nella barra degli strumenti a destra, quindi facendo clic sul simbolo del foglio. Un segnaposto per la prima etichetta gerarchica senza corrispondenza verrà attaccato al puntatore, dove può essere posizionato ovunque lungo il bordo del simbolo del foglio. Facendo nuovamente clic con lo strumento si continuerà ad importare ulteriori pin del foglio fino a quando non ci saranno più pin gerarchici da importare dal sottofoglio. I pin foglio possono essere importati anche selezionando **Importa pin fogli** nel menu contestuale del simbolo del foglio.

You can edit the properties of a sheet pin in the Sheet Pin Properties dialog. Open this dialog by double-clicking a sheet pin, selecting a sheet pin and using the `E` hotkey, or right-clicking a sheet pin and selecting **Properties...**.



The sheet pin's **name** can be edited in the textbox or by selecting from the dropdown list of hierarchical labels in the subsheet. A sheet pin's name has to match the corresponding hierarchical label in the subsheet, so if a pin name is changed the label must change as well.

Shape changes the shape of the sheet pin, and has no electrical effect. It can be set to Input, Output, Bidirectional, Tri-state, or Passive. The pin's **font**, **text size**, **color**, and emphasis (bold or italic) can also be changed.

Esempi di progettazioni gerarchiche

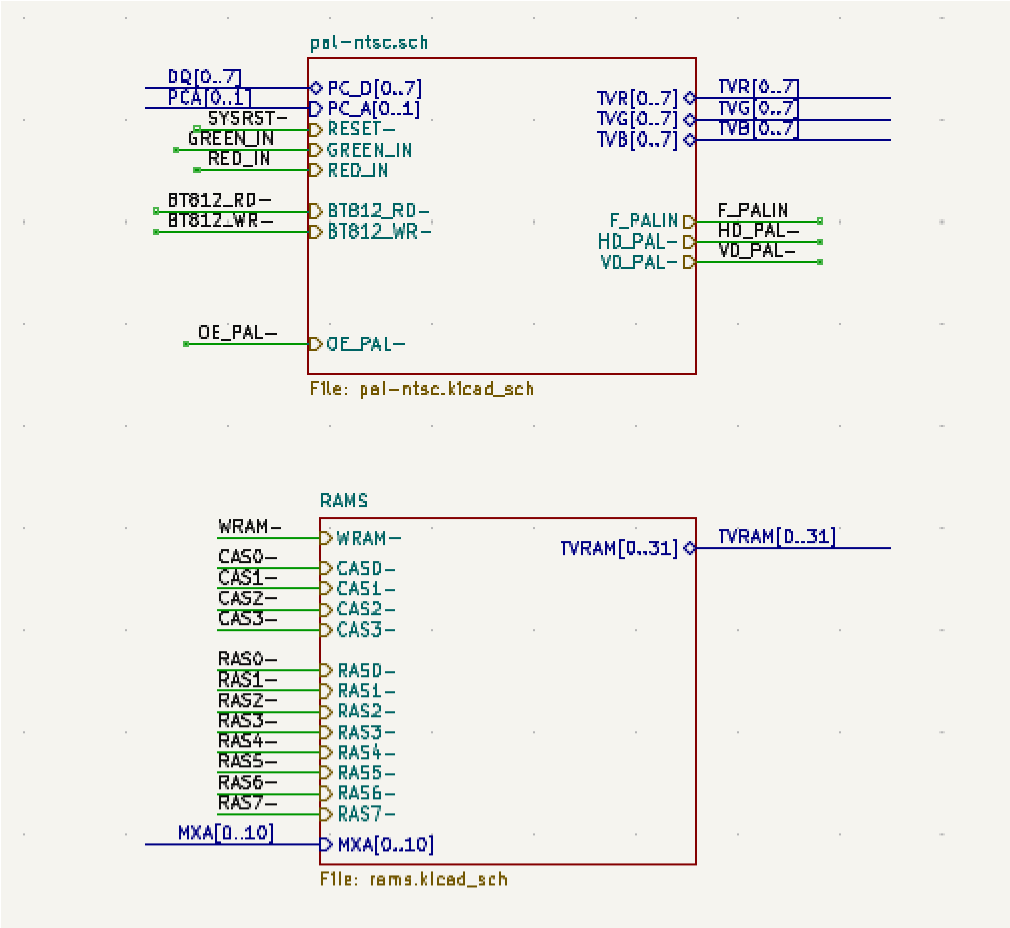
I disegni gerarchici possono essere classificati in diverse categorie:

- **Semplice:** ogni foglio viene usato solo una volta.
- **Complessa:** alcuni fogli sono istanziati più volte.
- **Piatta:** sottocaso di gerarchia **Semplice**, senza connessioni tra i sottofogli e il loro genitore. Le gerarchie piatte possono essere usate per rappresentare una progettazione non gerarchica.

Ogni modello gerarchico può essere utile; quello più appropriato dipende dal progetto.

