

# Schematic Editor

The KiCad Team

# Table of Contents

Introduction to the KiCad Schematic Editor .....	2
Description .....	2
Aperçu technique .....	2
Generic Schematic Editor commands .....	3
Commandes à la souris .....	4
Raccourcis clavier .....	4
Grid .....	8
Snapping .....	9
Sélection du Zoom .....	9
Affichage des coordonnées du curseur .....	9
Barre de menu .....	10
Barre d'outils supérieure .....	10
Barre d'outils latérale droite .....	11
Barre d'outils latérale gauche .....	12
Menus contextuels et édition rapide .....	13
Barre de menus .....	14
Menu Fichiers .....	14
Menu Préférences .....	16
Menu Aide .....	22
Barre d'outils principale .....	23
Gestion des feuilles schématiques .....	23
Outil de recherche .....	23
Outil de Netliste .....	24
Outil d'annotation .....	25
Outil de vérification des règles électriques .....	27
Outil de Liste de Matériel .....	29
Edit Fields tool .....	31
Import tool for footprint assignment .....	33
Manage Symbol Libraries .....	34
Symbol Library Table .....	34
Création et édition de schémas .....	39
Introduction .....	39
Généralités .....	39
Symbol placement and editing .....	39
Electrical Connections .....	43
Compléments Graphiques .....	51
Rescuing cached symbols .....	53
Schématiques hiérarchiques .....	55
Introduction .....	55
Navigation dans la hiérarchie .....	55
Labels locaux, hiérarchiques et globaux .....	56
Summary of hierarchy creation .....	56

Symbole de feuille hiérarchique .....	56
Connexions - Pins hiérarchiques .....	57
Connexions - Labels hiérarchiques .....	57
Hiérarchie complexe .....	59
Hiérarchie à plat .....	60
Symbol Annotation Tool .....	63
Introduction .....	63
Quelques exemples .....	64
Vérification des règles électriques (ERC) .....	67
Introduction .....	67
Utilisation de l'ERC .....	67
Exemple d'ERC .....	68
Affichage du diagnostic .....	69
Pins d'alimentation et symboles d'alimentation (Power Flag) .....	69
Configuration .....	70
Fichier de rapport d'ERC .....	71
Transfer Schematic to PCB .....	72
Généralités .....	72
Options .....	72
Tracer / Imprimer .....	74
Introduction .....	74
Commandes de tracé communes .....	74
Tracer en Postscript .....	74
Tracer en PDF .....	75
Tracer en SVG .....	76
Tracer en DXF .....	76
Tracer en HPGL .....	76
Imprimer sur papier .....	78
Symbol Editor .....	79
General Information About Symbol Libraries .....	79
Symbol Library Overview .....	79
Symbol Library Editor Overview .....	79
Sélection et gestion des bibliothèques .....	83
Creating Library Symbols .....	83
Éléments graphiques .....	90
Multiple Units per Symbol and Alternate Body Styles .....	92
Création et édition de pins .....	95
Symbol Fields .....	102
Power Ports .....	103
Symbol Library Browser .....	108
Introduction .....	108
Viewlib - fenêtre principale .....	109
Symbol Library Browser Top Toolbar .....	109
Création d'une Netliste .....	111

Généralités .....	111
Formats de Netliste .....	111
Exemples de netlistes .....	114
Notes sur les netlistes .....	116
Autres formats .....	116
Création de Netlistes et BOM personnalisés .....	119
Fichier intermédiaire de Netliste .....	119
Conversion dans un nouveau format de netliste .....	121
L'approche XSLT .....	121
Exemples de lignes de commandes pour les scripts Python .....	130
Structure du fichier de netliste intermédiaire .....	130
Complément sur xsltproc .....	135
Simulator .....	139
Assigning models .....	139
Spice directives .....	144
Simulation .....	144

## Manuel de référence

### NOTE

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

### Copyright

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

Toutes les marques apparaissant dans ce document appartiennent à leurs propriétaires respectifs.

### Contributeurs

Jean-Pierre Charras, Fabrizio Tappero, Graham Keeth

### Traduction

Marc Berlioux <[marc.berlioux@gmail.com](mailto:marc.berlioux@gmail.com)>, 2015-2016

### Retours

Merci de signaler vos corrections de bugs, suggestions ou nouvelles versions ici :

- About KiCad documentation: <https://gitlab.com/kicad/services/kicad-doc/issues>
- Bugs logiciel KiCad : <https://gitlab.com/kicad/code/kicad/issues>

# Introduction to the KiCad Schematic Editor

## Description

The KiCad Schematic Editor is a schematic capture software distributed as a part of KiCad and available under the following operating systems:

- Linux
- Apple macOS
- Windows

Regardless of the OS, all KiCad files are 100% compatible from one OS to another.

The Schematic Editor is an integrated application where all functions of drawing, control, layout, library management and access to the PCB design software are carried out within the editor itself.

The KiCad Schematic Editor is intended to cooperate with the KiCad PCB Editor, which is KiCad's printed circuit design software. It can also export netlist files, which lists all the electrical connections, for other packages.

The Schematic Editor includes a symbol library editor, which can create and edit symbols and manage libraries. It also integrates the following additional but essential functions needed for modern schematic capture software:

- La vérification des règles électriques ou ERC (Electrical Rules Check), pour le contrôle des connexions manquantes ou incorrectes.
- L'exportation de fichiers de tracé en plusieurs formats (Postscript, PDF, HPGL, SVG)
- Bill of Materials generation (via Python or XSLT scripts, which allow many flexible formats).

## Aperçu technique

The Schematic Editor is limited only by the available memory. There is thus no real limitation to the number of components, component pins, connections or sheets. In the case of multi-sheet schematics, the representation is hierarchical.

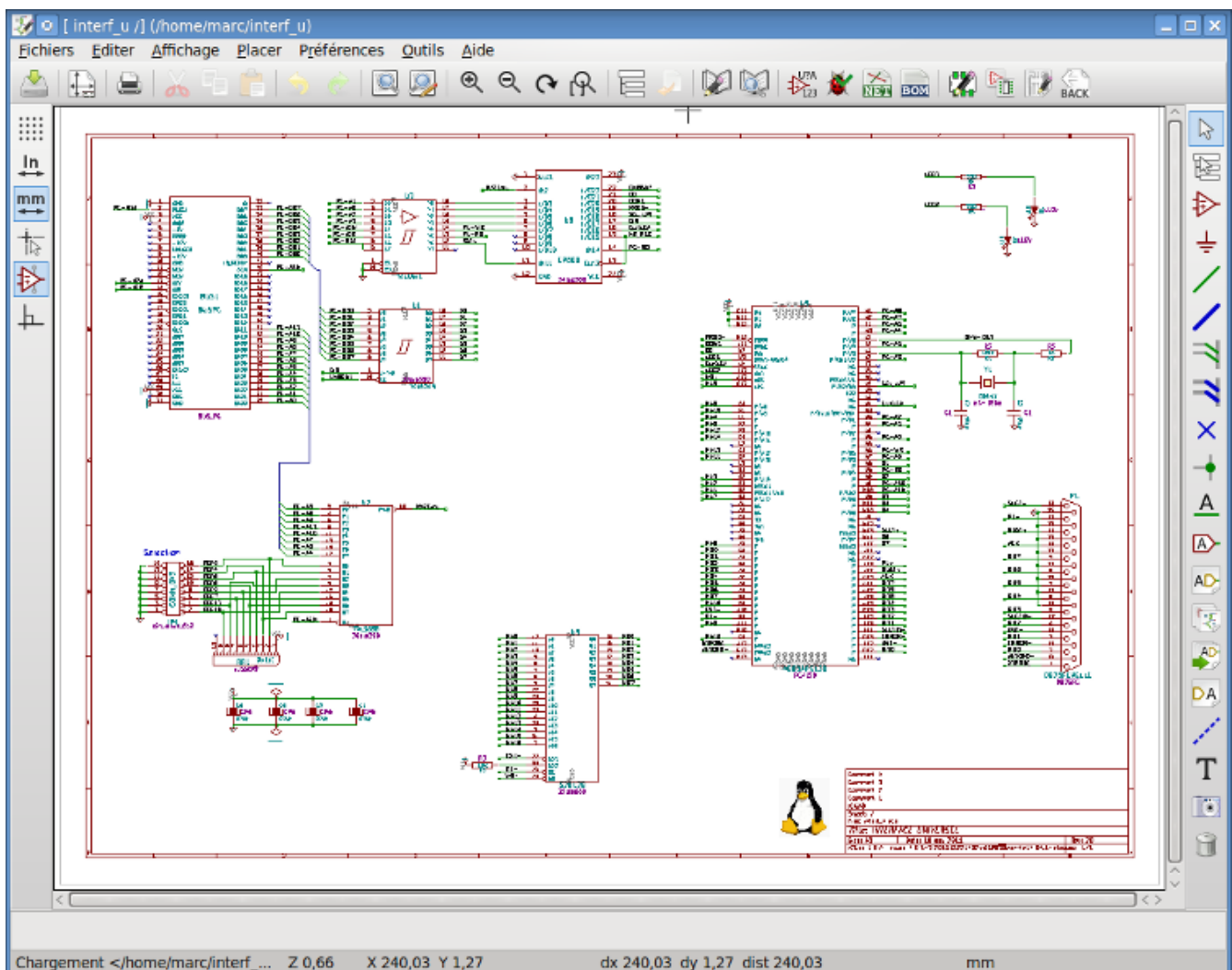
The Schematic Editor can use multi-sheet schematics in a few ways:

- Schémas à hiérarchie simple (chaque schéma n'est utilisé qu'une fois).
- Schémas à hiérarchie complexe (certains schémas sont utilisés plus d'une fois, en plusieurs instances).
- Schémas à hiérarchie plate (les schémas ne font pas explicitement partie d'un schéma maître).

# Generic Schematic Editor commands

Commands can be executed by:

- En cliquant sur les menus, en haut de la fenêtre.
- En cliquant sur les boutons de la barre d'outil principale, au sommet de la fenêtre, sous les menus.
- En cliquant sur les boutons de la barre d'outils à droite de la fenêtre (outils de placement d'éléments).
- En cliquant sur les boutons de la barre d'outils à gauche de la fenêtre (options d'affichage).
- En utilisant la souris (commandes complémentaires importantes), notamment au moyen du clic droit sur un élément du schéma, qui affiche un menu contextuel (options de zoom, de dimension de grille et d'édition des éléments).
- Function keys ( **F1** , **F2** , **F3** , **F4** , **Insert** and **Space** ). Specifically: **Escape** cancels the command in progress. **Insert** allows the duplication of the last element created.
- Pressing hotkeys. For a list of hotkeys, see the **Help** → **List Hotkeys** menu entry or press **Ctrl** + **F1** . Many hotkeys select a tool but do not perform the tool's action until the canvas is clicked. This behavior can be changed by unchecking **First hotkey selects tool** in the **Common** Preferences pane. With this option unchecked, pressing a hotkey will select the tool and immediately perform the tool's action at the current cursor location.



# Commandes à la souris

## Commandes de base

### Bouton gauche

- Single click: Selects the item under the cursor and displays the item's characteristics in the status bar.
- Double click: edits the item if it is editable.
- Long click (click and hold): opens a pop-up menu to clarify the selection.

### Bouton droit

- Opens a pop-up menu. If an item is selected, the items in the menu are related to the selected item. If an item is under the cursor when the right mouse button is clicked, the item is selected.

## Selection operations

Schematic editor items can be selected by clicking on them. Multiple items can be selected at once. Add items to the selection with **Shift** + click, and remove items from the selection with **Ctrl** + **Shift** + click.

**NOTE** | On Apple keyboards, use the **Cmd** key instead of **Ctrl**.

left mouse button	Select item.
<b>Shift</b> + left mouse button	Add item to selection.
<b>Ctrl</b> + <b>Shift</b> + left mouse button	Remove item from selection.
long click	Clarify selection from a pop-up menu.
<b>Ctrl</b> + left mouse button	Highlight net.

Items can also be selected by drawing a box around them using the left mouse button.

Dragging from left to right includes all items fully enclosed by the box. Dragging from right to left includes all items touched by the box, even if they are not fully enclosed.

The **Shift** and **Ctrl** + **Shift** modifiers also work with drag selections to add and remove items from the selection, respectively.

## Raccourcis clavier

- The **Ctrl** + **F1** displays the current hotkey list.
- All hotkeys can be redefined using the hotkey editor (**Preferences** → **Preferences...** → **Hotkeys**).






























The default hotkey list is below. Many additional actions do not have hotkeys by default, but hotkeys can be assigned to them with the hotkey editor.



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**.

Action	Default Hotkey	Description
Click	<b>Return</b>	Performs left mouse button click
Double-click	<b>End</b>	Performs left mouse button double-click
Cursor Down	<b>Down</b>	
Cursor Down Fast	<b>Ctrl</b> + <b>Down</b>	
Cursor Left	<b>Left</b>	
Cursor Left Fast	<b>Ctrl</b> + <b>Left</b>	
Cursor Right	<b>Right</b>	
Cursor Right Fast	<b>Ctrl</b> + <b>Right</b>	
Cursor Up	<b>Up</b>	
Cursor Up Fast	<b>Ctrl</b> + <b>Up</b>	
Switch to Fast Grid 1	<b>Alt</b> + <b>1</b>	
Switch to Fast Grid 2	<b>Alt</b> + <b>2</b>	
Switch to Next Grid	<b>N</b>	
Switch to Previous Grid	<b>Shift</b> + <b>N</b>	
Reset Grid Origin	<b>Z</b>	
Grid Origin	<b>S</b>	Set the grid origin point
New...	<b>Ctrl</b> + <b>N</b>	Create a new document in the editor
Open...	<b>Ctrl</b> + <b>O</b>	Open existing document
Pan Down	<b>Shift</b> + <b>Down</b>	
Pan Left	<b>Shift</b> + <b>Left</b>	
Pan Right	<b>Shift</b> + <b>Right</b>	
Pan Up	<b>Shift</b> + <b>Up</b>	
Print...	<b>Ctrl</b> + <b>P</b>	Print

Action	Default Hotkey	Description
Reset Local Coordinates		
Save	+	Save changes
Save As...	+  +	Save current document to another location
Always Show Cursor	+  +	Display crosshairs even in selection tool
Switch units	+	Switch between imperial and metric units
Update PCB from Schematic...		Update PCB with changes made to schematic
Center		Center
Zoom to Objects	+	Zoom to Objects
Zoom to Fit		Zoom to Fit
Zoom In at Cursor		Zoom In at Cursor
Zoom Out at Cursor		Zoom Out at Cursor
Refresh		Refresh
Zoom to Selection	+	Zoom to Selection
Change Edit Method	+	Change edit method constraints
Copy	+	Copy selected item(s) to clipboard
Cut	+	Cut selected item(s) to clipboard
Delete		Deletes selected item(s)
Duplicate	+	Duplicates the selected item(s)
Find	+	Find text
Find and Replace	+  +	Find and replace text
Find Next		Find next match
Find Next Marker	+	
Paste	+	Paste item(s) from clipboard

Action	Default Hotkey	Description
List Hotkeys...	 + 	Displays current hotkeys table and corresponding commands
Preferences...	 + 	Show preferences for all open tools
Clear Net Highlighting		Clear any existing net highlighting
Edit Library Symbol...	 +  + 	Open the library symbol in the Symbol Editor
Edit with Symbol Editor	 + 	Open the selected symbol in the Symbol Editor
Highlight Net		Highlight net under cursor
Show Datasheet		Opens the datasheet in a browser
Add Sheet		Add a hierarchical sheet
Add Wire to Bus Entry		Add a wire entry to a bus
Add Global Label	 + 	Add a global label
Add Hierarchical Label		Add a hierarchical label
Add Junction		Add a junction
Add Label		Add a net label
Add No Connect Flag		Add a no-connection flag
Add Power		Add a power port
Add Text		Add text
Add Symbol		Add a symbol
Add Bus		Add a bus
Add Lines		Add connected graphic lines
Add Wire		Add a wire
Finish Wire or Bus		Complete drawing at current segment
Unfold from Bus		Break a wire out of a bus
Autoplace Fields		Runs the automatic placement algorithm on the symbol or sheet's fields

Action	Default Hotkey	Description
Mirror Vertically		Flips selected item(s) from top to bottom
Properties...		Displays item properties dialog
Repeat Last Item		Duplicates the last drawn item
Rotate Counterclockwise		Rotates selected item(s) counter-clockwise
Drag		Drags the selected item(s)
Move		Moves the selected item(s)
Select Connection	+	Select a complete connection
Select Node	+	Select a connection item under the cursor
Leave Sheet	+	Display the parent sheet in the schematic editor

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

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

It is possible to import hotkey settings from a `user.hotkeys` file using menu **Preferences** → **Preferences...** → **Hotkeys** → **Import Hotkeys....**

## Grid

In the Schematic Editor the cursor always moves over a grid. The grid can be customized:

- Size can be changed using the right click menu or using **View** → **Grid Properties....**
- Color can be changed in the **Colors** page of the **Preferences** dialog (menu **Preferences** → **General Options**).
- Visibility can be switched using the left-hand toolbar button.

The default grid size is 50 mil (0.050") or 1.27 millimeters.

This is the preferred grid to place symbols and wires in a schematic, and to place pins when designing a symbol in the Symbol Editor.

### 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 clicking **Align Elements to Grid**.

## Snapping

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

On Apple keyboards, use the **Cmd** key instead of **Ctrl**.

Modifier Key	Effect
<b>Ctrl</b>	Disable grid snapping.
<b>Shift</b>	Disable snapping wires to pins.