Gerarchia semplice

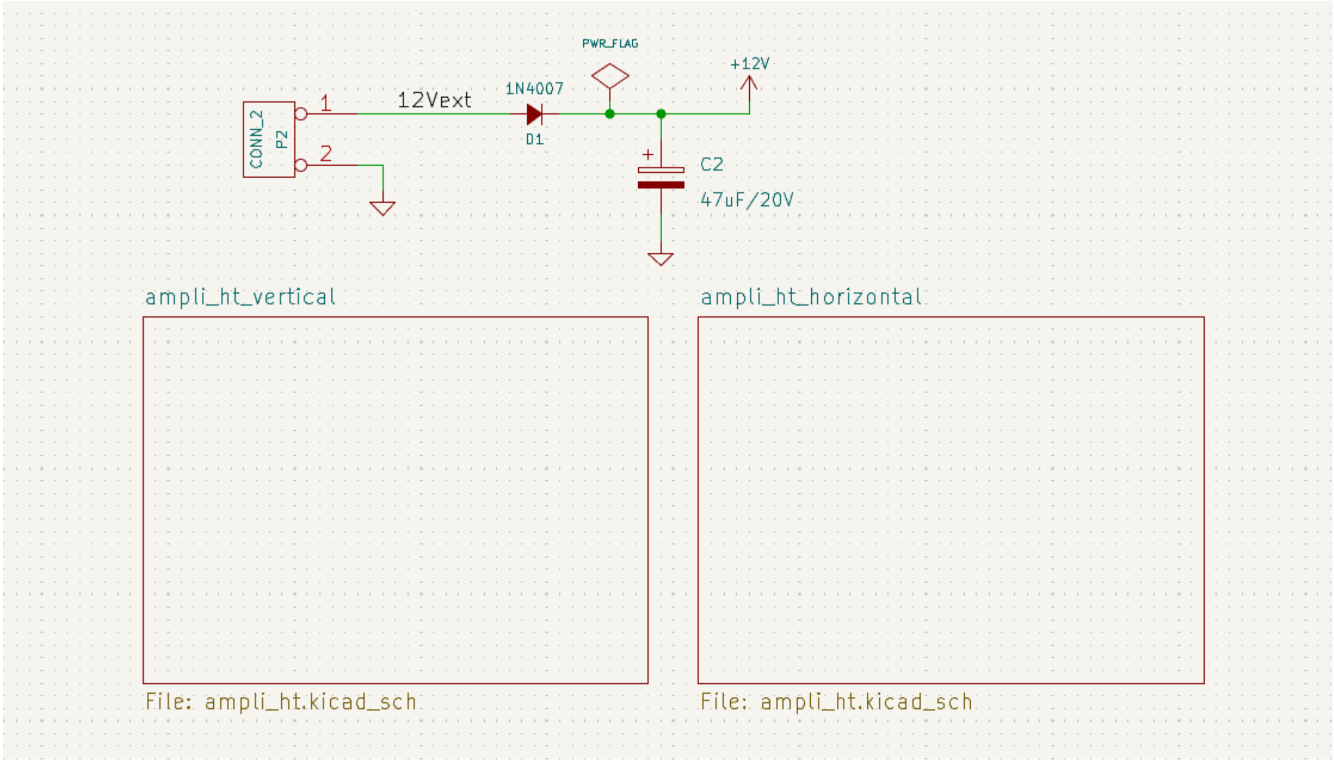
An example of a simple hierarchy is the `video` demo project included with KiCad. The root sheet contains seven unique subsheets, each with hierarchical labels and sheet pins linking the sheets to each other in the root sheet. Two of the subsheet symbols are shown below.



Gerarchia complessa

The `complex_hierarchy` demo project is an example of a complex hierarchy. The root sheet contains two subsheet symbols, which both refer to the same sheet file (`ampli_ht.kicad_sch`). This allows the design to include two copies of the same amplifier circuit. Although the two sheet symbols refer to the same filename, the sheet names are unique (`ampli_ht_vertical` and `ampli_ht_horizontal`). Inside each subsheet the circuits are identical except for the reference designators, which as always are unique.

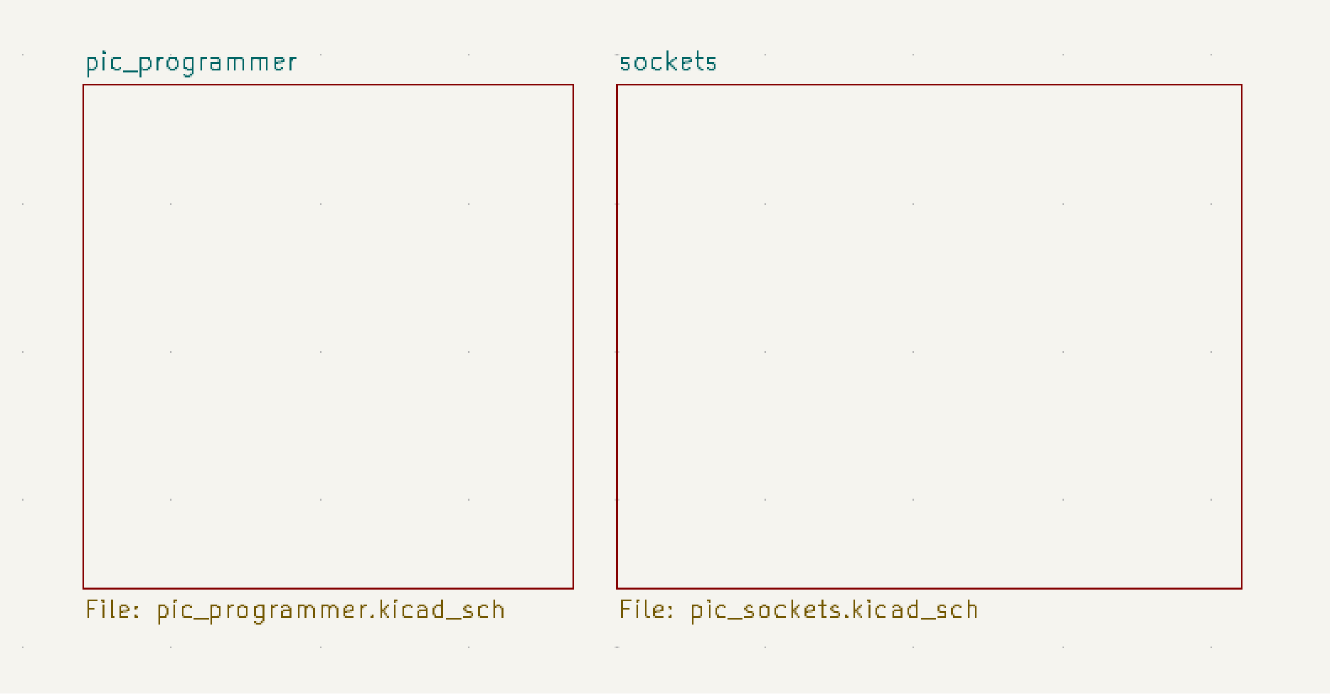
This project contains no sheet pin connections. The only connections between the root sheet and the subsheets are global power connections made with power symbols. However, sheets in a complex hierarchy could include sheet pin connections if appropriate for the design.



Gerarchia piatta

The `flat_hierarchy` demo project is an example of a flat hierarchy. The root sheet contains two unique subsheet symbols with no hierarchical sheet pins. The root sheet in this project does nothing except hold the subsheets, and the subsheets are used only as additional pages in the schematic.