## Sélection du Zoom

Pour changer le niveau du zoom :

- Cliquez du bouton droit pour ouvrir le menu contextuel et choisissez la valeur de zoom désirée.
- Or use hotkeys:
  - **F1** : Zoom in
  - **F2** : Zoom out
  - **F4** : Center the view around the cursor pointer position
  - **Home** : Zoom and center the view to fit the entire schematic sheet
  - **Ctrl** + **Home** : Zoom and center the view to fit all of the objects in the schematic
  - **Ctrl** + **F5** : Activate the Zoom to Selection tool
- Zoom fenêtre :
  - Mouse wheel: Zoom in/out
  - Shift+Mouse wheel: Pan up/down
  - Ctrl+Mouse wheel: Pan left/right

Mouse scroll gestures are configurable in the **Mouse and Touchpad** page of the **Preferences** dialog.

## Affichage des coordonnées du curseur

The display units are in inches, mils, or millimeters.

Les informations suivantes sont affichées en bas et à droite de la fenêtre :

- Le facteur de Zoom
- La position absolue du curseur (X Y)

La position relative du curseur (dx dy)

- The grid size
- The active unit system
- The active tool

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

Z 2.59    X 447.04 Y 212.09    dx 447.04 dy 212.09 dist 494.80    grid 1.27    mm    Add Wire

## Barre de menu

The top menu bar allows the opening and saving of schematics, program configuration and viewing the documentation.

Fichiers   E\_diter   Affichage   Placer   Préférences   Outils   Aide












## Barre d'outils supérieure
























This toolbar gives access to the main functions of the Schematic Editor.

If the Schematic Editor is run in standalone mode, this is the available tool set:



Note that when KiCad runs in project mode, the first two icons are not available as they work with individual files.

	Create a new schematic (only in standalone mode).
	Open a schematic (only in standalone mode).
	Save complete schematic project.
	Set the schematic-specific options.
	Select the sheet size and edit the title block.
	Open print dialog.
	Open plot dialog.
	Paste a copied/cut item or block to the current sheet.
	Undo: Revert the last change.
	Redo: Revert the last undo operation.
	Show the dialog to search symbols and texts in the schematic.



















	Show the dialog to search and replace texts in the schematic.
	Refresh screen.
	Zoom in.
	Zoom out.
	Zoom to fit the entire schematic sheet.
	Zoom to fit all objects in the schematic.
	Zoom to fit selected items.
	View and navigate the hierarchy tree.
	Leave the current sheet and go up in the hierarchy.
	Rotate selected items counter-clockwise.
	Rotate selected items clockwise.
	Mirror selected items vertically.
	Mirror selected items horizontally.
	Call the symbol library editor to view and modify libraries and symbols.
	Browse symbol libraries.
	Open the footprint library editor to view and modify libraries and footprints.
	Annotate symbols.
	Electrical Rules Checker (ERC), automatically validate electrical connections.
	Open the footprint assignment tool to assign footprints to symbols.
	Bulk edit symbol fields in a spreadsheet interface.
	Generate the Bill of Materials (BOM).
	Open the PCB editor.
	Open the Python scripting console.

## Barre d'outils latérale droite

Cette barre d'outils contient les outils pour :

- Place symbols, wires, buses, junctions, labels, text, etc.








- Create hierarchical subsheets and connection symbols.

	Cancel the active command or tool and go into selection mode.
	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.
	Display the symbol selector dialog to select a new symbol to be placed.
	Display the power symbol selector dialog to select a power symbol to be placed.
	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. It is done to notify the Electrical Rules Checker that lack of connection for a particular pin is intentional and should not be reported.
	Place a junction. This connects two crossing wires or a wire and a pin, when it can be ambiguous (i.e. if a wire end or a pin is not directly connected to another wire end).
	Place a local label. Local label connects items located <b>in the same sheet</b> . For connections between two different sheets, you have to use global or hierarchical labels.
	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 parent sheet that contains it.
	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.
	Draw a line. These are only graphical and do not connect anything.
	Place a text comment.
	Place a bitmap image.
	Delete clicked items.

## Barre d'outils latérale gauche

Cette barre d'outils permet de gérer les options d'affichage :



	Toggle grid visibility.
	Switch units to inches.
	Switch units to mils (0.001 inches).
	Switch units to millimeters.
	Choose the cursor shape (full screen/small).
	Toggle visibility of "invisible" pins.
	Toggle free angle/90 degrees wires and buses placement.

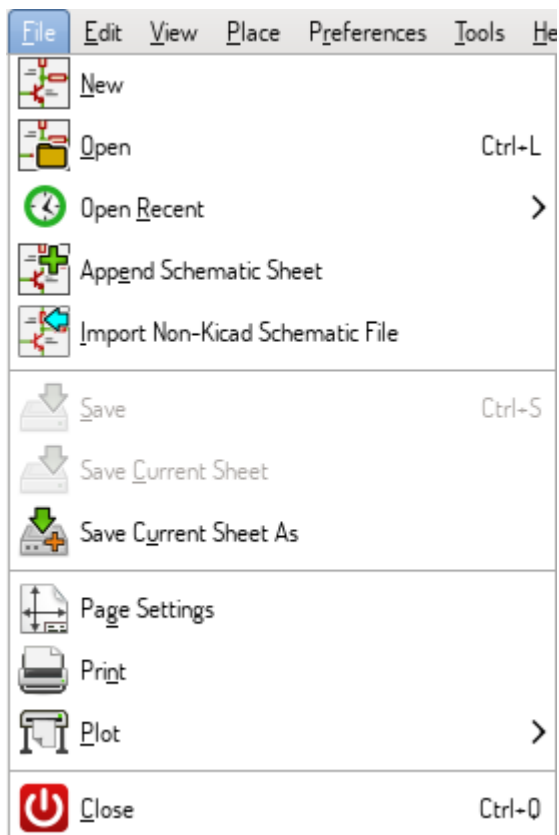
## Menus contextuels et édition rapide

Un clic droit ouvre un menu contextuel pour l'élément sélectionné ou survolé : ce menu permet d'ajuster :

- Le facteur de Zoom.
- La taille de grille.
- Copy/Paste/Delete commands.
- Add Wire/Bus.
- Les paramètres couramment édités de l'élément sélectionné.

# Barre de menus

## Menu Fichiers

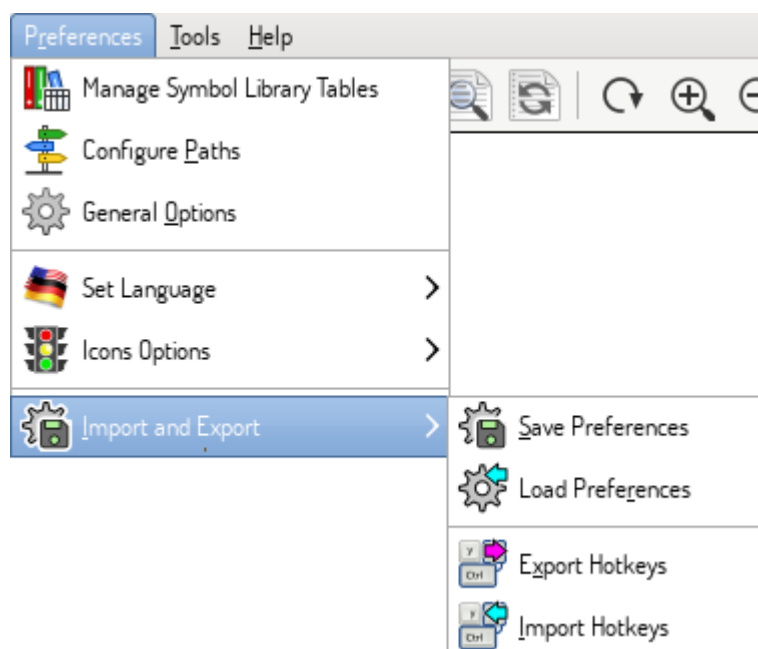


New	<b>Close current schematic and start a new one (only in standalone mode).</b>
Open	Load a schematic project (only in standalone mode).
Open Recent	Open a schematic project from the list of recently opened files (only in standalone mode).
Save	Save current sheet and all its subsheets.
Save As...	Save the current sheet under a new name (only in standalone mode).
Save Current Sheet Copy As...	Save a copy of the current sheet under a new name (only in project mode).
Insert Schematic Sheet Content...	Insert the contents of another schematic sheet into the current sheet (only in standalone mode).
Import	Import a non-KiCad schematic or a footprint assignment file.
Export	Export a netlist or a drawing of the schematic to the clipboard.
Schematic Setup...	Set up schematic formatting, electrical rules, net classes, and text variables.
Page Settings...	Configure page dimensions and title block.
Print	Print schematic project (See also chapter <a href="#">Plot and Print</a> ).
Plot	Export to PDF, PostScript, HPGL or SVG format (See chapter <a href="#">Plot and Print</a> ).
Quit	Terminate the application.

## Schematic Setup

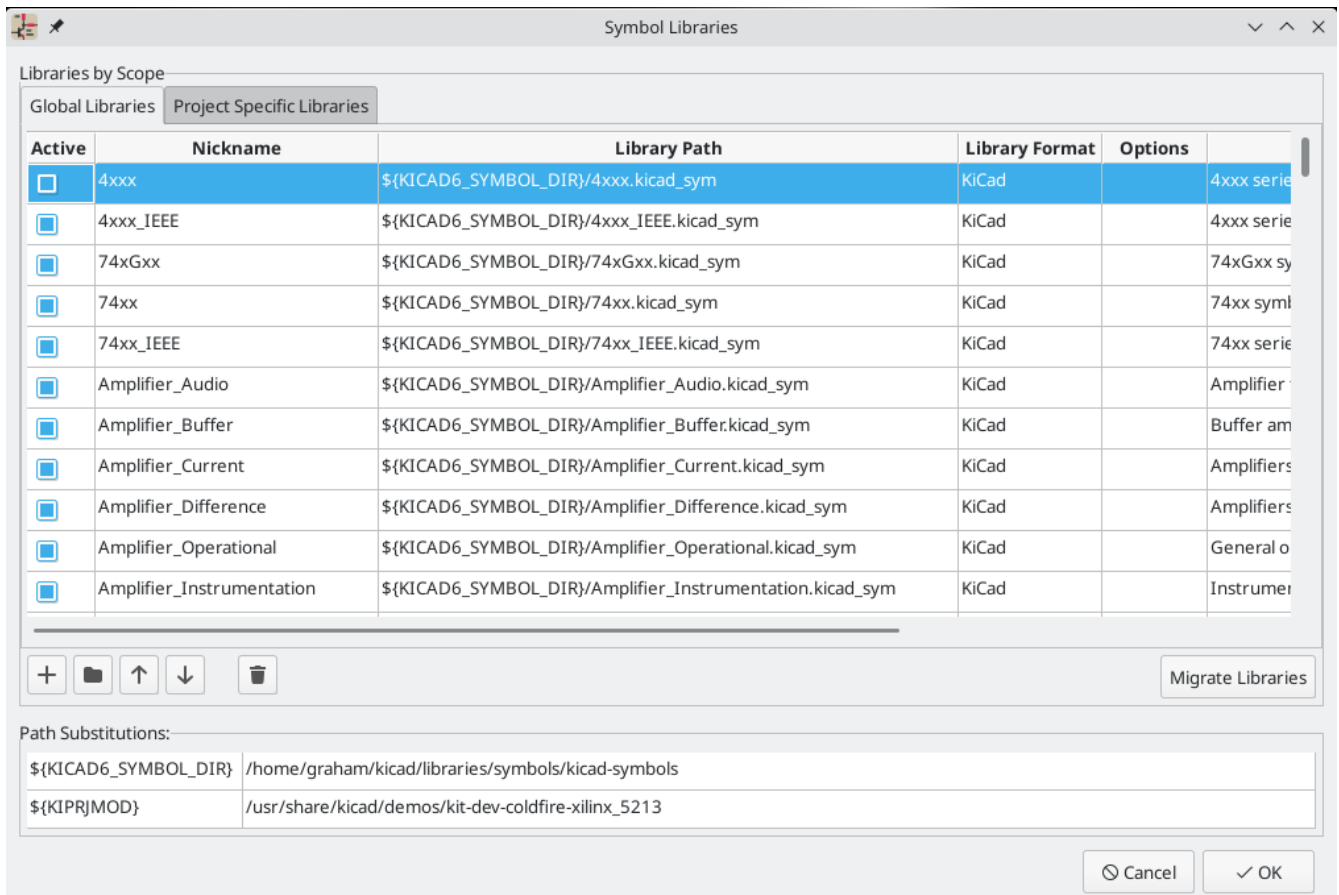
The Schematic Setup window is used to set schematic options that are specific to the currently active schematic. For example, the Schematic Setup window contains formatting options, electrical rule configuration, netclass setup, and schematic text variable setup.

## Menu Préférences



Configure Paths...	Set the default search paths.
Manage Symbol Library Tables...	Add/remove symbol libraries.
Preferences...	Preferences (units, grid size, field names, etc.).
Set Language	Select interface language.

# Manage Symbol Library Tables



KiCad uses two library tables to store the list of available symbol libraries, which differ by the scope:

## Global Libraries

Libraries listed in the Global Library table are available to every project. They are saved in the `sym-lib-table` in the KiCad configuration directory, which is system-dependent:


- Windows: `%APPDATA%\kicad\6.0\sym-lib-table`
- Linux: `~/.config/kicad/6.0/sym-lib-table`
- macOS: `~/Library/Preferences/kicad/6.0/sym-lib-table`

## Project Specific Libraries

Libraries listed in Project Specific Libraries table are available to the currently opened project. They are saved in a `sym-lib-table` file in the project directory.

Both library tables are visible by clicking on **Global Libraries** or **Project Specific Libraries** tab in the Manage Library Tables window.

## Add a new library

Add a library either by clicking the  button and selecting a file or clicking the **+** button and typing a path to a library file. The selected library will be added to the currently opened library table (Global/Project Specific).

## Remove a library

Remove a library by selecting one or more libraries and clicking the  button.

## Library properties

Each row in the table stores several fields describing a library:

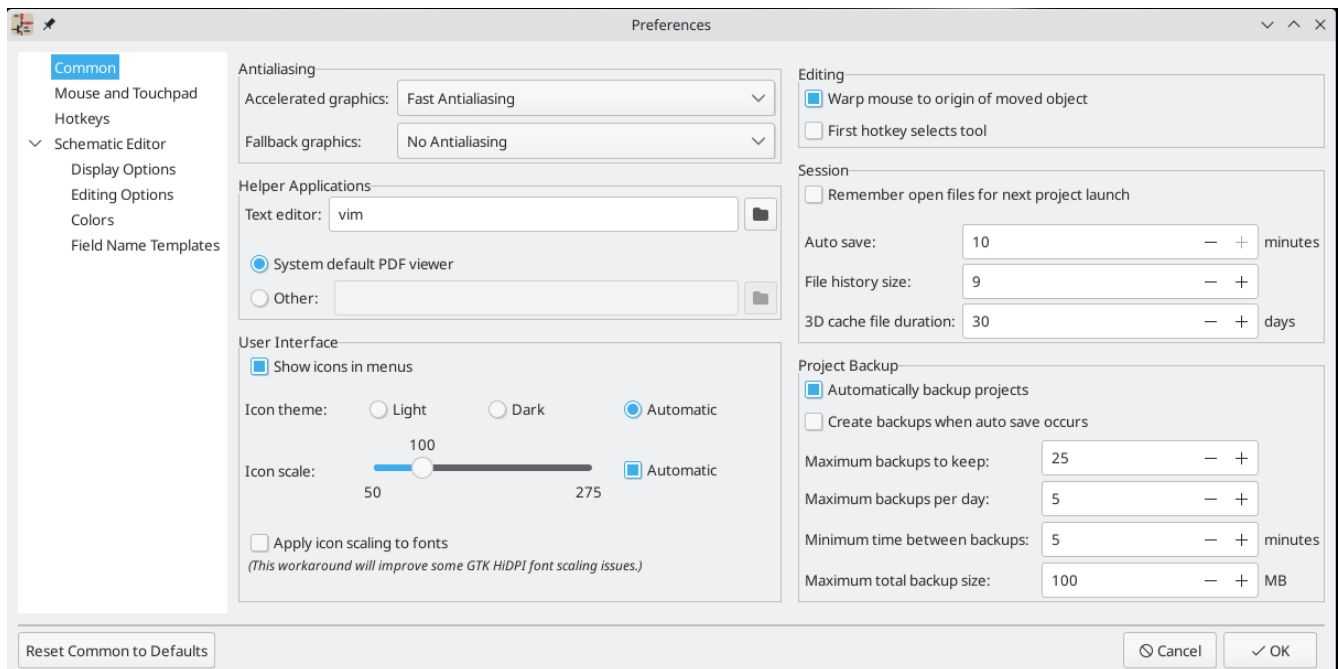
Active	Enables/disables the library. It is useful to temporarily reduce the loaded library set.
Nickname	Nickname is a short, unique identifier used for assigning symbols to components. Symbols are represented by '<Library Nickname>:<Symbol Name>' strings.
Library Path	Path points to the library location.
Plugin Type	Determines the library file format. KiCad 6.0 libraries use the "KiCad" format, while KiCad 5.x libraries use the "Legacy" format. Legacy libraries are read-only.
Options	Stores library specific options, if used by plugin.
Description	Briefly characterizes the library contents.

## Préférences

### Common Preferences

NOTE

TODO: write this section

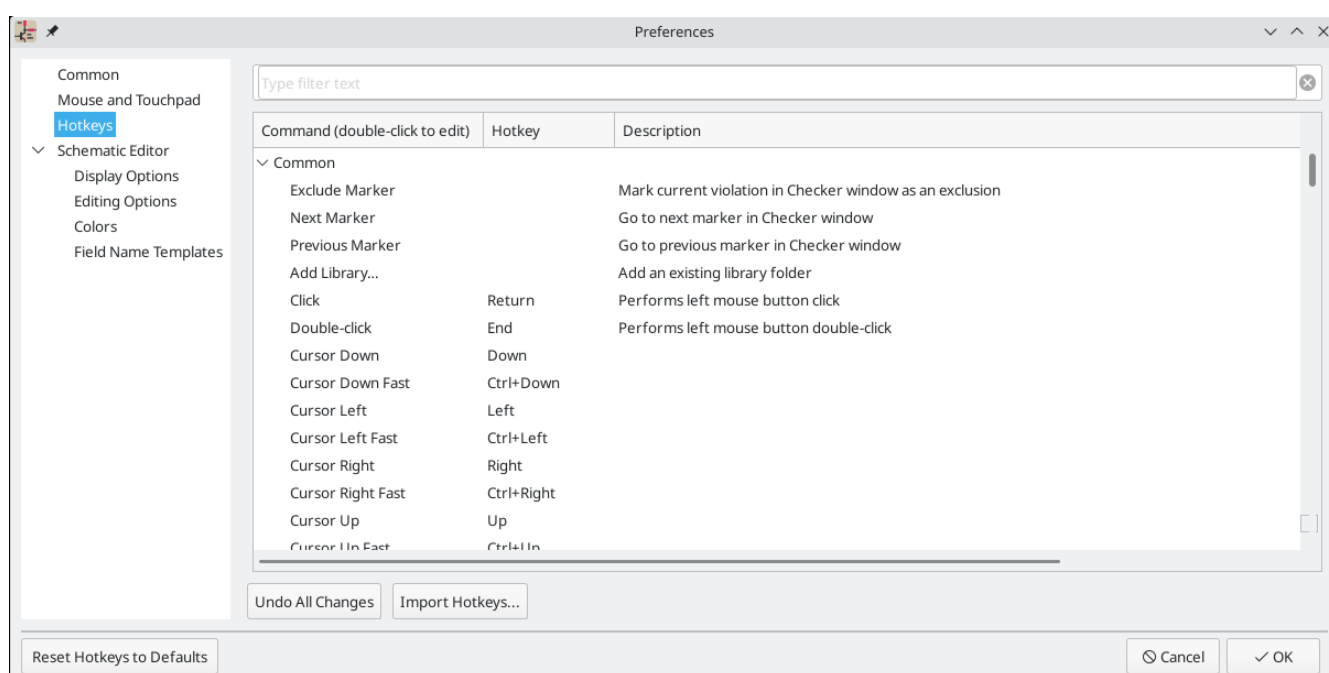


## Mouse and Touchpad

Center and warp cursor on zoom	If checked, the pointed location is warped to the screen center when zooming in/out.
Use touchpad to pan	When enabled, view is panned using scroll wheels (or touchpad gestures) and to zoom one needs to hold <b>Ctrl</b> . Otherwise scroll wheels zoom in/out and <b>Ctrl</b> / <b>Shift</b> are the panning modifiers.
Pan while moving object	If checked, automatically pans the window if the cursor leaves the window during drawing or moving.

## Raccourcis clavier

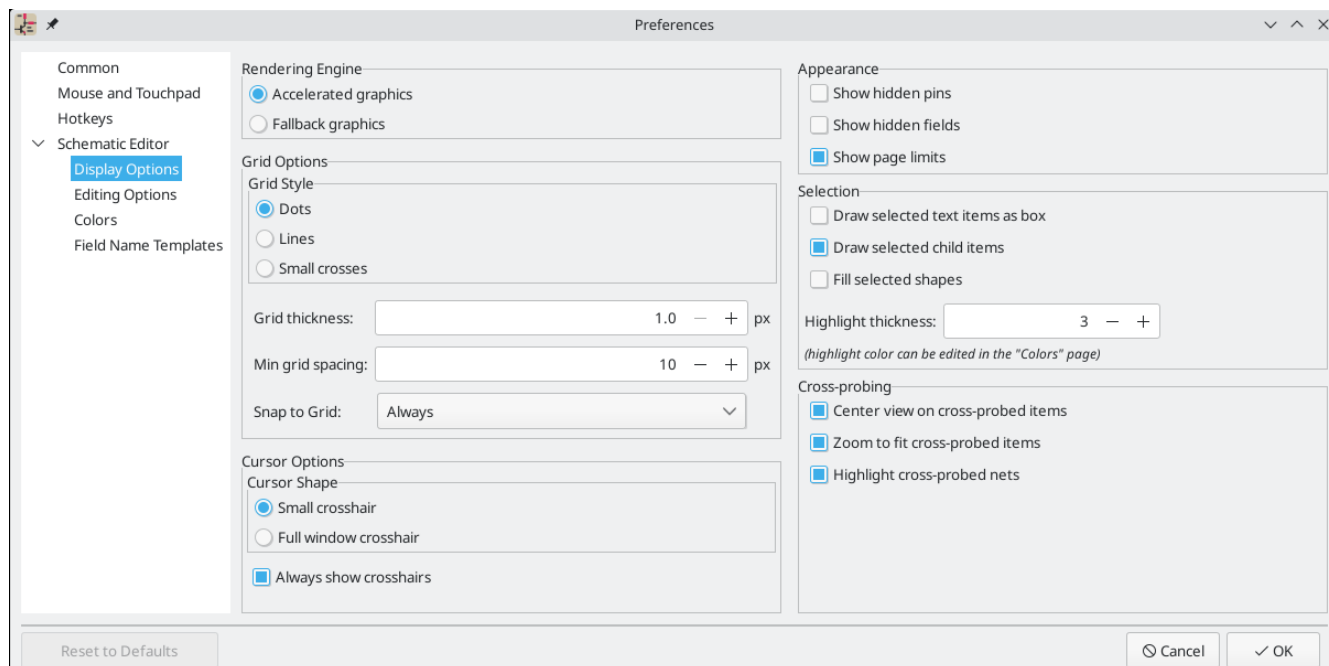
Redefine hotkeys.



Select a new hotkey by double clicking an action or right click on an action to show a popup menu:

Edit	Define a new hotkey for the action (same as double click).
Undo Changes	Reverts the recent hotkey changes for the action.
Clear Assigned Hotkey	
Restore Default	Sets the action hotkey to its default value.

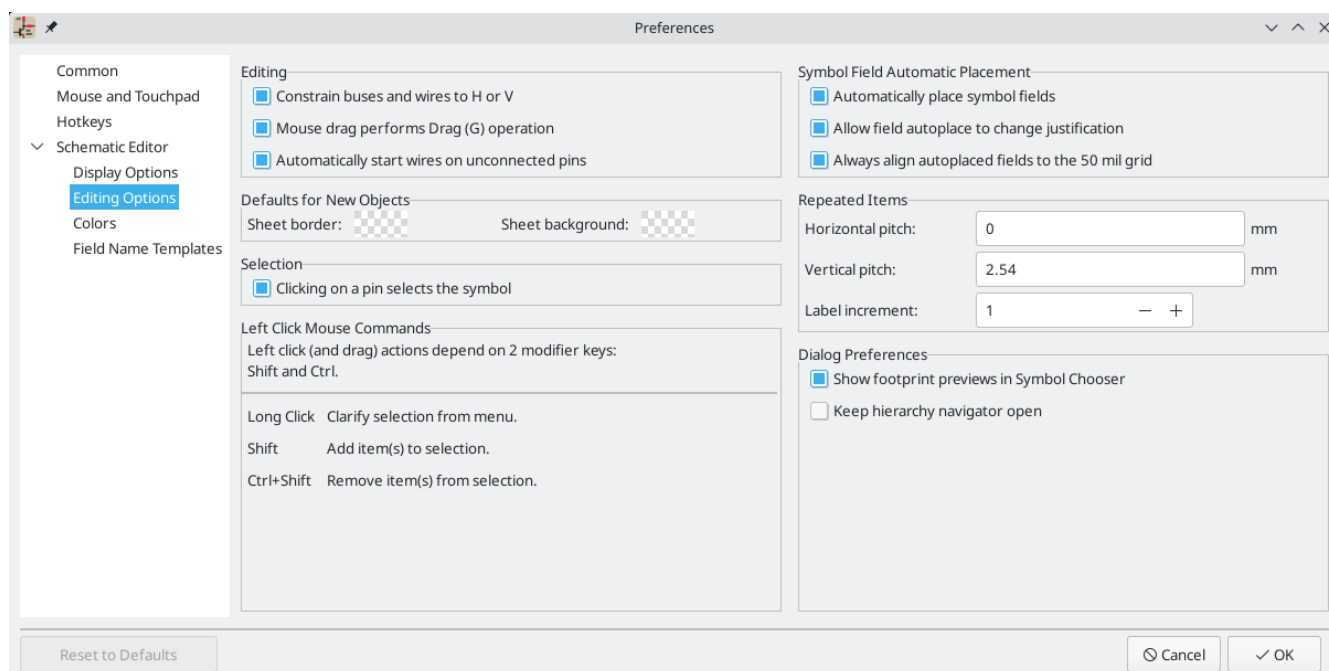
## Display Options



Grid Size	<p>Grid size selection.</p> <p>It is <b>recommended</b> to work with normal grid (0.050 inches or 1,27 mm). Smaller grids are used for component building.</p>
Bus thickness	Pen size used to draw buses.
Line thickness	Pen size used to draw objects that do not have a specified pen size.
Part ID notation	Style of suffix that is used to denote symbol units (U1A, U1.A, U1-1, etc.)
Icon scale	Adjust toolbar icons size.
Show Grid	Grid visibility setting.
Restrict buses and wires to H and V orientation	If checked, buses and wires are drawn only with vertical or horizontal lines. Otherwise buses and wires can be placed at any orientation.
Show hidden pins:	Display invisible (or <i>hidden</i> ) pins, typically power pins.
Show page limits	If checked, shows the page boundaries on screen.
Footprint previews in symbol chooser	<p>Displays a footprint preview frame and footprint selector when placing a new symbol.</p> <p><b>Note:</b> it may cause problems or delays, use at your own risk.</p>



## Editing Options



The screenshot shows the 'Preferences' dialog box with the 'Editing Options' tab selected. The left sidebar lists categories: Common, Mouse and Touchpad, Hotkeys, Schematic Editor, Display Options, Editing Options (selected), Colors, and Field Name Templates. The main area is divided into several sections:

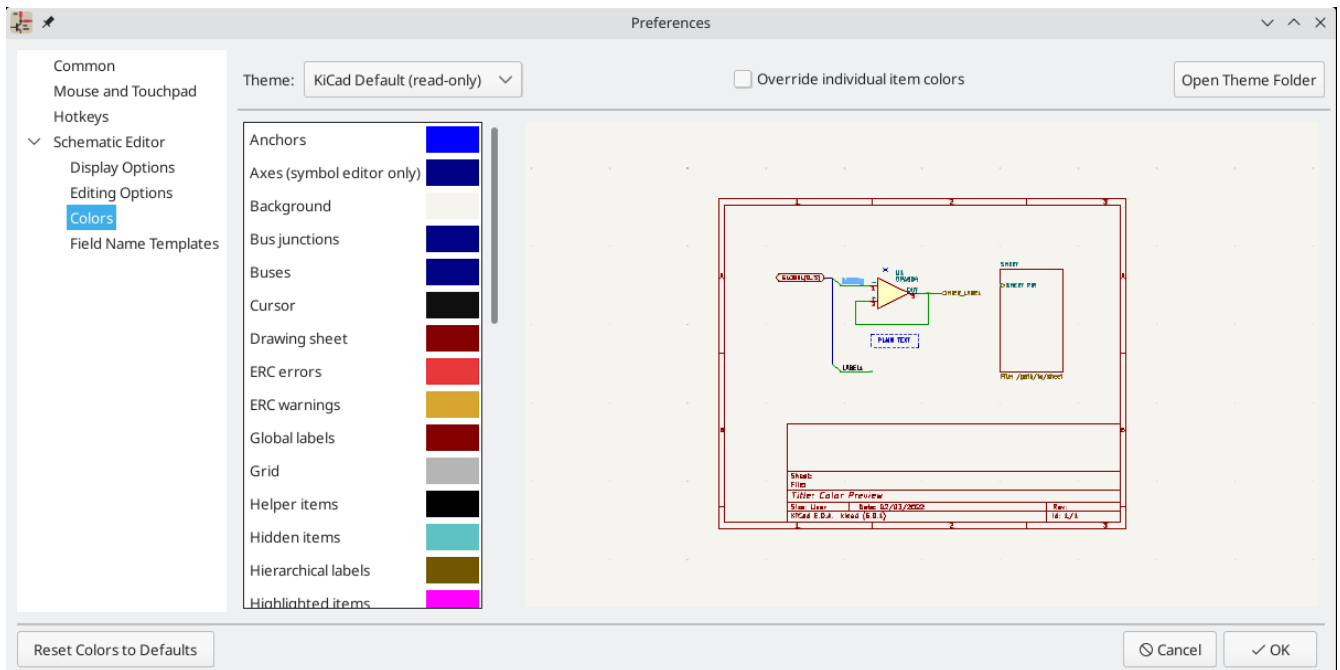
- Editing:** Three checked options: 'Constrain buses and wires to H or V', 'Mouse drag performs Drag (G) operation', and 'Automatically start wires on unconnected pins'.
- Defaults for New Objects:** 'Sheet border' and 'Sheet background' both set to a checkerboard pattern.
- Selection:** One checked option: 'Clicking on a pin selects the symbol'.
- Left Click Mouse Commands:** A list of actions: 'Left click (and drag) actions depend on 2 modifier keys: Shift and Ctrl.', 'Long Click: Clarify selection from menu.', 'Shift: Add item(s) to selection.', and 'Ctrl+Shift: Remove item(s) from selection.'
- Symbol Field Automatic Placement:** Three checked options: 'Automatically place symbol fields', 'Allow field autoplace to change justification', and 'Always align autoplace fields to the 50 mil grid'.
- Repeated Items:** 'Horizontal pitch' set to 0 mm, 'Vertical pitch' set to 2.54 mm, and 'Label increment' set to 1 with minus and plus buttons.
- Dialog Preferences:** Two options: 'Show footprint previews in Symbol Chooser' (checked) and 'Keep hierarchy navigator open' (unchecked).

At the bottom, there are buttons for 'Reset to Defaults', 'Cancel', and 'OK'.

Measurement units	Select the display and the cursor coordinate units (inches or millimeters).
Horizontal pitch of repeated items	Increment on X axis during element duplication (default: 0) (after placing an item like a symbol, label or wire, a duplication is made by the <span>Insert</span> key)
Vertical pitch of repeated items	Increment on Y axis during element duplication (default: 0.100 inches or 2,54 mm).
Increment of repeated labels	Increment of label value during duplication of texts ending in a number, such as bus members (usual value 1 or -1).
Default text size	Text size used when creating new text items or labels.
Auto-save time interval	Time in minutes between saving backups.
Automatically place symbol fields	If checked, symbol fields (e.g. value and reference) in newly placed symbols might be moved to avoid collisions with other items.
Allow field autoplace to change justification	Extension of 'Automatically place symbol fields' option. Enable text justification adjustment for symbol fields when placing a new part.
Always align autoplace fields to the 50 mil grid	Extension of 'Automatically place symbol fields' option. If checked, fields are autoplace using 50 mils grid, otherwise they are placed freely.

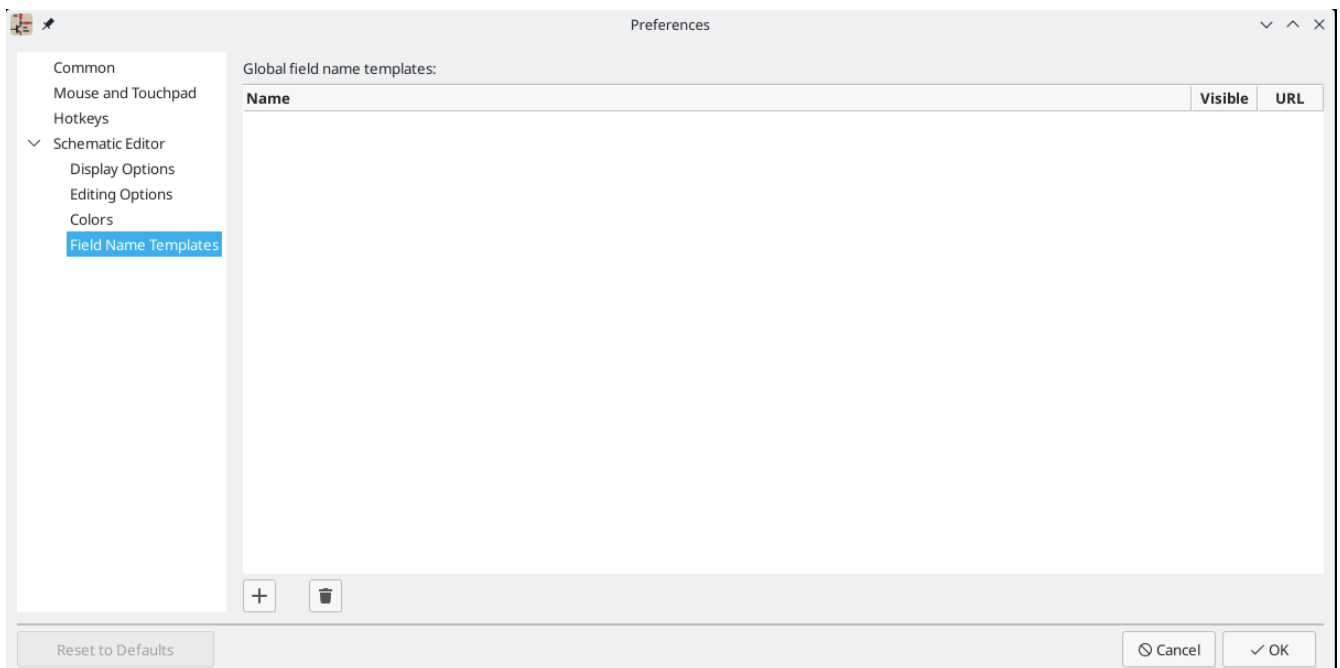
## Colors

Color scheme for various graphic elements. Click on any of the color swatches to select a new color for a particular element.