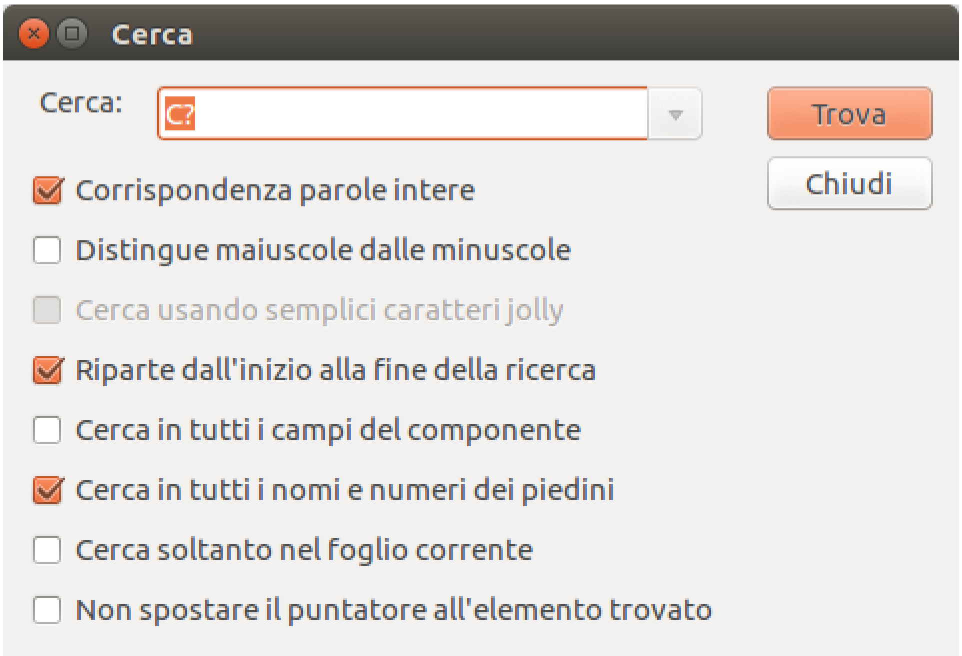
NOTE	This is the simplest way to create multi-page schematics in KiCad.



Ispezione di uno schema

Strumento trova

The Find tool searches for text in the schematic, including reference designators, pin names, symbol 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.



Lo strumento trova ha diverse opzioni:

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 schematic. When unselected, the search will match if the search term is part of a larger word in the schematic.

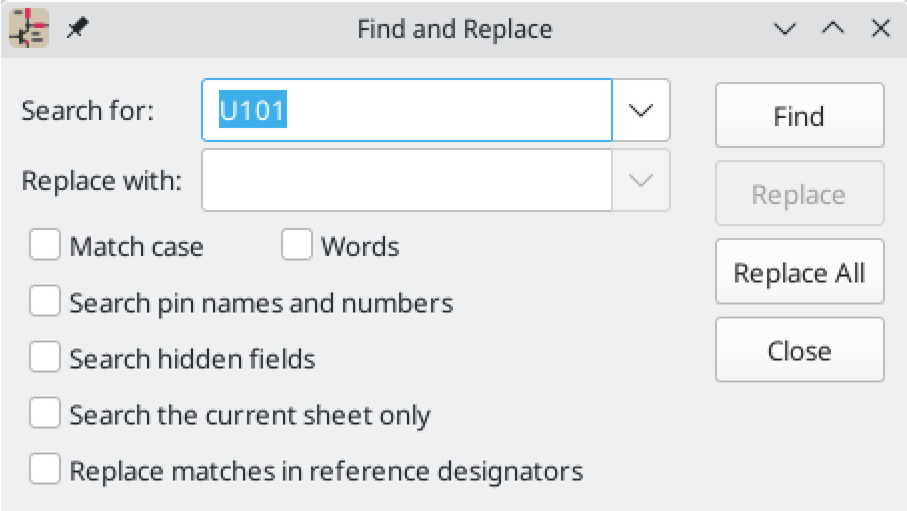
Wildcards: When selected, wildcards can be used in the search terms. `?` matches any single character, and `*` matches any number of characters. Note that when this option is selected, partial matches are not returned: searching for `abc*` will match the string `abcd`, but searching for `abc` will not.

Search pin names and numbers: Selects whether the search should apply to pin names and numbers.

Search hidden fields: Selects whether the search should apply only to visible fields or if it should include hidden symbol fields.

Search the current sheet only: Selects whether the search should be limited to the current schematic sheet or to the entire schematic.

There is also a Find and Replace tool which is activated with the button in the top toolbar. This tool behaves the same as the Find tool, but additionally can replace some or all matches with different text.



If the **Replace matches in reference designators** option is checked, reference designators will be modified if they contain matching text. Otherwise reference designators will not be affected.

Evidenziazione net

An electrical net can be highlighted in the schematic editor to visualize all of the places it appears in the schematic. Net highlighting can be activated in the Schematic Editor or by highlighting the corresponding net in the PCB editor when cross-probe highlighting is enabled (see below). When net highlighting is active, the highlighted net will be shown in a different color. By default this color is pink, but it is configurable in the Color section of the Preferences dialog.

Nets can be highlighted by clicking on a wire or pin using the Highlight Net tool in the right toolbar ( ). Alternatively, the Highlight Net hotkey (`) highlights the net under the cursor.

Net highlighting can be cleared by using the Clear Net Highlight action (hotkey ~) or by using the Highlight net tool on an empty region in the schematic. By default, Esc also clears net highlighting, but this can be disabled if desired in **Preferences** → **Schematic Editor** → **Editing Options**.

Cross-probing from the PCB

KiCad allows bi-directional cross-probing between the schematic and the PCB. There are several different types of cross-probing.

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.