## Default Fields

Define additional custom fields and corresponding values that will appear in newly placed symbols.



## Menu Aide

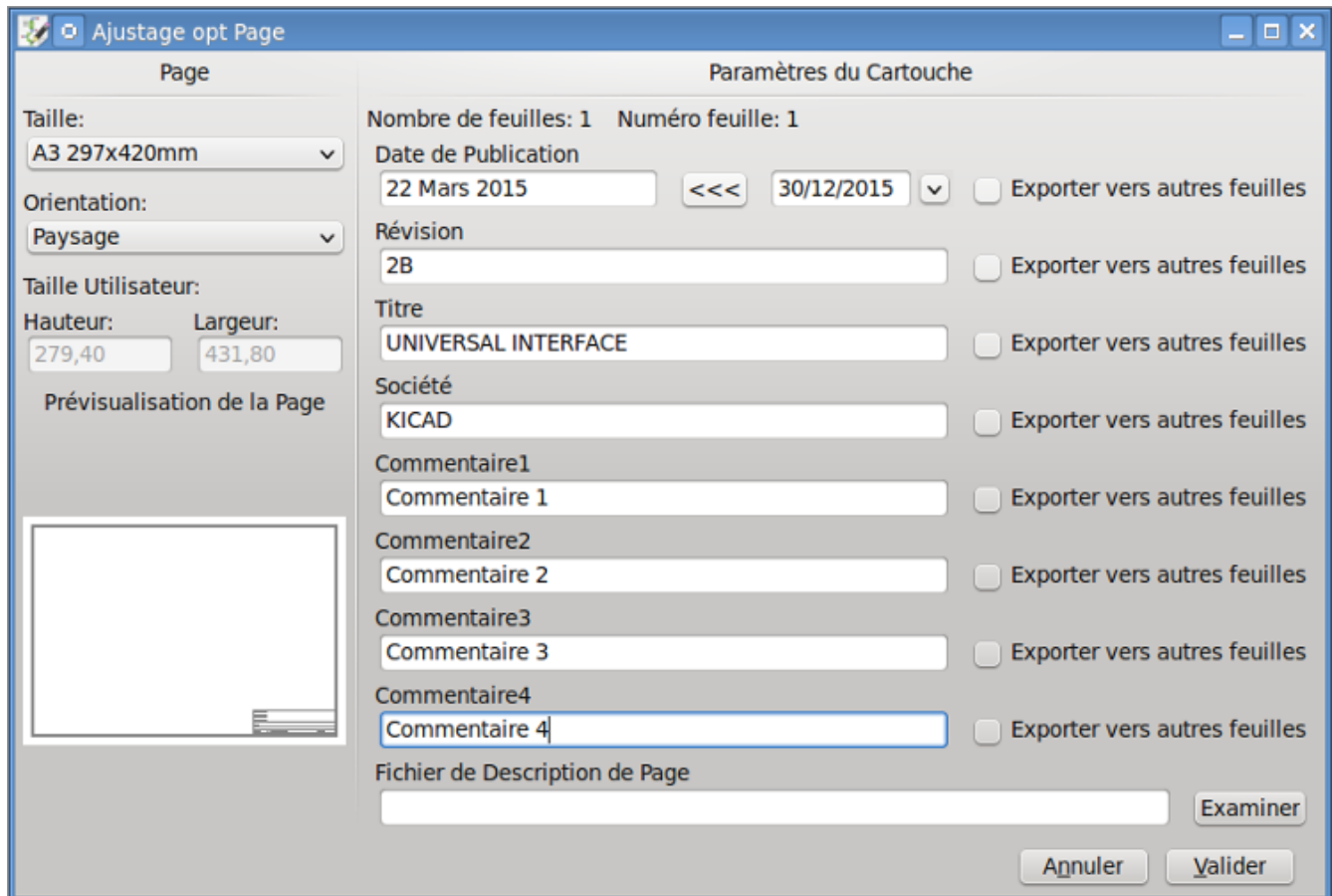
Access to on-line help (this document) for an extensive tutorial about KiCad.

Use the **Report a Bug** item to report a bug online. Full KiCad version and user system information is available via the **Copy Version Info** button in the **About KiCad** window.

# Barre d'outils principale

## Gestion des feuilles schématiques

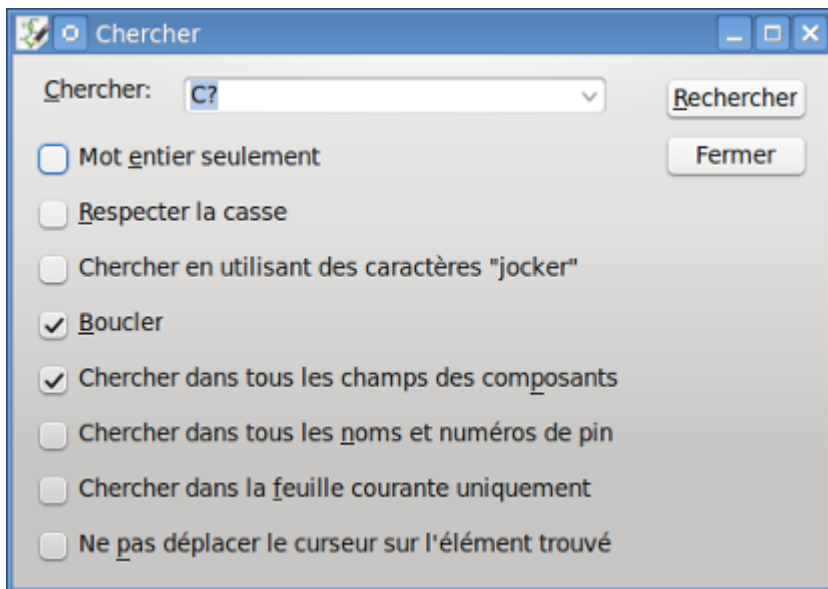
The Sheet Settings icon (  ) allows you to define the sheet size and the contents of the title block.



Le nombre de feuilles, numéro de feuille, sont mis à jour automatiquement. La date ne sera pas changée automatiquement, mais vous pouvez la fixer à aujourd'hui en cliquant sur le bouton "←".

## Outil de recherche

The Find icon (  ) can be used to access the search tool.



You can search for a reference, a value or a text string in the current sheet or in the whole hierarchy. Once found, the cursor will be positioned on the found element in the relevant sub-sheet.

## Outil de Netliste

The Netlist icon () opens the netlist generation tool.

The tool creates a file which describe all connections in the entire hierarchy.

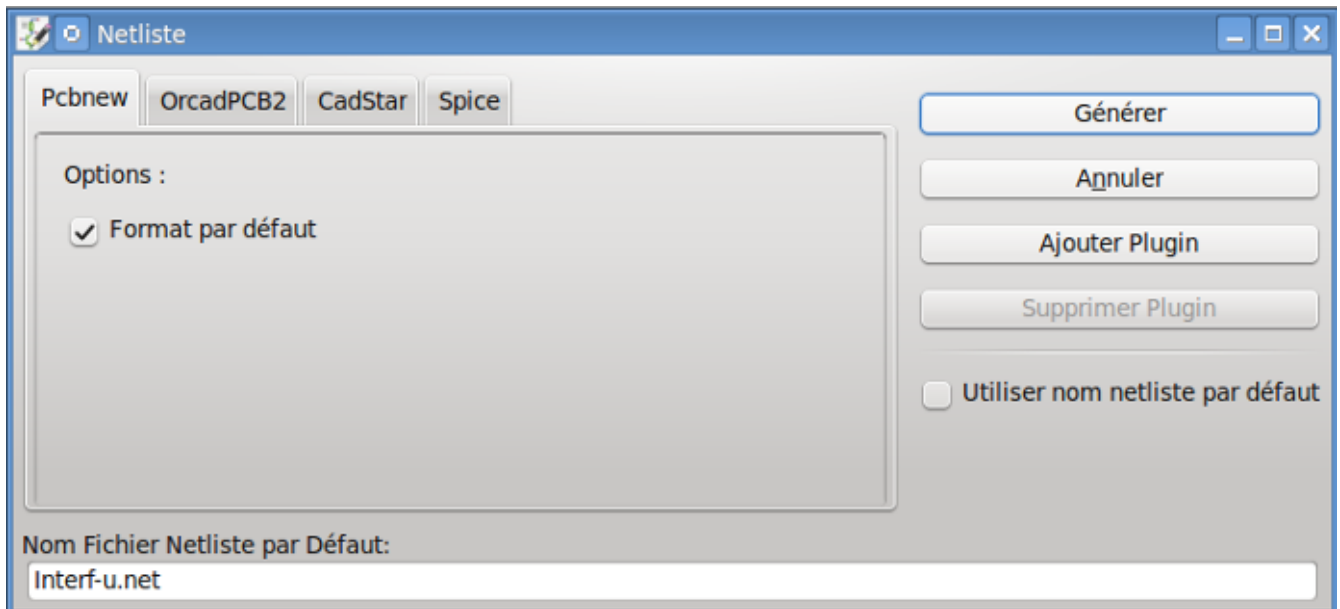
In a multisheet hierarchy, any local label is visible only inside the sheet to which it belongs. For example: the label LABEL1 of sheet 3 is different from the label LABEL1 of sheet 5 (if no connection has been intentionally introduced to connect them). This is due to the fact that the sheet name path is internally associated with the local label.

### NOTE

Even though there is no text length limit for labels in KiCad, please take into account that other programs reading the generated netlist may have such constraints.

### NOTE

Avoid spaces in labels, because they will appear as separated words in the generated file. It is not a limitation of KiCad, but of many netlist formats, which often assume that a label has no spaces.



Options :

Default Format	Check to select Pcbnew as the default format.
----------------	---

D'autres formats de netlistes peuvent être générés :

- Orcad PCB2
- CadStar
- Spice (simulators)

External plugins can be added to extend the netlist formats list (PadsPcb Plugin was added in the picture above).


There is more information about creating netlists in [Create a Netlist](#) chapter.

## Outil d'annotation

The icon  launches the annotation tool. This tool assigns references to components.

Pour des composants multi-unités (comme par exemple le 7400 qui contient 4 portes), un suffixe d'unité sera attribué (ainsi notre 7400 désigné par la référence U3 sera divisé en quatre unités référencées U3A, U3B, U3C et U3D).

You can unconditionally annotate all the components or only the new components, i.e. those which were not previously annotated.


Annotation de la Schématique
[-] [ ] [X]



### Sélection

☒ Utiliser la schématique entière
 ☐ Utiliser la feuille active uniquement

☒ Garder l'annotation existante
 ☐ Supprimer l'annotation existante
 ☐ Reset, mais ne pas échanger les unités déjà numérotées des boîtiers mutli-unités

### Ordre de l'Annotation

☒ Trier les composants par position X
 ☐ Trier les composants par position Y

### Choix Annotation

☒ Utiliser le premier nombre libre de la schématique
 ☐ Démarrer à numéro de feuille \*100 et utiliser le premier nombre libre
 ☐ Démarrer à numéro de feuille \*1000 et utiliser le premier nombre libre

### Dialogue

☐ Gardez ce Dialogue Ouvert
 ☒ Toujours demander pour confirmation

Fermer

Suppression Annotation

Numérotation

## Portée

Use the entire schematic	All sheets are re-annotated (default).
Use the current page only	Only the current sheet is re-annotated (this option is to be used only in special cases, for example to evaluate the amount of resistors in the current sheet.).
Keep existing annotation	Conditional annotation, only the new components will be re-annotated (default).
Reset existing annotation	Unconditional annotation, all the components will be re-annotated (this option is to be used when there are duplicated references).
Reset, but do not swap any annotated multi-unit parts	Keeps all groups of multiple units (e.g. U2A, U2B) together when reannotating.

## Ordre d'annotation

Selects the order in which components will be numbered (either horizontally or vertically).

## Choix de l'annotation

Selects the assigned reference format.

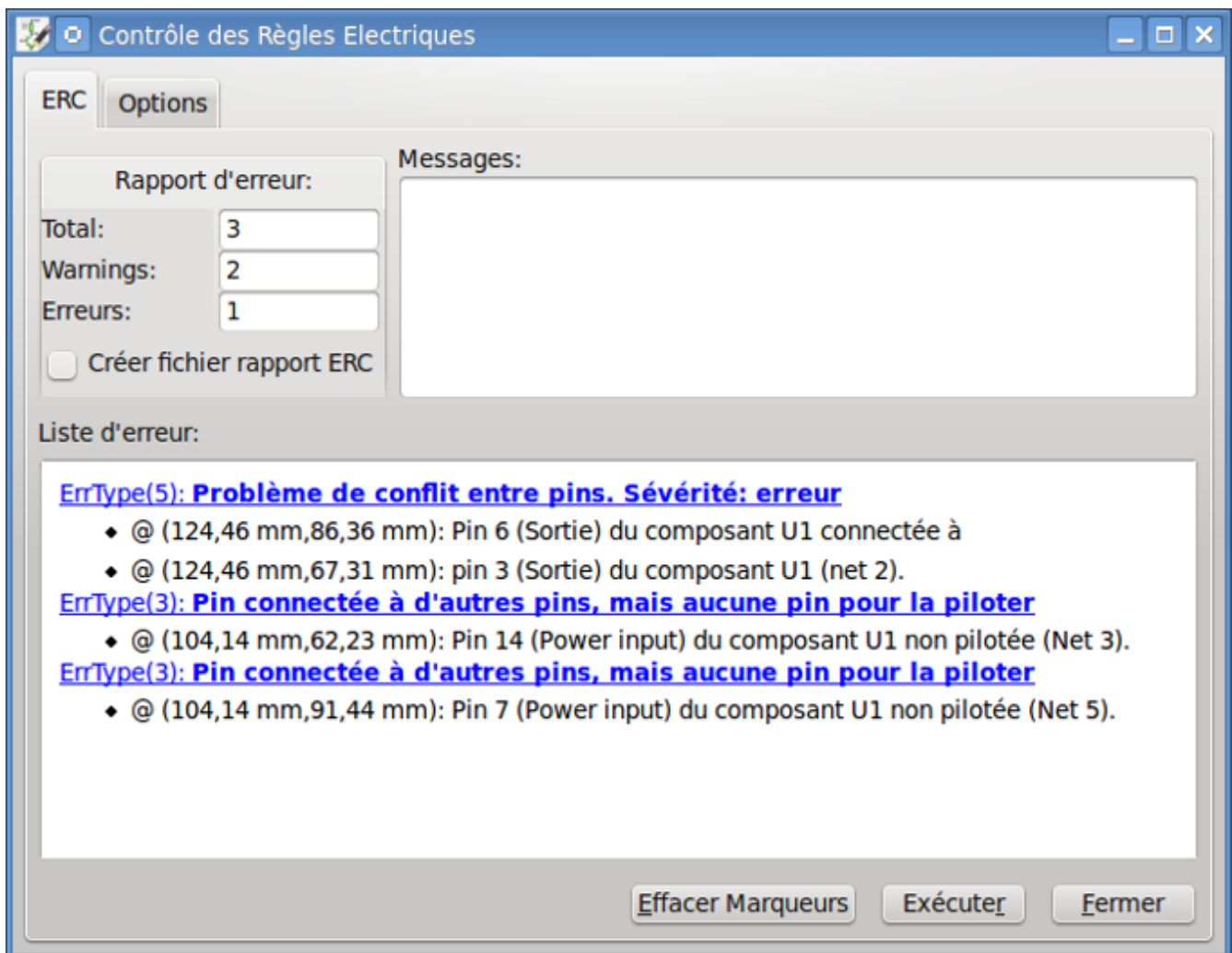
## Outil de vérification des règles électriques

The icon  launches the electrical rules check (ERC) tool.

This tool performs a design verification and is able to detect forgotten connections, and inconsistencies.

Once you have run the ERC, KiCad places markers to highlight problems. The error description is displayed after left clicking on the marker. An error report file can also be generated.

## Fenêtre principale de l'ERC



Errors are displayed in the Electrical Rules Checker dialog:

- Total : nombre total d'erreurs et avertissements.
- Erreurs : nombre d'erreurs.
- Warnings : nombre d'avertissements.