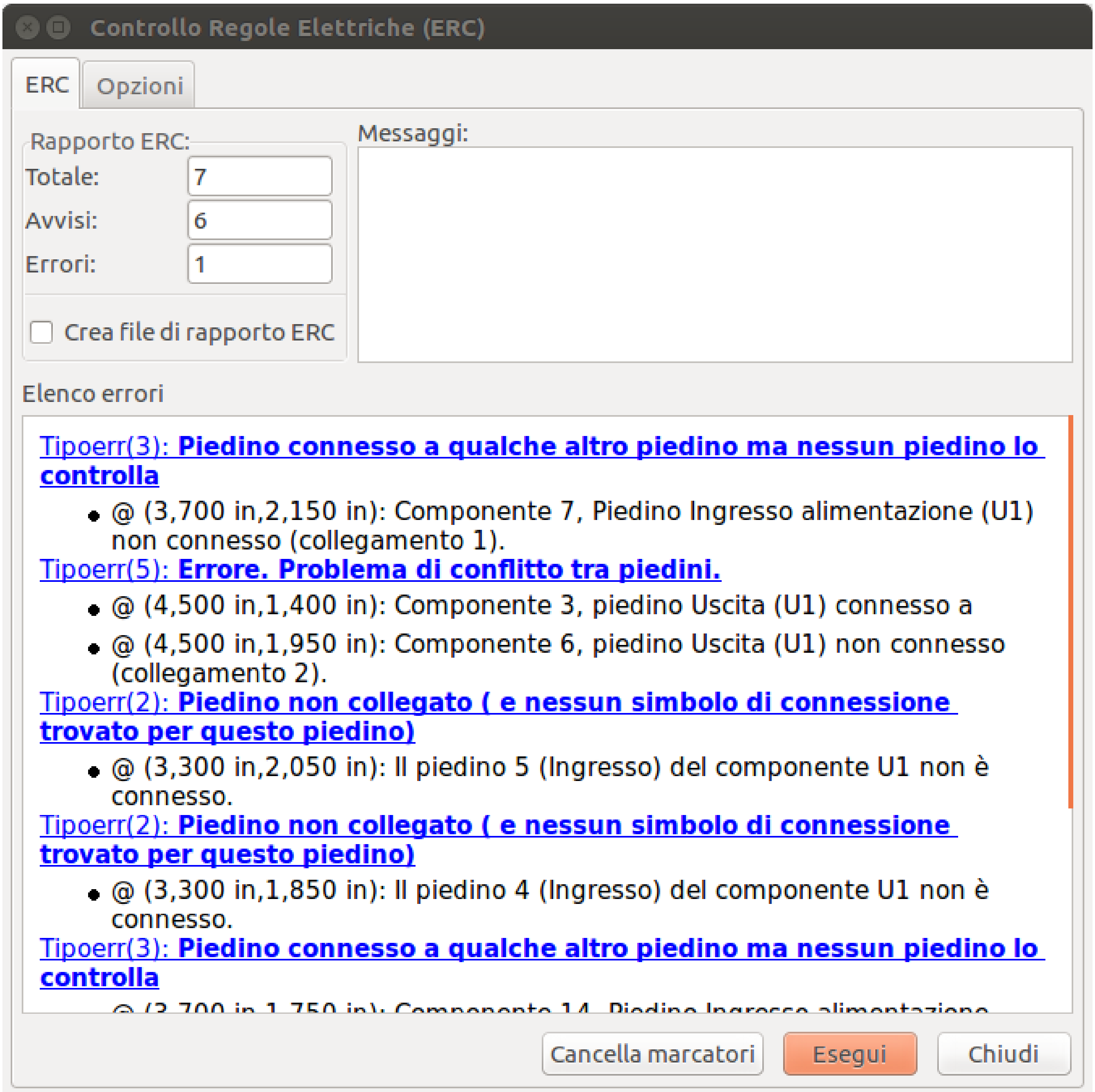
Highlight cross-probing allows you to highlight a net in the schematic and PCB at the same time. If the option "Highlight cross-probed nets" is enabled in the Display Options section of the Preferences dialog, highlighting a net or bus in the schematic editor will cause the corresponding net or nets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

Controllo Regole Elettriche (ERC)

Lo strumento per il controllo regole elettriche (ERC) controllo la presenza di certi errori presenti nello schema, come piedini sconnessi, simboli gerarchici sconnessi, uscite in corto-circuito, o altri tipi di connessioni non consentite, ecc. Le violazioni vengono segnalate come errori o avvisi a seconda della gravità del problema rilevato.

ERC è imperfetto e non è in grado di rilevare tutti gli errori, ma può rilevare molti problemi e sviste comuni. Tutti i problemi rilevati devono essere controllati e risolti prima di procedere. La qualità dell'ERC è direttamente correlata alla cura posta nel dichiarare le proprietà elettriche del pin durante la creazione del simbolo. Se i simboli sono progettati in modo errato, ERC non riporterà informazioni accurate.

l'ERC può essere avviato facendo clic sul pulsante  nella barra strumenti in cima e facendo clic sul pulsante **Esegui ERC**.

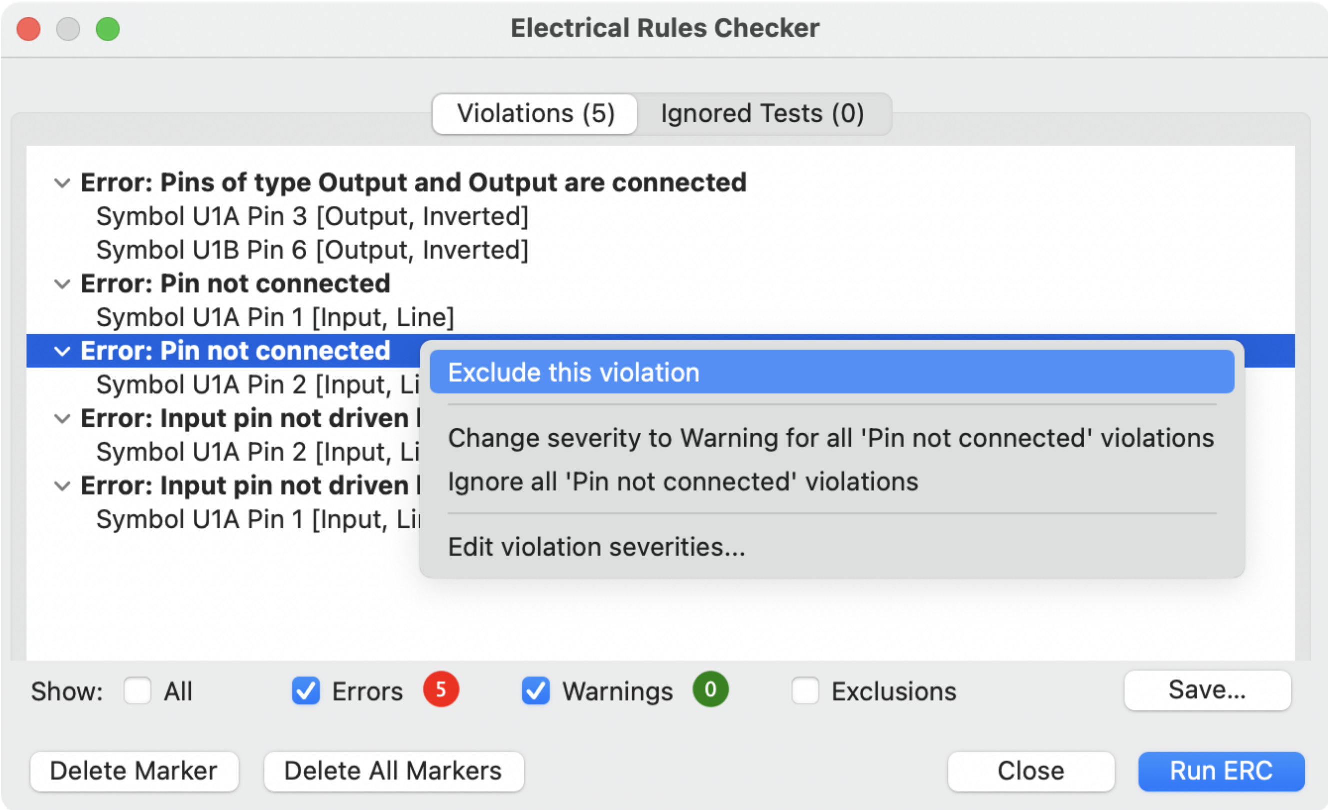


Any warnings or errors are reported in the **Violations** tab, and markers for each violation are placed in the schematic so that they point to the relevant part of the schematic. Warnings are indicated by yellow arrows, and errors have red arrows. Excluded violations are shown as green arrows.

NOTE Selezionando una violazione nella finestra ERC si passa al marcatore di violazione selezionato nello schema.

The numbers at the bottom of the window show the number of errors, warnings, and exclusions. Each type of violation can be filtered from the list using the respective checkboxes. Clicking **Delete Markers** will clear all violations until ERC is run again.

Violations can be right-clicked in the dialog to ignore them or change their severity:

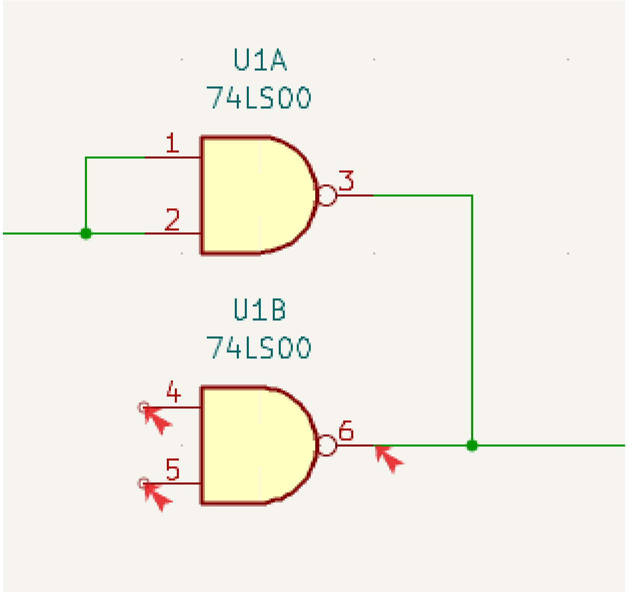


- **Exclude this violation:** ignores this particular violation, but does not affect any other violations.
- **Change severity:** changes a type of violation from warning to error, or error to warning. This affects all violations of a given type.
- **Ignore all:** ignores all violations of a given type. This test will now appear in the **Ignored Tests** tab rather than the **Violations** tab.

You can also exclude the selected marker with **Inspect** → **Exclude Marker**, and show or hide each category of marker (errors, warnings, and exclusions) with the **View** menu.

Excluded and ignored violations are remembered between runs of the design rule checker.