Options :

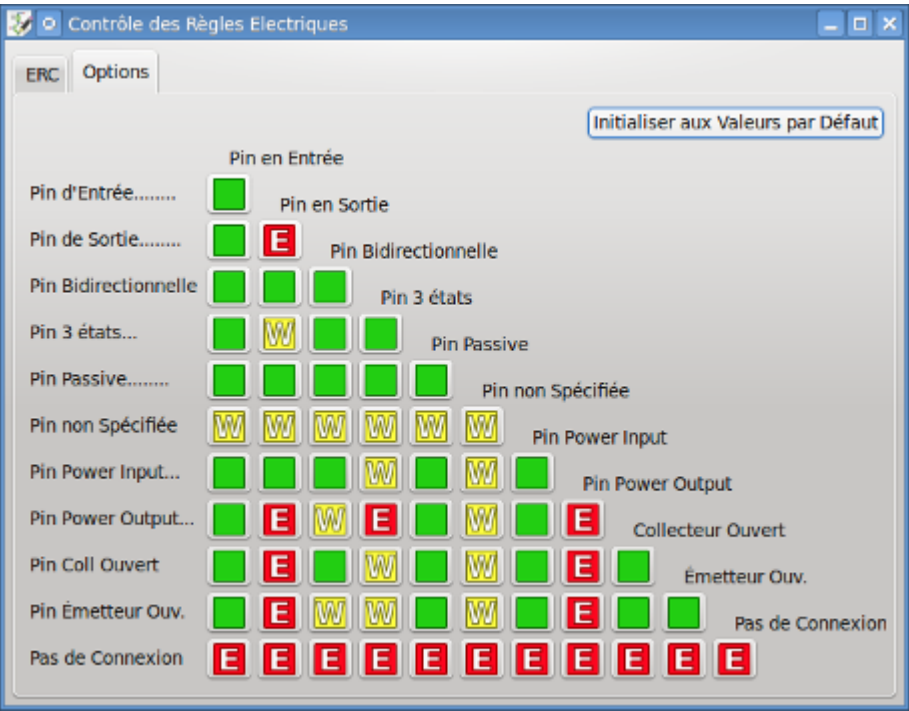
Create ERC file report	Check this option to generate an ERC report file.
------------------------	---

Commandes :

Delete Markers	Remove all ERC error/warnings markers.
Run	Start an Electrical Rules Check.
Close	Close the dialog.

- En cliquant sur une erreur, vous êtes emmenés au marqueur correspondant sur le schéma.

## Options de l'ERC



This tab allows you to define the connectivity rules between pins; you can choose between 3 options for each case:

- Pas d'erreur (Vert)
- Avertissement (W jaune)
- Erreur (E rouge)

Chaque carré de la matrice peut être modifié en cliquant une ou plusieurs fois dessus.

Options :


Test similar labels	Report labels that differ only by letter case (e.g. label/Label/LaBeL). Net names are case-sensitive therefore such labels are treated as separate nets.
Test unique global labels	Report global labels that occur only once for a particular net. Normally it is required to have at least two make a connection.

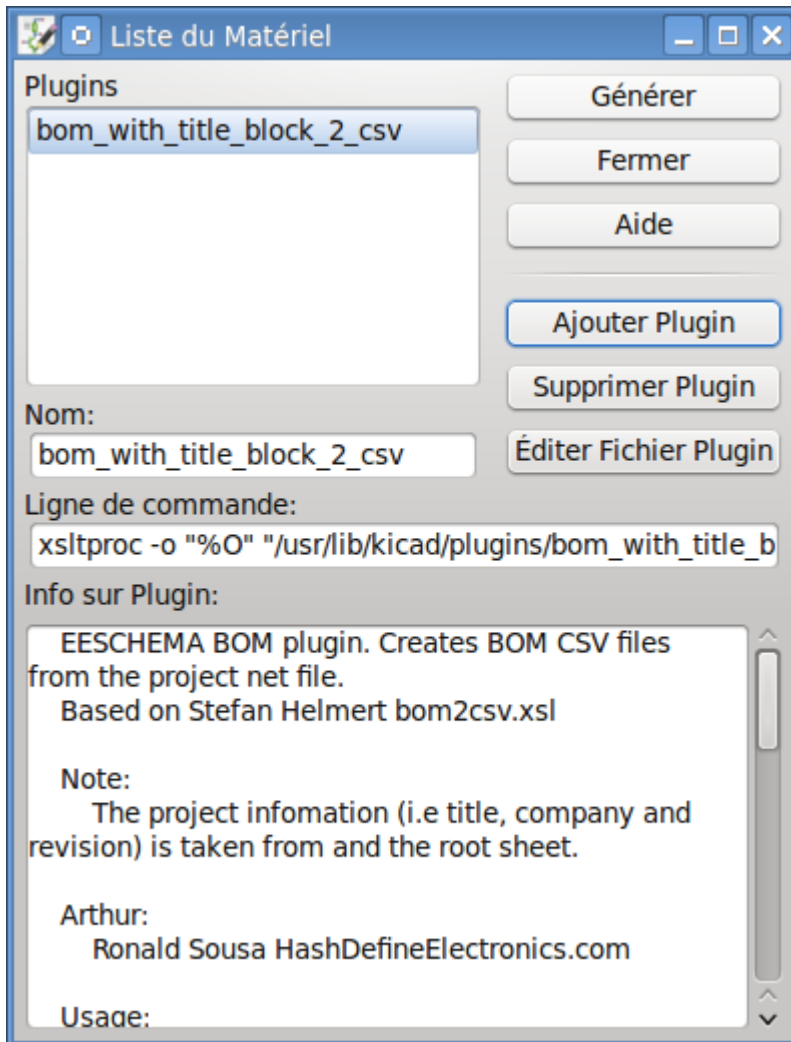


Commandes :

Initialize to Default	Restores the original settings.
-----------------------	---------------------------------

## Outil de Liste de Matériel

The icon  launches the bill of materials (BOM) generator. This tool generates a file listing the components and/or hierarchical connections (global labels).

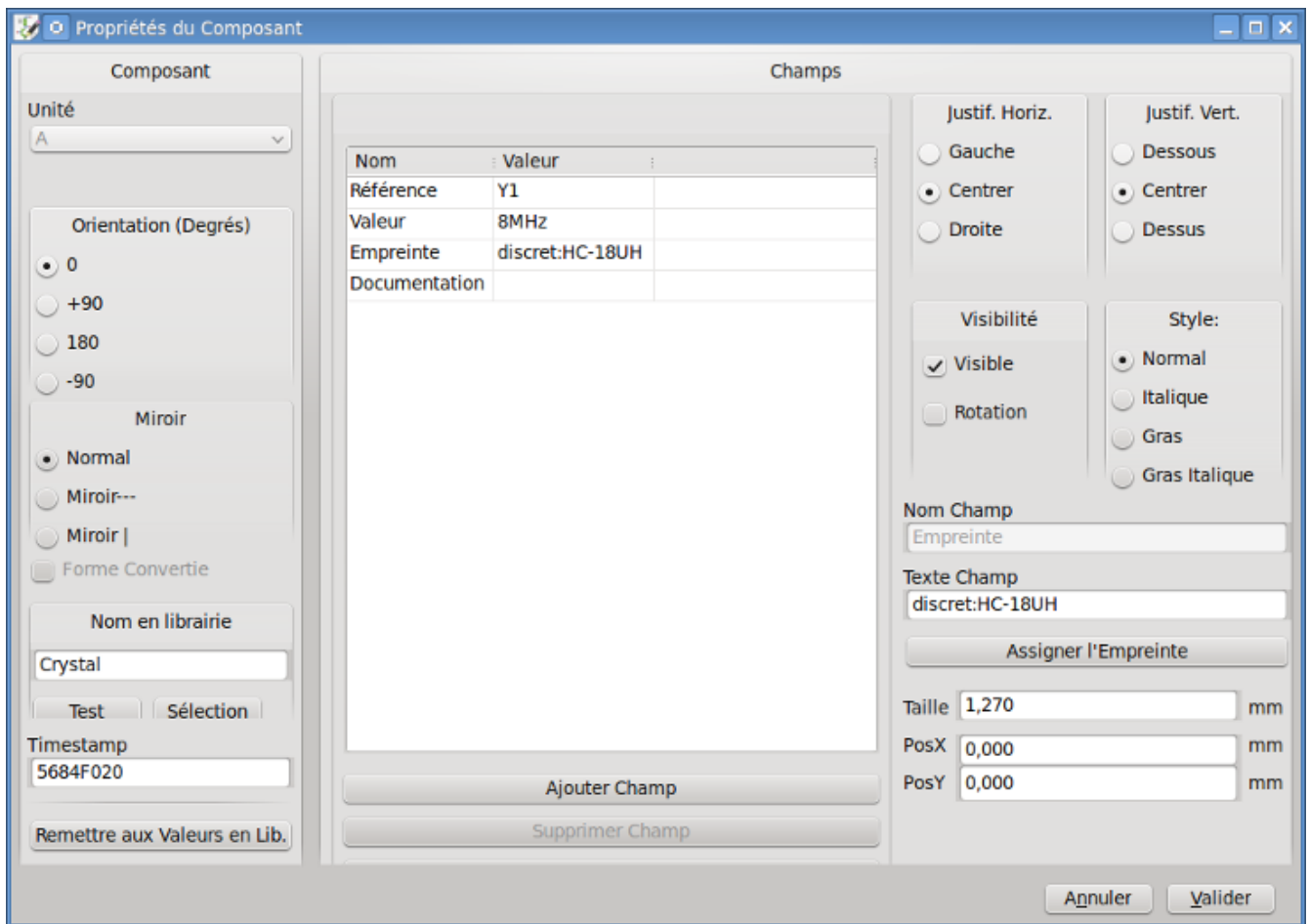


The Schematic Editor's BOM generator makes use of external plugins, either as XSLT or Python scripts. There are a few examples installed inside the KiCad program files directory.

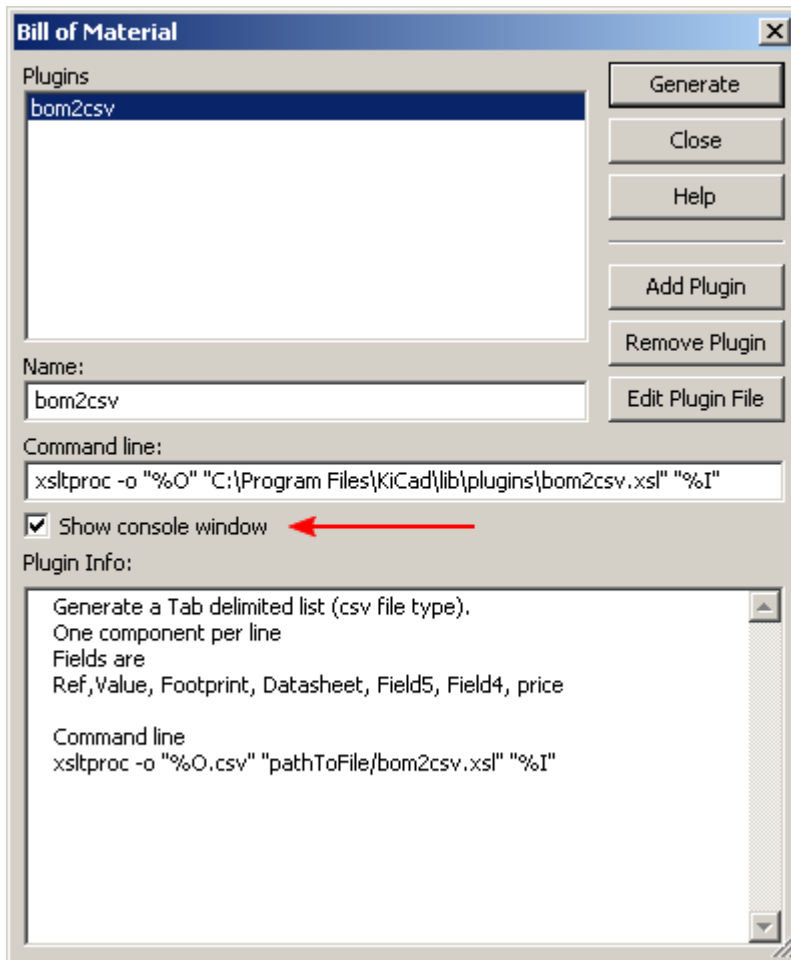
Quelques champs de composants utiles à utiliser pour le BOM :

- Valeur : nom unique pour chaque composant utilisé.
- Empreinte : entrée soit manuellement, soit par rétro-annotation (voir ci-dessous).
- Champ 1 : nom du fabricant.
- Champ 2 : référence fabricant.
- Champ 3 : référence distributeur.

Exemple :

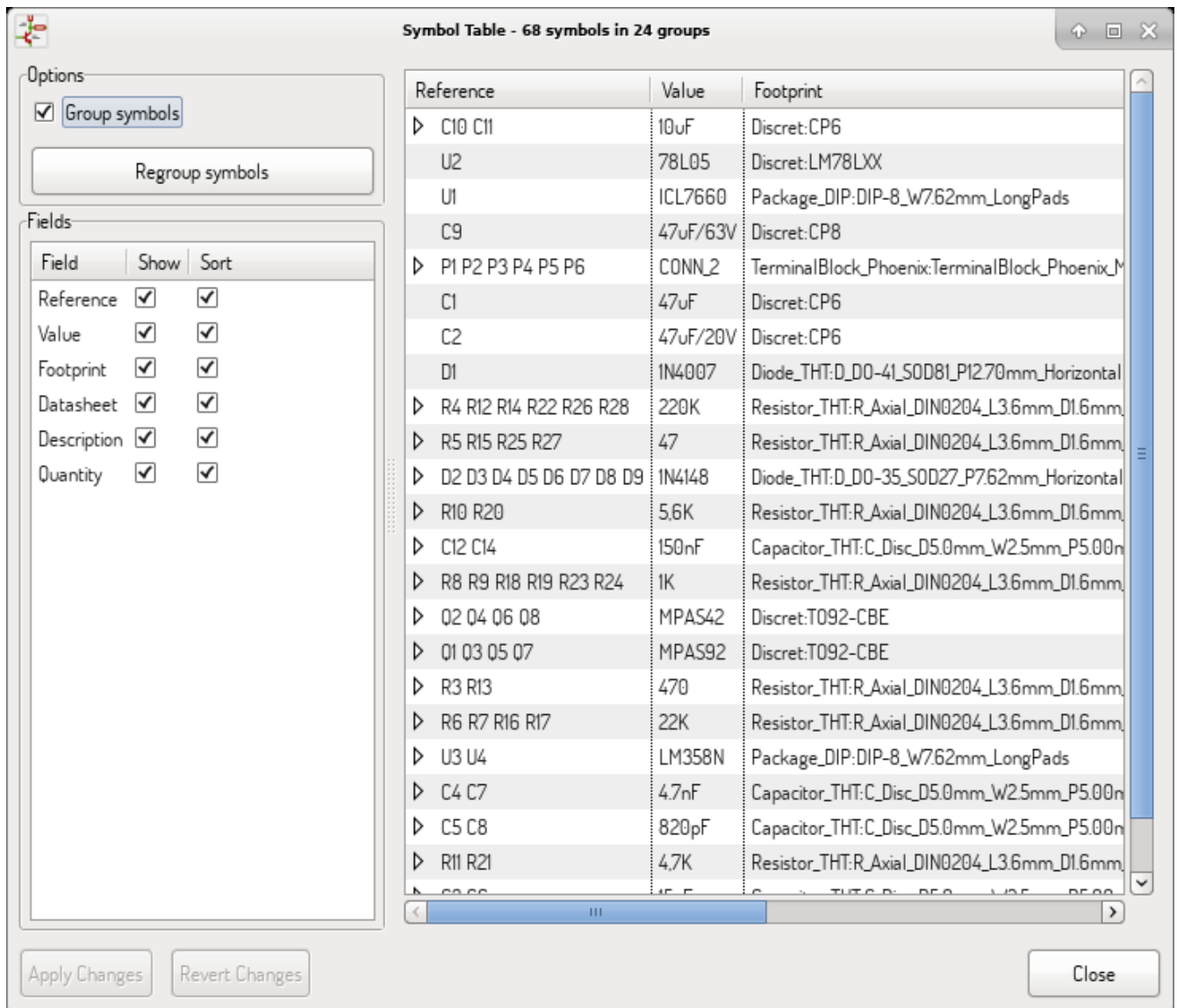


On **MS Windows**, BOM generator dialog has a special option (pointed by red arrow) that controls visibility of external plugin window. + By default, BOM generator command is executed console window hidden and output is redirected to *Plugin info* field. Set this option to show the window of the running command. It may be necessary if plugin has provides a graphical user interface.



## Edit Fields tool

The icon  opens a spreadsheet to view and modify field values for all symbols.



Once you modify field values, you need to either accept changes by clicking on 'Apply' button or undo them by clicking on 'Revert' button.

## Tricks to simplify fields filling

There are several special copy/paste methods in spreadsheet. They may be useful when entering field values that are repeated in a few components.

These methods are illustrated below.

Copy (Ctrl+C)	Selection	Paste (Ctrl+V)																																													
<table> <tr><td>abc</td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	abc															<table> <tr><td>abc</td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	abc															<table> <tr><td>abc</td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	abc														
abc																																															
abc																																															
abc																																															
<table> <tr><td>11</td><td>12</td><td>13</td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12	13													<table> <tr><td>11</td><td>12</td><td>13</td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12	13													<table> <tr><td>11</td><td>12</td><td>13</td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12	13												
11	12	13																																													
11	12	13																																													
11	12	13																																													
<table> <tr><td>11</td><td></td><td></td></tr> <tr><td>21</td><td></td><td></td></tr> <tr><td>31</td><td></td><td></td></tr> <tr><td>41</td><td></td><td></td></tr> <tr><td>51</td><td></td><td></td></tr> </table>	11			21			31			41			51			<table> <tr><td>11</td><td></td><td></td></tr> <tr><td>21</td><td></td><td></td></tr> <tr><td>31</td><td></td><td></td></tr> <tr><td>41</td><td></td><td></td></tr> <tr><td>51</td><td></td><td></td></tr> </table>	11			21			31			41			51			<table> <tr><td>11</td><td>11</td><td>11</td></tr> <tr><td>21</td><td>21</td><td>21</td></tr> <tr><td>31</td><td>31</td><td>31</td></tr> <tr><td>41</td><td>41</td><td>41</td></tr> <tr><td>51</td><td>51</td><td>51</td></tr> </table>	11	11	11	21	21	21	31	31	31	41	41	41	51	51	51
11																																															
21																																															
31																																															
41																																															
51																																															
11																																															
21																																															
31																																															
41																																															
51																																															
11	11	11																																													
21	21	21																																													
31	31	31																																													
41	41	41																																													
51	51	51																																													
<table> <tr><td>11</td><td>12</td><td></td></tr> <tr><td>21</td><td>22</td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12		21	22											<table> <tr><td>11</td><td>12</td><td></td></tr> <tr><td>21</td><td>22</td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12		21	22											<table> <tr><td>11</td><td>12</td><td></td></tr> <tr><td>21</td><td>22</td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12		21	22										
11	12																																														
21	22																																														
11	12																																														
21	22																																														
11	12																																														
21	22																																														
<table> <tr><td>11</td><td>12</td><td>13</td></tr> <tr><td>21</td><td>22</td><td>23</td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12	13	21	22	23										<table> <tr><td>11</td><td>12</td><td>13</td></tr> <tr><td>21</td><td>22</td><td>23</td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12	13	21	22	23										<table> <tr><td>11</td><td>12</td><td>13</td></tr> <tr><td>21</td><td>22</td><td>23</td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> <tr><td></td><td></td><td></td></tr> </table>	11	12	13	21	22	23									
11	12	13																																													
21	22	23																																													
11	12	13																																													
21	22	23																																													
11	12	13																																													
21	22	23																																													

#### NOTE

These techniques are also available in other dialogs with a grid control element.