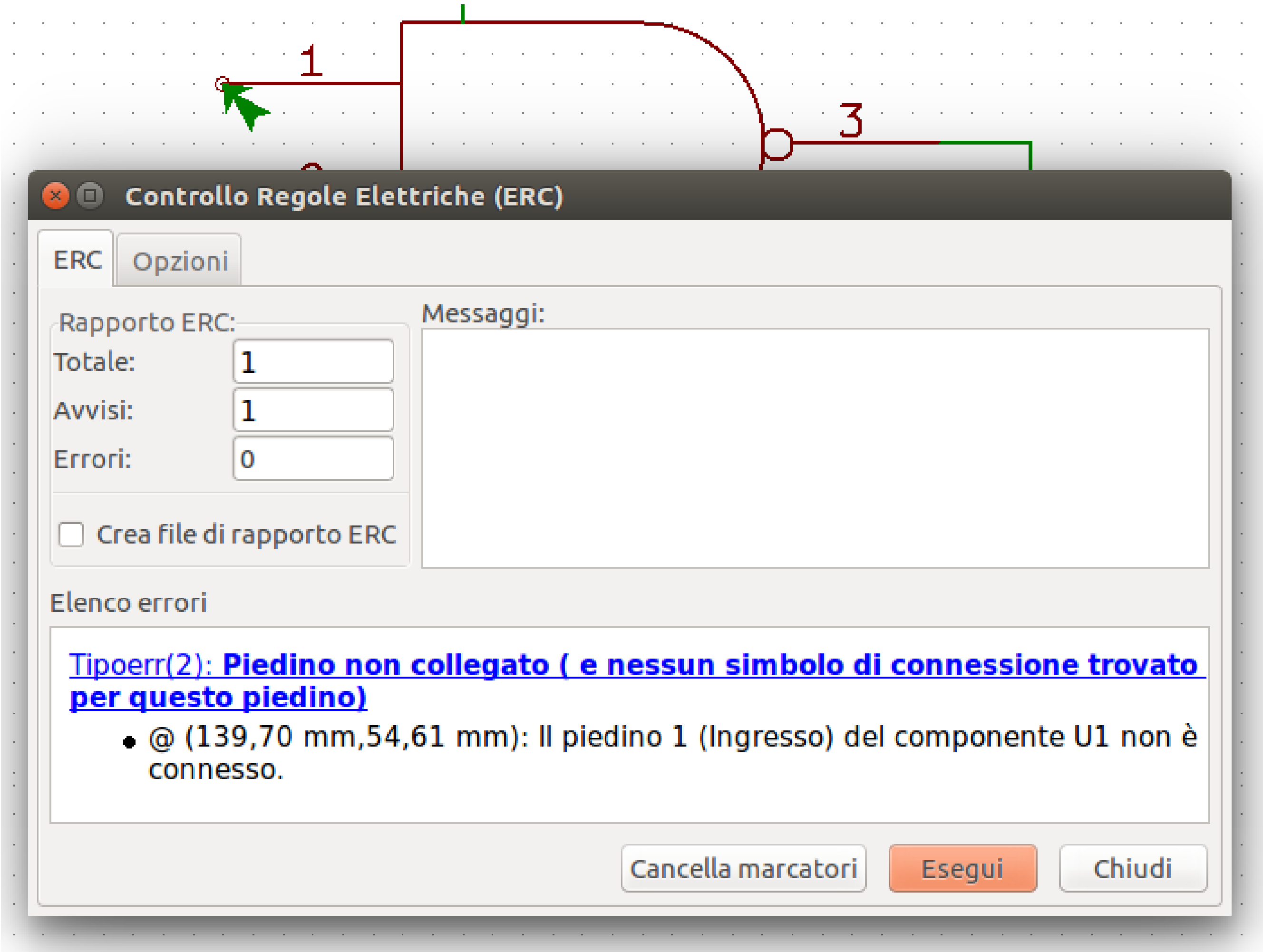
Esempio di ERC



There are three errors in the screenshot above.

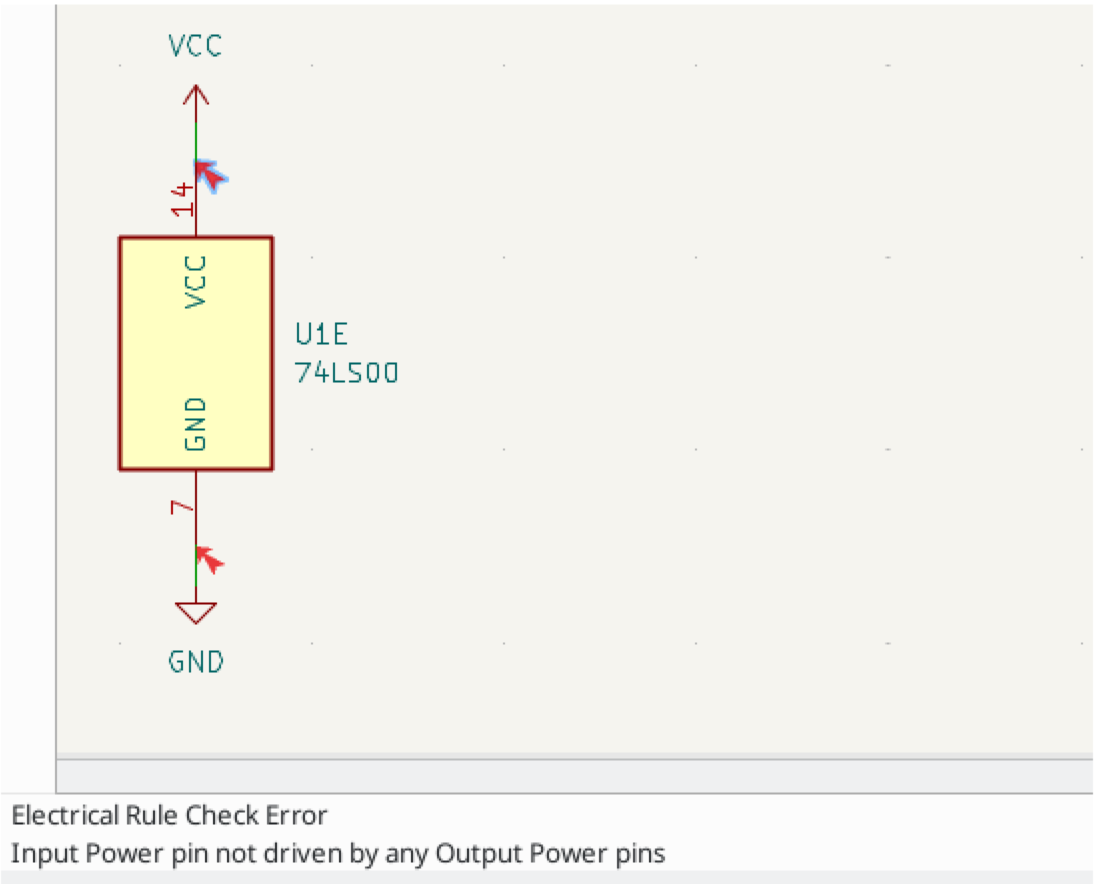
- Due uscite sono state collegate assieme (freccia rossa sulla destra).
- Two inputs have been left unconnected (red arrows at left). This is actually two errors per pin: each pin is unconnected, and each pin is an input pin that is not driven by an output pin.

Selecting an ERC marker displays a description of the violation in the message pane at the bottom of the window.

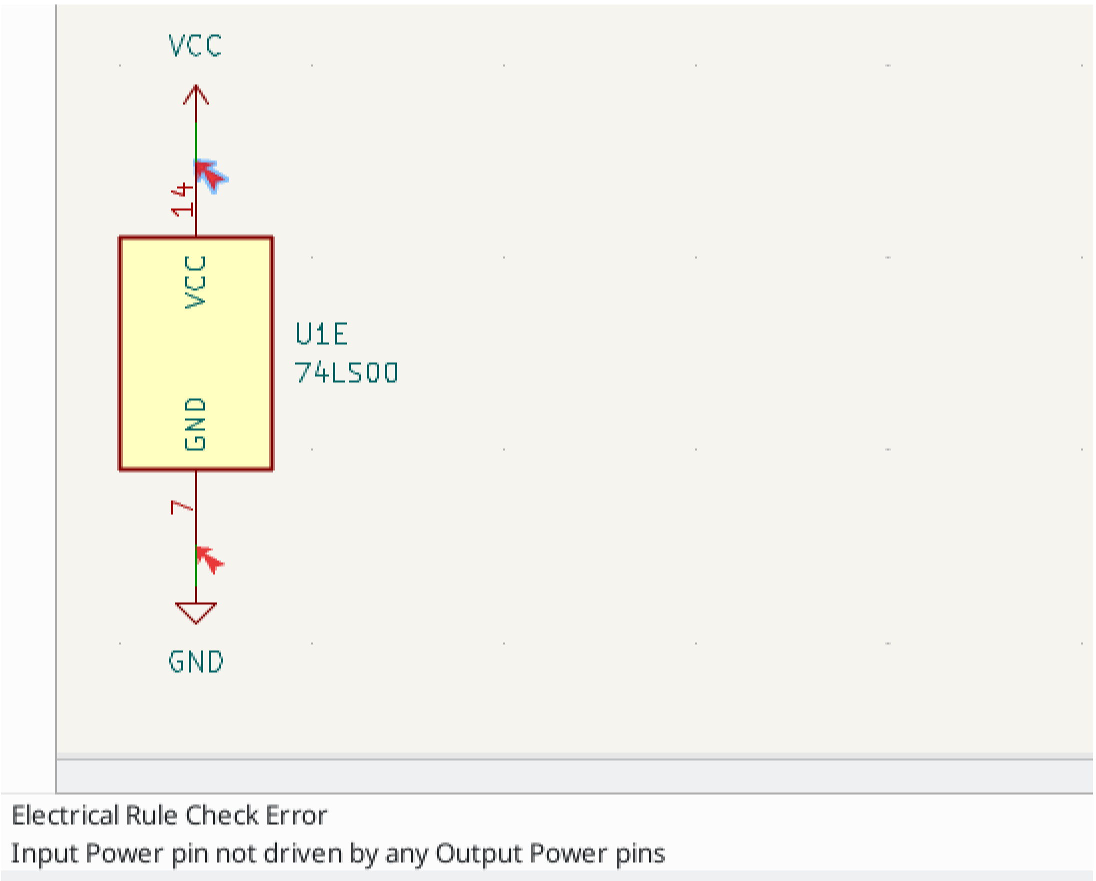


Piedini e segnalazioni di potenza

It is common to have an "Input Power pin not driven by any Output Power pins" error on power pins, as shown in the example below, even though the power pins seem to be properly connected to a power rail. This happens in designs where the power is provided through connectors or other components that are not marked as power outputs. In these cases ERC won't detect any Output Power pins connected to the net and will determine the Input Power pin is not driven by a power source.



To avoid this warning, connect the net to `PWR_FLAG` symbol on such a power net as shown in the following example. The `PWR_FLAG` symbol is found in the `power` symbol library. Alternatively, connect any power output pin to the net; `PWR_FLAG` is simply a symbol with a single power output pin.