## Import tool for footprint assignment

### Accès :

The icon  launches the back-annotate tool.

This tool allows footprint changes made in the PCB Editor to be imported back into the footprint fields in the Schematic Editor.

# Manage Symbol Libraries

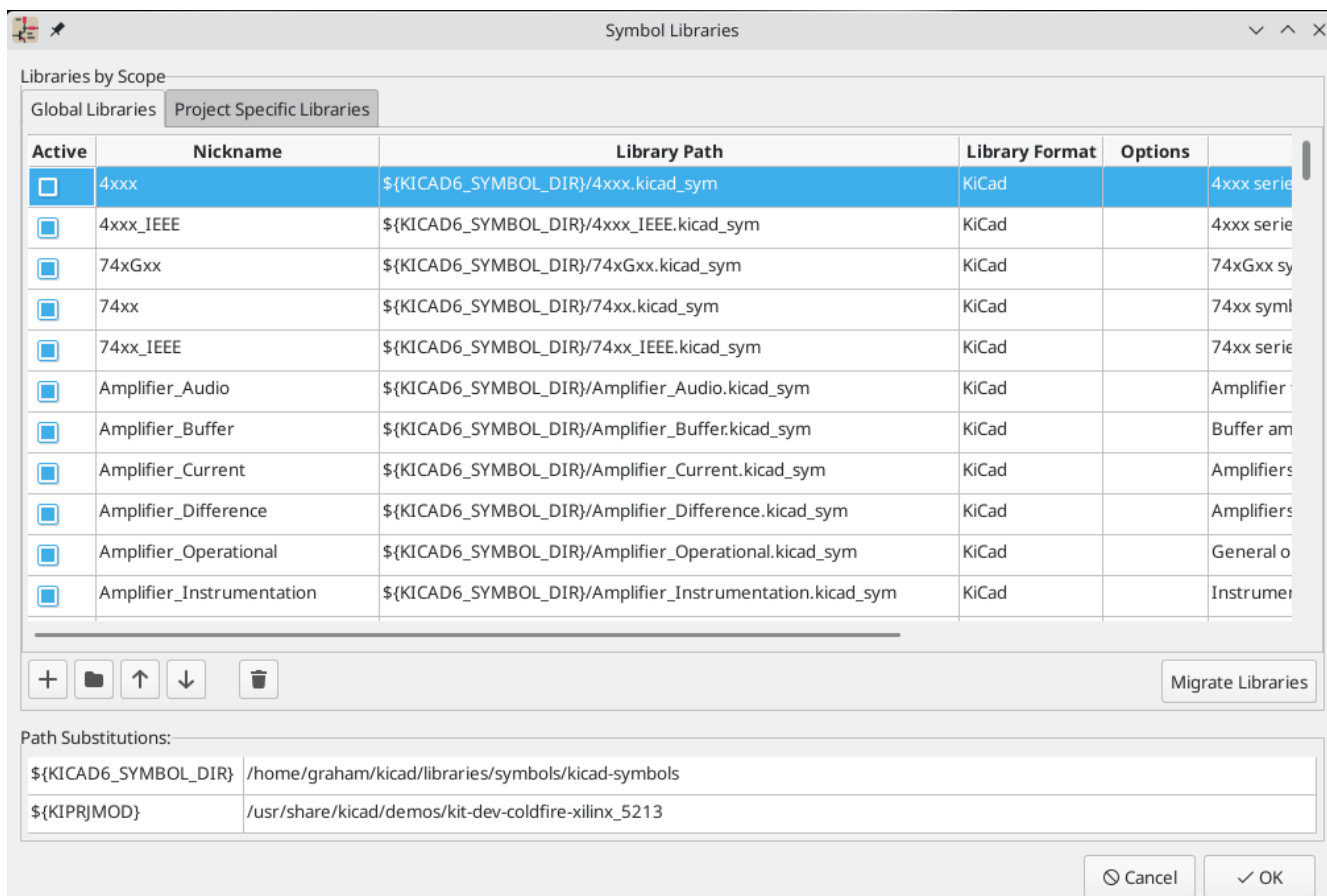
Symbol libraries hold collections of symbols used when creating schematics. Each symbol in a schematic is uniquely identified by a full name that is composed of a library nickname and a symbol name. An example is `Audio:AD1853`.

## Symbol Library Table

The symbol library table holds a list of all library files KiCad knows about. The symbol library table is constructed from the global symbol library table file and the project specific symbol library table file.

When a symbol is loaded, KiCad uses the library nickname, `Audio` in our example, to lookup the library location in the symbol library table.

The image below shows the symbol library table editing dialog which can be opened by invoking the **Manage Symbol Libraries...** entry in the **Preferences** menu.



## Global Symbol Library Table

The global symbol library table contains the list of libraries that are always available regardless of the currently loaded project file. The table is saved in the file `sym-lib-table` in the user's KiCad configuration folder. The [location of this folder](#) is dependent upon the operating system being used.

## Project Specific Symbol Library Table

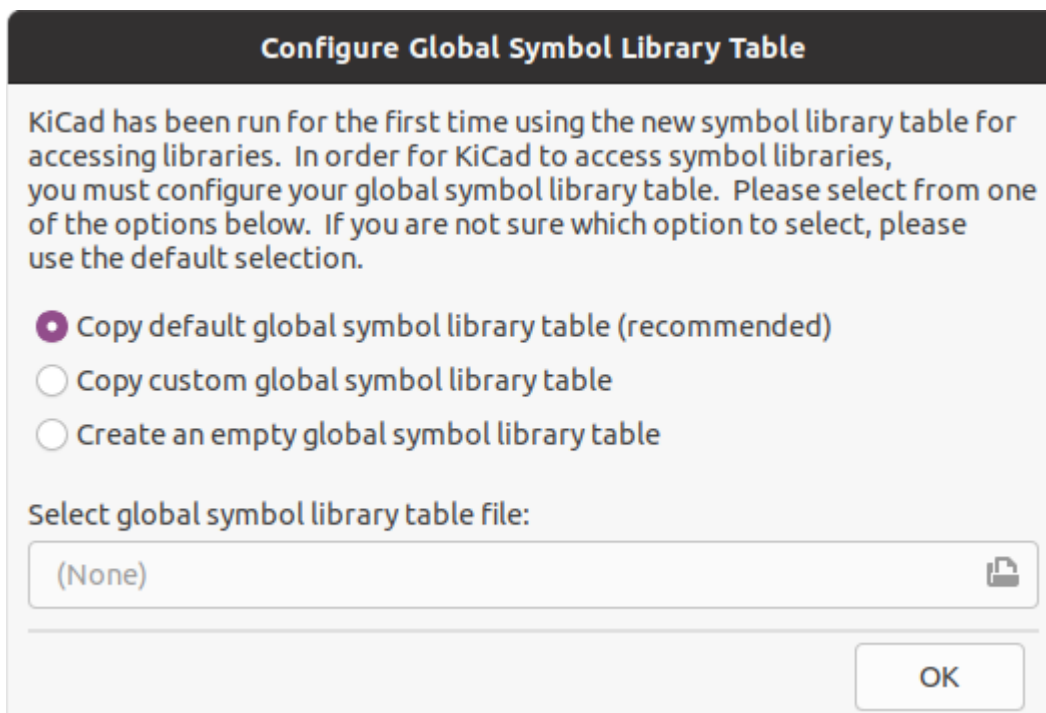
The project specific symbol library table contains the list of libraries that are available specifically for the currently loaded project file. The project specific symbol library table can only be edited when it is loaded

along with the project file. If no project file is loaded or there is no symbol library table file in the current project path, an empty table is created which can be edited and later saved along with the project file.

## Initial Configuration

The first time the KiCad Schematic Editor is run and the global symbol table file `sym-lib-table` is not found in the KiCad configuration folder, KiCad will present the "Configure Global Symbol Library Table" dialog to the user. The dialog presents the user with three options.

- **Copy default global symbol library table (recommended).** If this option is selected, KiCad will copy the default symbol library table file stored in the system's Kicad template folder to the file `sym-lib-table` in the user's KiCad configuration folder. If the default template `sym-lib-table` file cannot be found, this option will be grayed out. The missing default table is usually caused by the KiCad default libraries not being installed (on some systems they are installed by a separate package). If the libraries are installed in a non-standard location, use the second option and browse to the library table location manually.
- **Copy custom global symbol library table.** If this option is selected, the user must browse to the desired symbol library table file, which will be copied to the user's KiCad configuration directory.
- **Create an empty global symbol library table.** An empty symbol library table file will be created in the user's KiCad configuration directory. The user must add libraries to the table manually.



### NOTE

The default symbol library table includes all of the symbol libraries that are installed as part of KiCad. This may or may not be desirable depending on usages and the speed of the system. The amount of time required to load the symbol libraries is proportional to the number of libraries in the symbol library table. If symbol library load times are excessive, remove rarely and/or never used libraries from the global library table and add them to the project library table as required.

## Adding Table Entries

In order to use a symbol library, it must first be added to either the global table or the project specific table. The project specific table is only applicable when you have a project file open.

**NOTE** | Each library entry must have a unique nickname.

The library nickname does not have to be related in any way to the actual library file name or path. The colon `:` and `\` characters cannot be used anywhere in the library nickname. Each library entry must have a valid path and/or file name depending on the type of library. Paths can be defined as absolute, relative, or by environment variable substitution (see section below).

The appropriate library format must be selected in order for the library to be properly read. "KiCad" format is used for KiCad version 6 libraries ( `.kicad_sym` files), while "Legacy" format is used for libraries from older versions of KiCad ( `.lib` files). Legacy libraries are read-only, but can be migrated to KiCad format libraries using the **Migrate Libraries** button (see section [Migrating Legacy Libraries](#)).

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

- Please note that you cannot have duplicate library nicknames in the same table. However, you can have duplicate library nicknames in both the global and project specific symbol library table.
- The project specific table entry will take precedence over the global table entry when duplicate nicknames occur.
- When entries are defined in the project specific table, a `sym-lib-table` file containing the entries will be written into the folder of the currently open project file.

## Environment Variable Substitution

One of the most powerful features of the symbol library table is environment variable substitution. This allows for definition of custom paths to where symbol libraries are stored in environment variables. Environment variable substitution is supported by using the syntax `${ENV_VAR_NAME}` in the library path.

By default, at run time KiCad defines two environment variables relevant for locating symbol libraries:

- the `$KIPRJMOD` environment variable that always points to the currently open project directory. `$KIPRJMOD` cannot be modified.
- the `$KICAD6_SYMBOL_DIR` environment variable. This points to the path where the default symbol libraries that were installed with KiCad.

You can override `$KICAD6_SYMBOL_DIR` by redefining it in **Preferences** → **Configure Paths...** This is useful for using libraries installed in a nonstandard location.

`$KIPRJMOD` allows you to store libraries in the project path without having to define the absolute path (which is not always known) to the library in the project specific symbol library table.

## Usage Patterns

Symbol libraries can be defined either globally or specifically to the currently loaded project. Symbol libraries defined in the user's global table are always available and are stored in the `sym-lib-table` file in



the user's KiCad configuration folder. The project-specific symbol library table is active only for the currently open project file.

There are advantages and disadvantages to each method. Defining all libraries in the global table means they will always be available when needed. The disadvantage of this is that load time will increase.

Defining all symbol libraries on a project specific basis means that you only have the libraries required for the project which decreases symbol library load times. The disadvantage is that you always have to remember to add each symbol library that you need for every project.

One usage pattern would be to define commonly used libraries globally and the libraries only required for the project in the project specific library table. There is no restriction on how to define libraries.

## Migrating Legacy Libraries

Legacy libraries ( `.lib` files) are read-only, but they can be migrated to KiCad version 6 libraries ( `.kicad_sym` ). KiCad version 6 libraries cannot be viewed or edited by KiCad versions older than 6.0.0.

Legacy libraries can be converted to KiCad 6 libraries by selecting them in the symbol library table and clicking the **Migrate Libraries** button. Multiple libraries can be selected and migrated at once by `Ctrl`-clicking or `shift`-clicking.

Libraries can also be converted one at a time by opening them in the Symbol Editor and saving them as a new library.

## Legacy Project Remapping

When loading a schematic created prior to the symbol library table implementation, KiCad will attempt to remap the symbol library links in the schematic to the appropriate library table symbols. The success of this process is dependent on several factors:

- the original libraries used in the schematic are still available and unchanged from when the symbol was added to the schematic.
- all rescue operations were performed when detected to create a rescue library or keep the existing rescue library up to date.
- the integrity of the project symbol cache library has not been corrupted.

### WARNING

The remapping will make a back up of all the files that are changed during remapping in the rescue-backup folder in the project folder. Always make a back up of your project before remapping just in case something goes wrong.

### WARNING

The rescue operation is performed even if it has been disabled to ensure the correct symbols are available for remapping. Do not cancel this operation or the remapping will fail to correctly remap schematics symbols. Any broken symbol links will have to be fixed manually.

**NOTE**

If the original libraries have been removed and the rescue was not performed, the cache library can be used as a recovery library as a last resort. Copy the cache library to a new file name and add the new library file to the top of the library list using a version of KiCad prior to the symbol library table implementation.

# Création et édition de schémas

## Introduction

Un schéma peut être représenté sur une seule feuille, mais, s'il est assez grand, il lui faudra plusieurs feuilles.

A schematic represented by several sheets is hierarchical, and all its sheets (each one represented by its own file) constitute a complete KiCad schematic. The manipulation of hierarchical schematics will be described in the [Hierarchical Schematics](#) chapter.

## Généralités

A schematic designed with KiCad is more than a simple graphic representation of an electronic device. It is normally the entry point of a development chain that allows for:


- Valider un ensemble de règles ([Vérification des règles électriques \(ERC\)](#)) pour détecter les erreurs et omissions.
- Générer automatiquement une liste de composants ([BOM](#)).
- [Générer une netliste](#) pour des logiciels de simulation, comme SPICE.
- [Defining a circuit](#) for transferring to PCB layout.

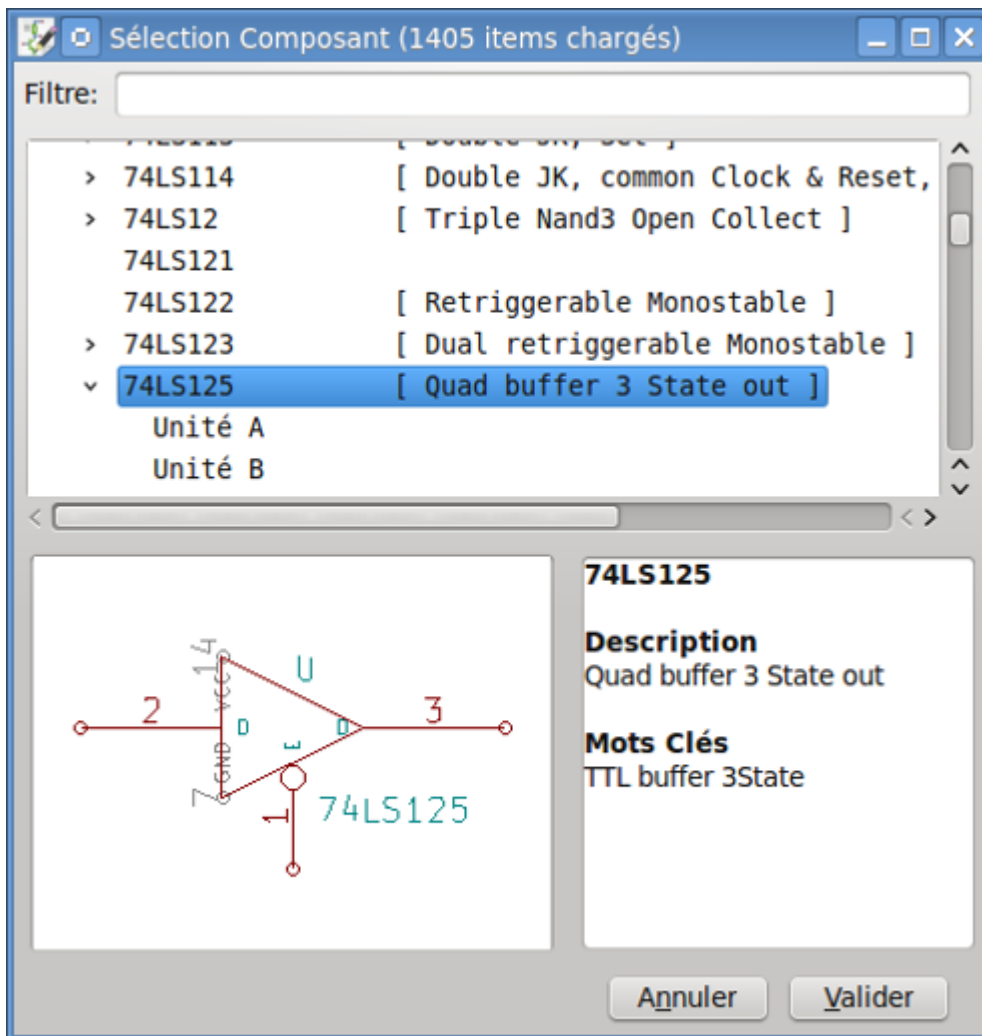
A schematic mainly consists of symbols, wires, labels, junctions, buses and power ports. For clarity in the schematic, you can place purely graphical elements like bus entries, comments, and polylines.