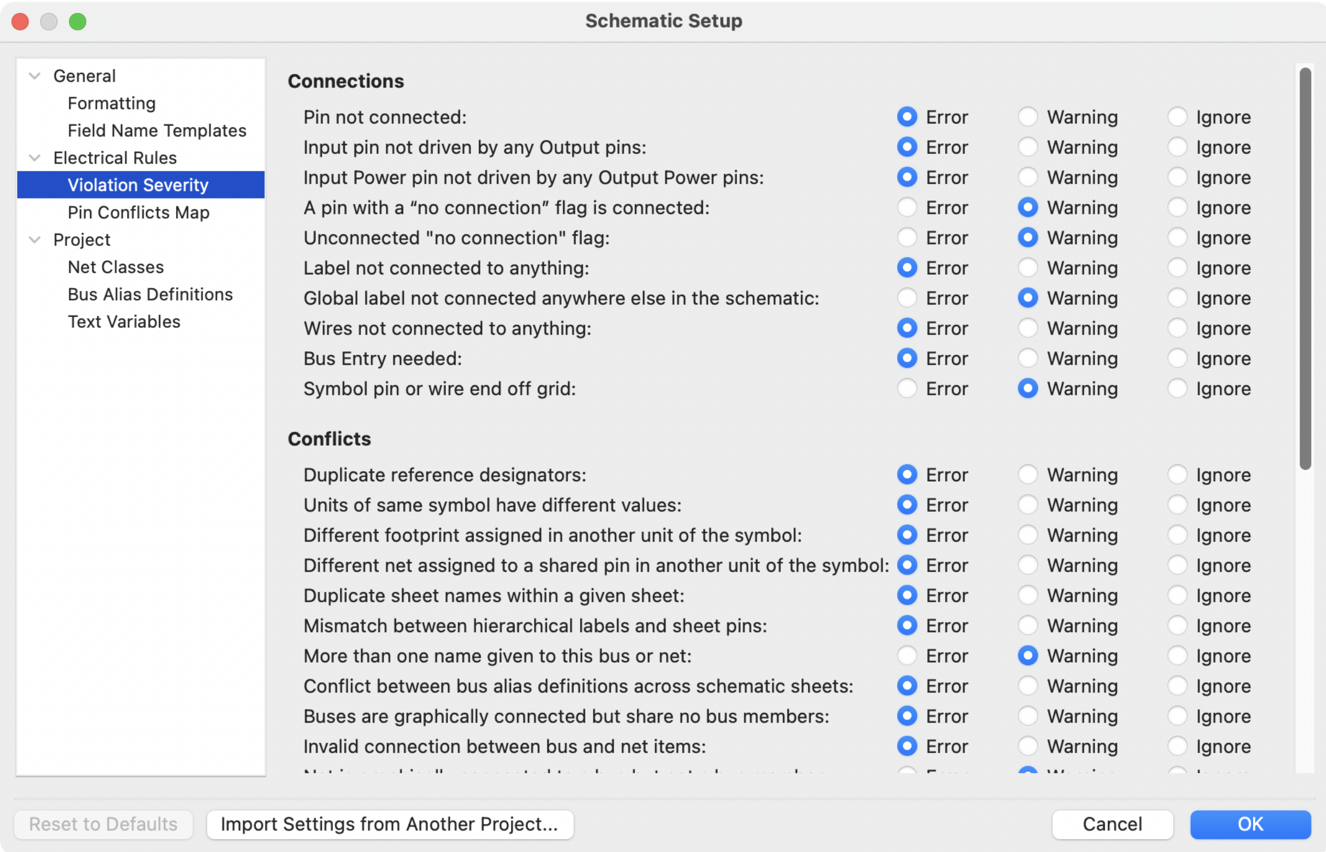


Le reti di massa spesso necessitano anche di un `PWR_FLAG`, poiché i regolatori di tensione hanno uscite dichiarate come uscite di potenza, ma i loro pin di massa sono tipicamente contrassegnati come ingressi di alimentazione. Pertanto le masse possono apparire non collegate a una fonte a meno che non venga utilizzato un simbolo `PWR_FLAG`.

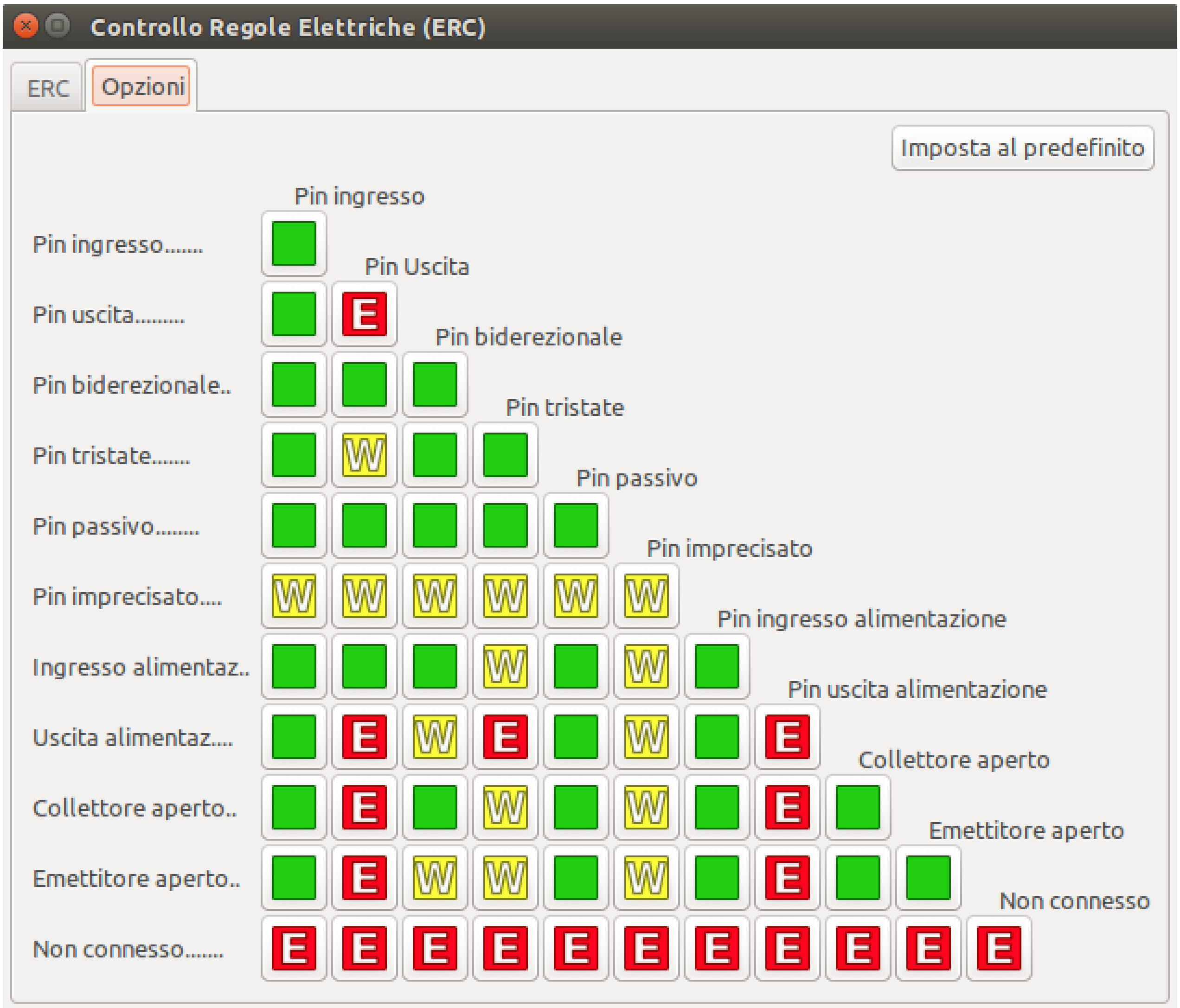
For more information about power pins and power flags, see the `PWR_FLAG` documentation.

Configurazione ERC

Il pannello **Importanza violazione** nella Impostazione dello schema permette di configurare che tipo di messaggi ERC devono essere riportati come errori, come semplici avvertimenti o ignorati.



Il pannello **Mappa conflitti pin** nelle Impostazioni dello schema, permette di configurare le regole di connessione per definire le condizioni elettriche per il controllo di errori e avvertimenti in base al tipo di pin connesso uno con l’altro. Ad esempio, per impostazione predefinita viene prodotto un errore quando un pin di uscita è collegato a un altro pin di uscita.



Le regole si possono cambiare facendo clic sul riquadro desiderato della matrice, in modo ciclare la scelta desiderata: normale, avvertimento, errore.

Elenco controlli ERC