Symbols are added to the schematic from symbol libraries. After the schematic is made, the set of connections and footprints is imported into the PCB editor for designing a board.

## Symbol placement and editing

### Find and place a symbol

To load a symbol into your schematic you can use the icon . A dialog box allows you to type the name of the symbol to load.



The Choose Symbols dialog will filter symbols by name, keywords, and description according to what you type into the search field. Advanced filters can be used just by typing them:

- **Wildcards:** use the characters `?` and `*` respectively to mean "any character" and "any number of characters".
- **Relational:** if a library part's description or keywords contain a tag of the format "Key:123", you can match relative to that by typing "Key>123" (greater than), "Key<123" (less than), etc. Numbers may include one of the following case-insensitive suffixes:

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

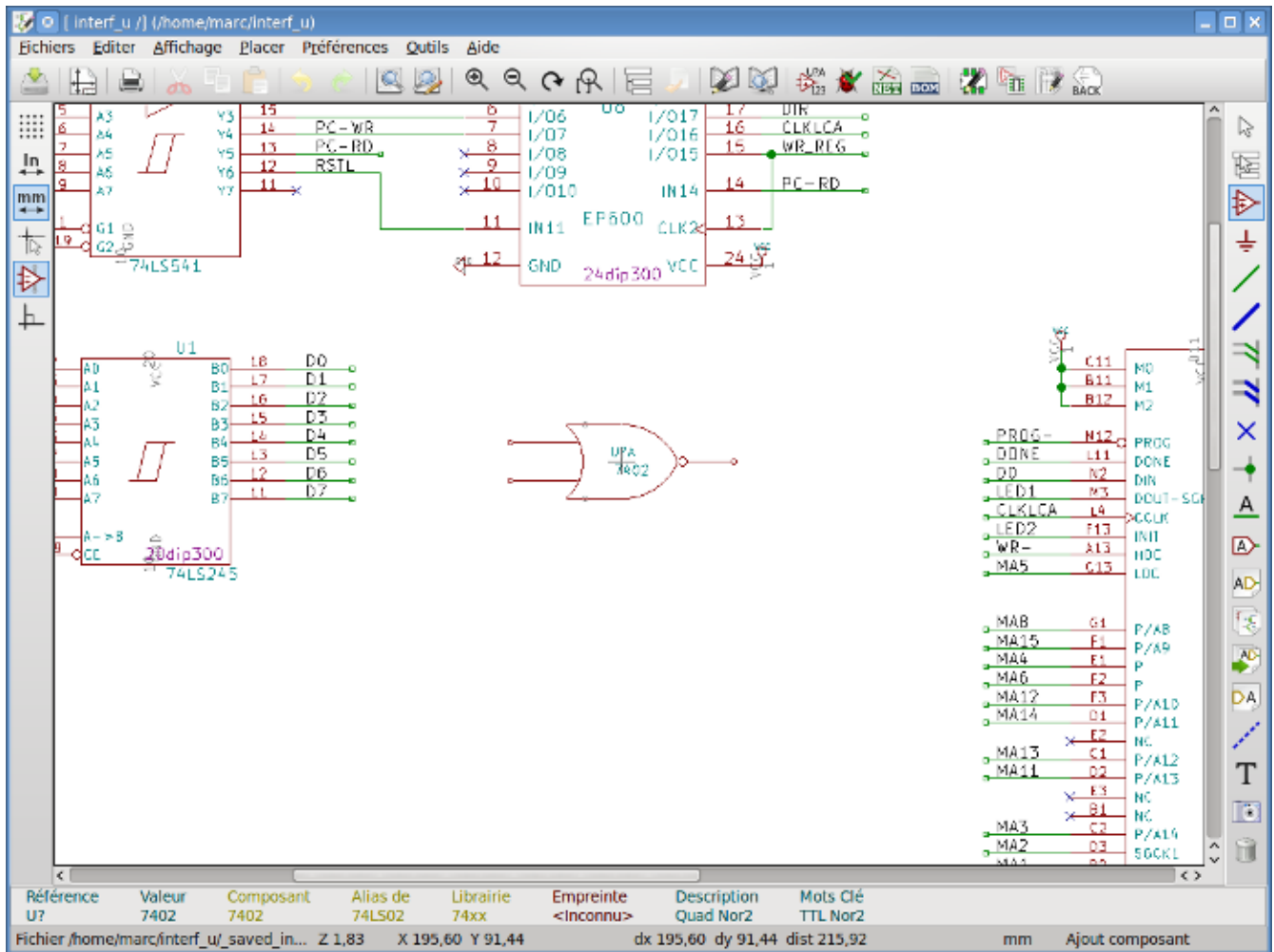
ki	mi	gi	ti
$2^{10}$	$2^{20}$	$2^{30}$	$2^{40}$

- **Regular expression:** if you're familiar with regular expressions, these can be used too. The regular expression flavor used is the [wxWidgets Advanced Regular Expression style](#), which is similar to Perl regular expressions.

If the symbol specifies a default footprint, this footprint will be previewed in the lower right. If the symbol includes footprint filters, alternate footprints that satisfy the footprint filters can be selected in the footprint dropdown menu at right.

After selecting a symbol to place, the symbol will be attached to the cursor. Left clicking the desired location in the schematic places the symbol into the schematic. Before placing the symbol in the schematic, you can rotate it, mirror it, and edit its fields, by either using the hotkeys or the right-click context menu. These actions can also be performed after placement.


Here is a symbol during placement:



If the "Place repeated copies" option is checked, after placing a symbol KiCad will start placing another copy of the symbol. This process continues until the user presses **Esc**.

For symbols with multiple units, if the "Place all units" option is checked, after placing the symbol KiCad will start placing the next unit in the symbol. This continues until the last unit has been placed or the user presses **Esc**.

## Placing power ports

A [power port symbol](#) is a symbol representing a connection to a power net. The symbols are grouped in the **power** library, so they can be placed using the symbol chooser. However, as power placements are frequent, the  tool is available. This tool is similar, except that the search is done directly in the **power** library.

## Symbol Editing and Modification (already placed component)

There are two ways to edit a symbol:

- Modification of the symbol itself: position, orientation, unit selection on a multi-unit symbol.
- Modification of one of the fields of the symbol: reference, value, footprint, etc.

When a symbol has just been placed, you may have to modify its value (particularly for resistors, capacitors, etc.), but it is useless to assign to it a reference number right away, or to select the unit (except for components with locked units, which you have to assign manually). This can be done automatically by the annotation function.

### Symbol modification

To modify some feature of a symbol, position the cursor on the symbol, and then either:

- Double-click on the symbol to open the full editing dialog.
- Faites un clic droit pour ouvrir le menu contextuel et choisissez l'une des commandes : Déplacer, Orienter, Éditer, Supprimer, etc...
- Use a hotkey to perform an action on the symbol (**E** to open the properties dialog, **R** to rotate, etc.). Note that hotkeys act on the selected symbol; if no symbol is selected hotkeys act on the symbol under the cursor.

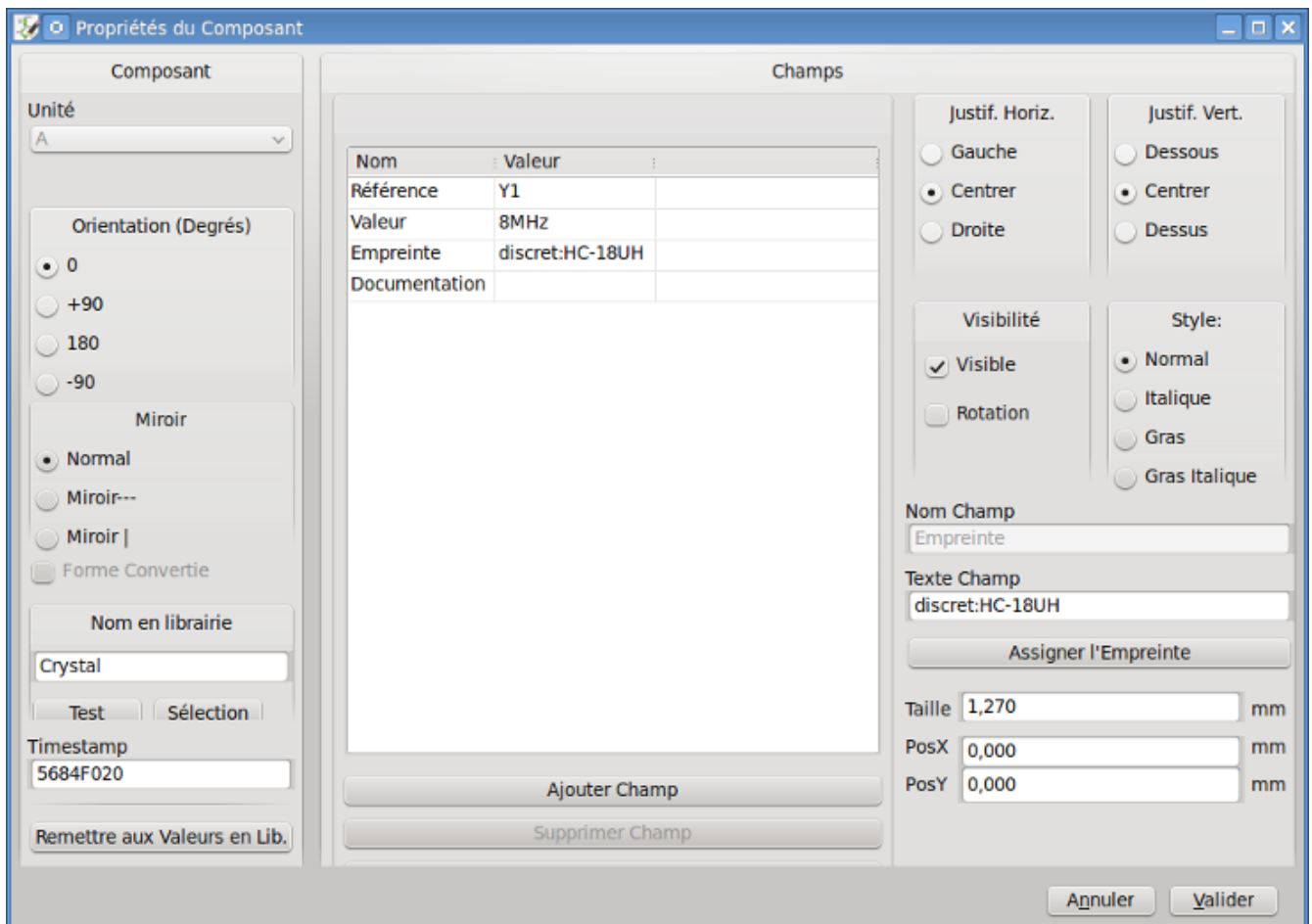
Symbols can also be selected by clicking on them or drag-selecting them. Selected symbols can be modified by clicking relevant buttons in the top toolbar or using a hotkey.

### Édition des champs du composant

Vous pouvez modifier la référence, la valeur, la position, l'orientation, la taille du texte et la visibilité des champs :

- Double-cliquez sur le champ à modifier.
- Faites un clic droit pour ouvrir le menu contextuel et choisissez l'une des commandes : Déplacer, Orienter, Éditer, Supprimer, etc...
- Position the cursor over the field (if nothing is selected) or select the field and press **E** to edit the field.
- Position the cursor over the symbol (if nothing is selected) or select the symbol and press **V**, **U**, or **F** hotkeys to directly edit the symbol's value, reference designator, or footprint fields, respectively.

For more options, or in order to create fields, double-click on the symbol to open the Symbol Properties dialog.



Each field can be visible or hidden, and displayed horizontally or vertically. The displayed position is always indicated for a normally displayed symbol (no rotation or mirroring) and is relative to the anchor point of the symbol.

The position and orientation properties of each field may be hidden in this dialog. They can be shown by right-clicking on the column header of the fields table and enabling the "Orientation", "X Position", and/or "Y Position" columns. Other columns can be shown or hidden as desired.

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.

## Electrical Connections

### Introduction

There are a number of elements that can be added to a schematic to electrically connect components. All of these elements can be placed with the buttons on the vertical right toolbar or using hotkeys.

Ces éléments peuvent être des :

- **Wires:** direct connection between pins.

**Buses:** connections for a group of signals.

- **Bus entries:** connections between wires and buses.
- **No-connection flags:** terminations for pins or wires that are intentionally unconnected. These flags prevent ERC violations for unconnected pins.
- **Junctions:** connections between crossing wires or buses.
- **Net labels:** local name for a signal. Signals within a sheet that have the same net label are connected.
- **Global labels:** global name for a signal. Signals with the same global label are connected even if they are not in the same sheet.
- **Hierarchical labels:** a label for a signal in a subsheet that enables the signal to be accessed in a parent sheet. See the [Hierarchical Schematics](#) section for more information about hierarchical labels, sheets, and pins.
- **Hierarchical sheets:** an instantiation of a subsheet within a parent sheet. The parent sheet can connect to the subsheet through the subsheet's hierarchical pins.
- **Hierarchical pins:** connection points between a parent sheet and a subsheet. Hierarchical pins appear at the parent sheet's level and correspond to hierarchical labels in the subsheet.

Several other types of items can be placed on the schematic but do not affect connectivity:

- **Graphical lines:** graphical lines for presentation.
- **Text:** textual comments and annotations.
- **Bitmap images:** raster graphics from an external file.

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

- **Power ports:** symbols for connecting wires to a power or ground net.
- **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).

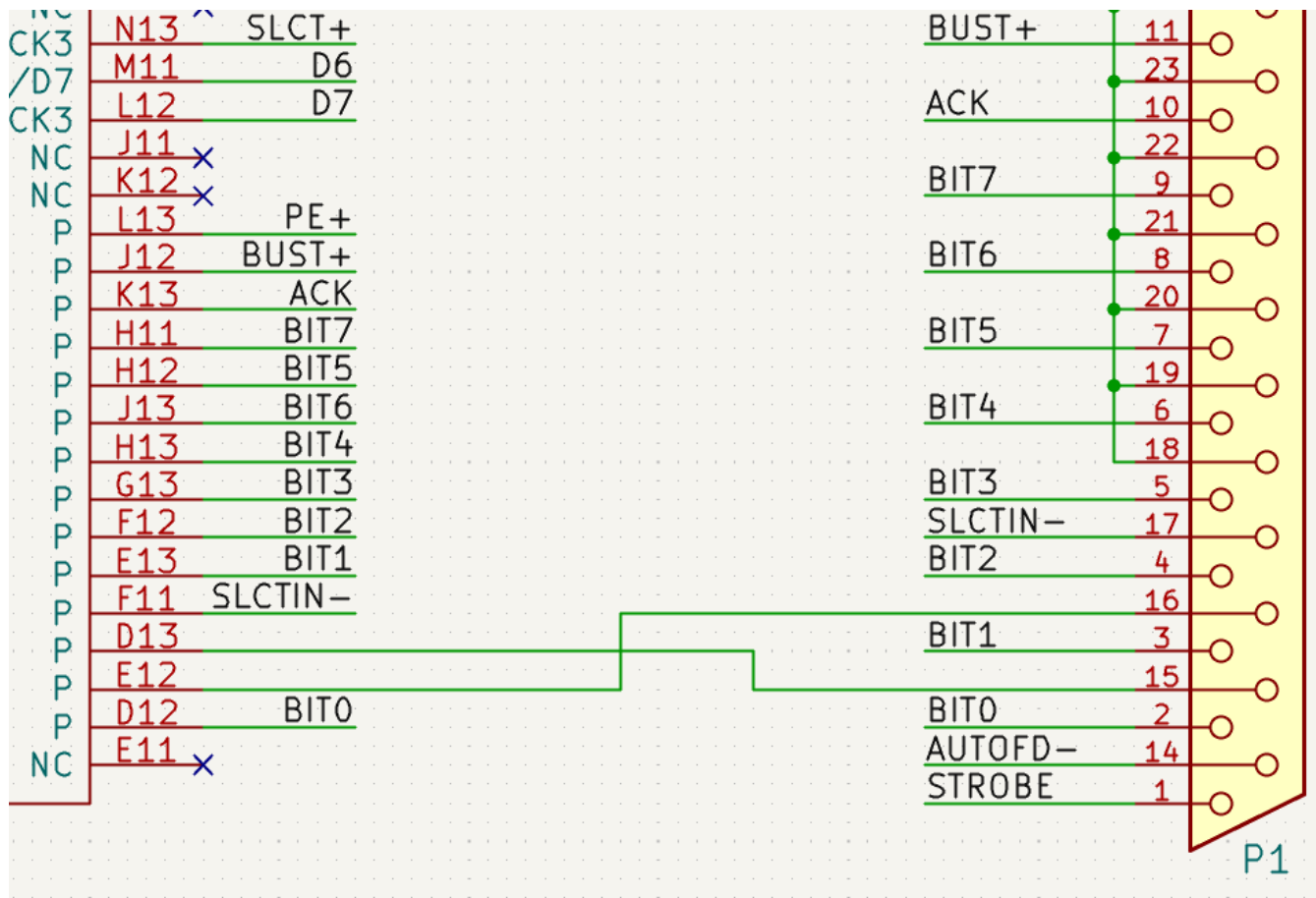
## Connexions (Fils et Labels)

Il y a deux moyens d'établir des connexions :

- Fils de pin à pin.
- Labels.

La figure ci-dessous montre les deux méthodes :





## Label Connections

The point of "contact" of a label is the small square in the corner of the label. The square disappears when the label is connected. The position of the connection point relative to the label text can be changed by choosing a different label orientation in the label properties, or by mirroring/rotating the label.

The label's connection point must be in contact with a wire or the end of a pin for the label to be connected.

## Wire Connections

Pour établir une connexion, un segment de fil doit être connecté par ses extrémités à un autre segment ou à une pin de composant.

Si il y a chevauchement (si un fil survole une pin sans être connecté à son extrémité), il n'y a pas de connexion.

### 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**.

## Wire Junctions

Wires that cross are not implicitly connected. It is necessary to join them with a junction dot if a connection is desired. Junction dots will be automatically added to wires that start or end on top of an existing wire.

Junction dots are used in the previous figure on the wires connected to P1 pins 18, 19, 20, 21, 22, and 23.

## Nets with Multiple Names

A signal can only have one name. If two different labels are placed on the same net, an ERC violation will be generated. Only one of the net names will be used in the netlist.

## 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 automatically be connected to the global VCC net.

### 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, the use of invisible power pins in symbols is not recommended outside of power port symbols, and is only supported for compatibility with legacy designs and symbols.

### 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 (options) 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 port symbol](#) for each net and connect them with a wire.

It is not recommended to use labels for power connection. These only have a "local" connection scope, and will not connect to invisible power pins.

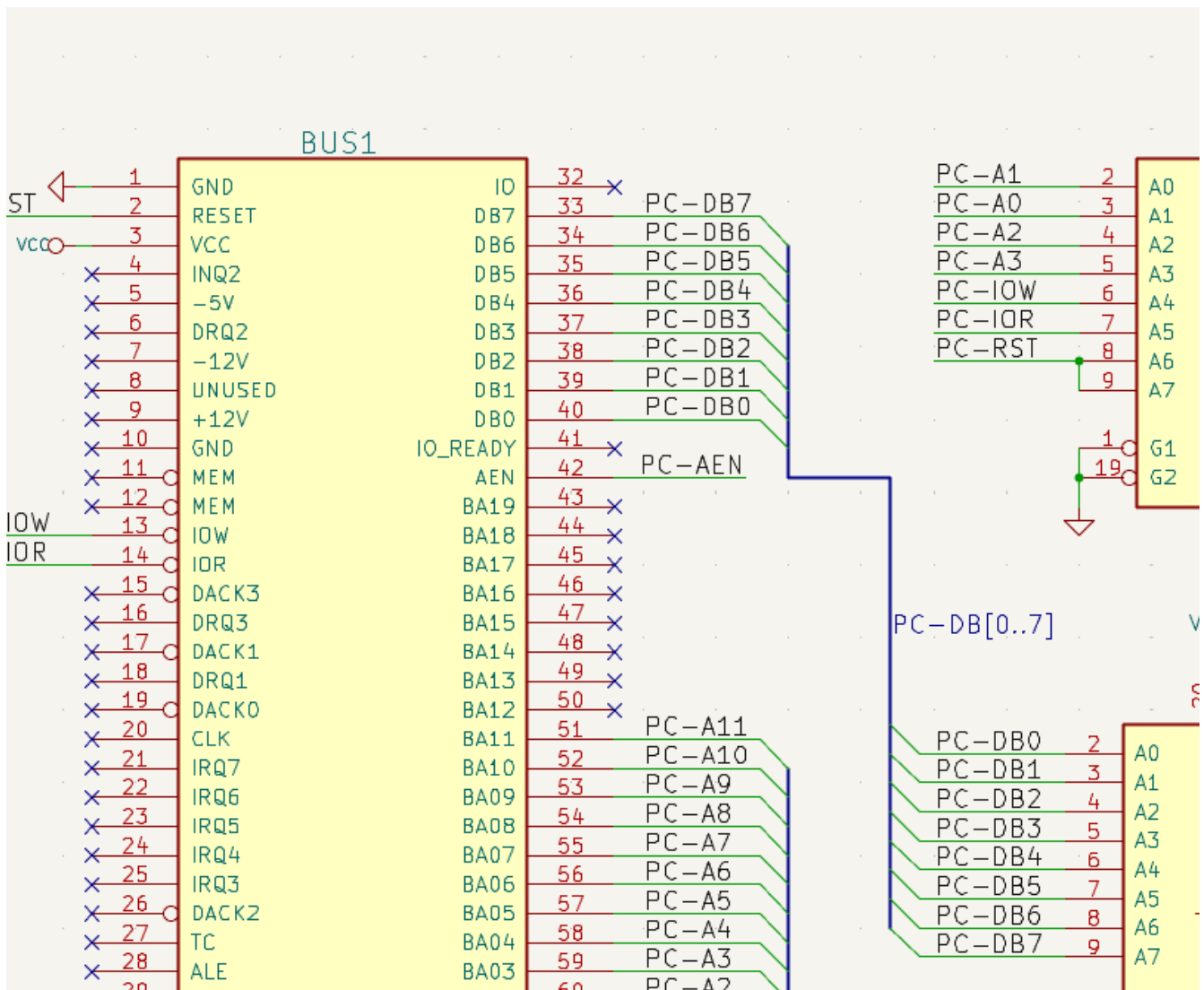
## Wiring

To begin connecting elements, you may either use the 'Wire' or 'Bus' tools from the right-hand toolbar, or you can auto-start a new wire from any existing pin or unconnected wire.

The wire drag action will drag the entire wire if you start dragging from the middle of the wire. Alternatively, it will drag just one corner if you start the drag action over a corner where two wires connect

## Connexions (Bus)

Dans le schéma ci-dessous, de nombreuses pins sont connectées à des bus.



## Membres d'un bus

Busés are a way to group related signals in the schematic in order to simplify complicated designs. Buses can be drawn like wires using the bus tool, and are named using labels the same way signal wires are. There are two types of bus in KiCad 6.0 and later: vector buses and group buses.

A **vector bus** is a collection of signals that start with a common prefix and end with a number. Vector buses are named `<PREFIX>[M..N]` where `PREFIX` is any valid signal name, `M` is the first suffix number, and `N` is the last suffix number. For example, the bus `DATA[0..7]` contains the signals `DATA0`, `DATA1`, and so on up to `DATA7`. It doesn't matter which order `M` and `N` are specified in, but both must be non-negative.

A **group bus** is a collection of one or more signals and/or vector buses. Group buses can be used to bundle together related signals even when they have different names. Group buses use a special label syntax:

```
<OPTIONAL_NAME>{SIGNAL1 SIGNAL2 SIGNAL3}
```

The members of the group are listed inside curly braces ( `{ }` ) separated by space characters. An optional name for the group goes before the opening curly brace. If the group bus is unnamed, the resulting nets on the PCB will just be the signal names inside the group. If the group bus has a name, the resulting nets will have the name as a prefix, with a period ( `.` ) separating the prefix from the signal name.

For example, the bus `{SCL SDA}` has two signal members, and in the netlist these signals will be `SCL` and `SDA`. The bus `USB1{DP DM}` will generate nets called `USB1.DP` and `USB1.DM`. For designs with larger buses

that are repeated across several similar circuits, using this technique can save time.

Group buses can also contain vector buses. For example, the bus `MEMORY{A[7..0] D[7..0] OE WE}` contains both vector buses and plain signals, and will result in nets such as `MEMORY.A7` and `MEMORY.OE` on the PCB.

Bus wires can be drawn and connected in the same manner as signal wires, including using junctions to create connections between crossing wires. Like signals, buses cannot have more than one name — if two conflicting labels are attached to the same bus, an ERC violation will be generated.

## Connexions entre membres de bus

Pins connected between the same members of a bus must be connected by labels. It is not possible to connect a pin directly to a bus; this type of connection will be ignored by KiCad.

Dans l'exemple ci-dessus, les connexions sont faites par des labels placés sur les fils connectés aux pins. Les entrées de bus (segments de fil à 45 degrés) sont purement décoratifs, et ne sont pas nécessaires pour établir des connexions logiques.

In fact, using the repetition command (`Insert`), connections can be very quickly made in the following way, if component pins are aligned in increasing order (a common case in practice on components such as memories, microprocessors...):

- Place the first label (for example `PCA0`)
- Use the repetition command as much as needed to place members. KiCad will automatically create the next labels (`PCA1`, `PCA2` ...) vertically aligned, theoretically on the position of the other pins.
- Dessinez le fil sous le premier label. Ensuite, utilisez la commande de répétition pour placer les autres fils sous les autres labels.
- Au besoin, placez les entrées de bus de la même façon (placez la première entrée, puis utilisez la commande de répétition).

### NOTE

In the **Schematic Editor** → **Editing Options** section of the Preferences menu, you can set the repetition parameters:

- Horizontal pitch.
- Vertical pitch.
- Label increment (labels can be incremented or decremented by 1, 2, 3, etc.).

## Bus unfolding

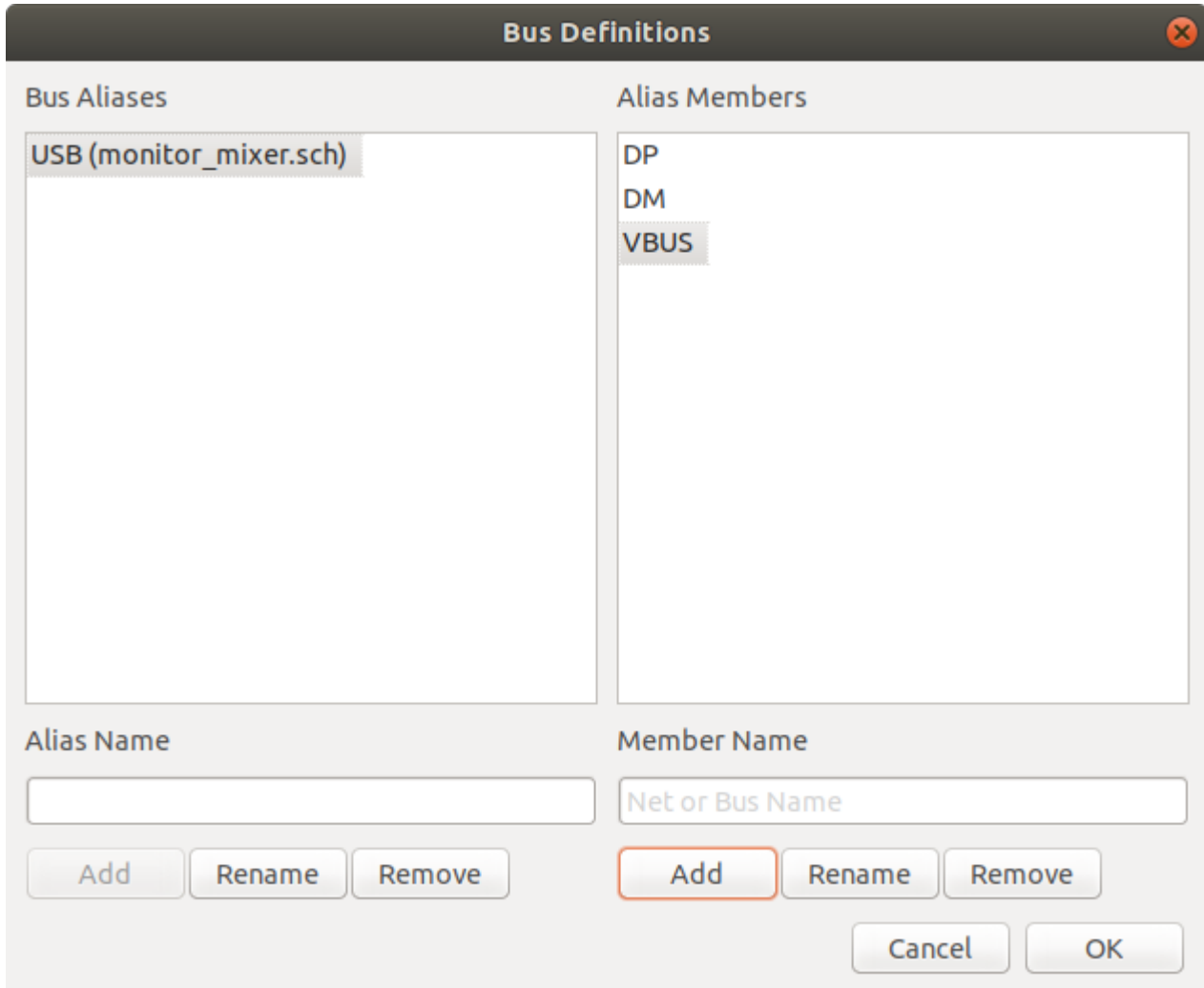
The unfold tool allows you to quickly break out signals from a bus. To unfold a signal, right-click on a bus object (a bus wire, etc) and choose **Unfold from Bus**. Alternatively, use the **Unfold Bus** hotkey (default: `C`) when the cursor is over a bus object. The menu allows you to select which bus member to unfold.

After selecting the bus member, the next click will place the bus member label at the desired location. The tool automatically generates a bus entry and wire leading up to the label location. After placing the label, you can continue placing additional wire segments (for example, to connect to a component pin) and complete the wire in any of the normal ways.

## Bus aliases

Bus aliases are shortcuts that allow you to work with large group buses more efficiently. They allow you to define a group bus and give it a short name that can then be used instead of the full group name across the schematic.

To create bus aliases, open the **Bus Definitions** dialog in the **Tools** menu.



An alias may be named any valid signal name. Using the dialog, you can add signals or vector buses to the alias. As a shortcut, you can type or paste in a list of signals and/or buses separated by spaces, and they will all be added to the alias definition. In this example, we define an alias called **USB** with members **DP**, **DM**, and **VBUS**.

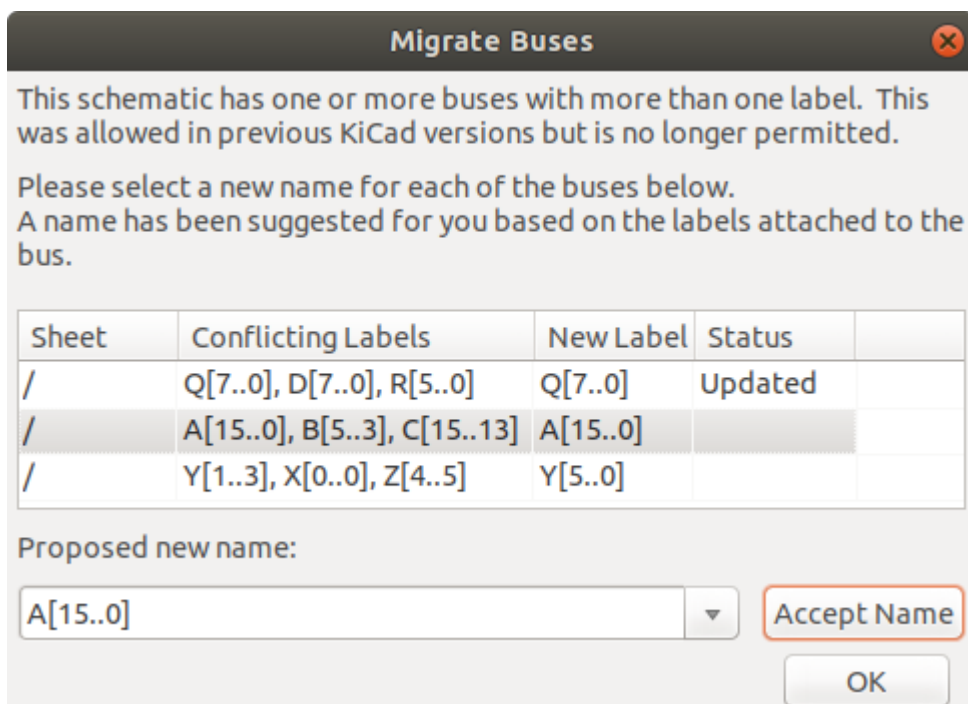
After defining an alias, it can be used in a group bus label by putting the alias name inside the curly braces of the group bus: **{USB}**. This has the same effect as labeling the bus **{DP DM VBUS}**. You can also add a prefix name to the group, such as **USB1{USB}**, which results in nets such as **USB1.DP** as described above. For complicated buses, using aliases can make the labels on your schematic much shorter. Keep in mind that the aliases are just a shortcut, and the name of the alias is not included in the netlist.

Bus aliases are saved in the schematic file. Any aliases created in a given schematic sheet are available to use in any other schematic sheet that is in the same hierarchical design.

## Buses with more than one label

KiCad 5.0 and earlier allowed the connection of bus wires with different labels together, and would join the members of these buses during netlisting. This behavior has been removed in KiCad 6.0 because it is incompatible with group buses, and also leads to confusing netlists because the name that a given signal will receive is not easily predicted.

If you open a design that made use of this feature in a modern version of KiCad, you will see the Migrate Buses dialog which guides you through updating the schematic so that only one label exists on any given set of bus wires.



Sheet	Conflicting Labels	New Label	Status	
/	Q[7..0], D[7..0], R[5..0]	Q[7..0]	Updated	
/	A[15..0], B[5..3], C[15..13]	A[15..0]		
/	Y[1..3], X[0..0], Z[4..5]	Y[5..0]		

Proposed new name:

A[15..0] ▼ Accept Name

OK

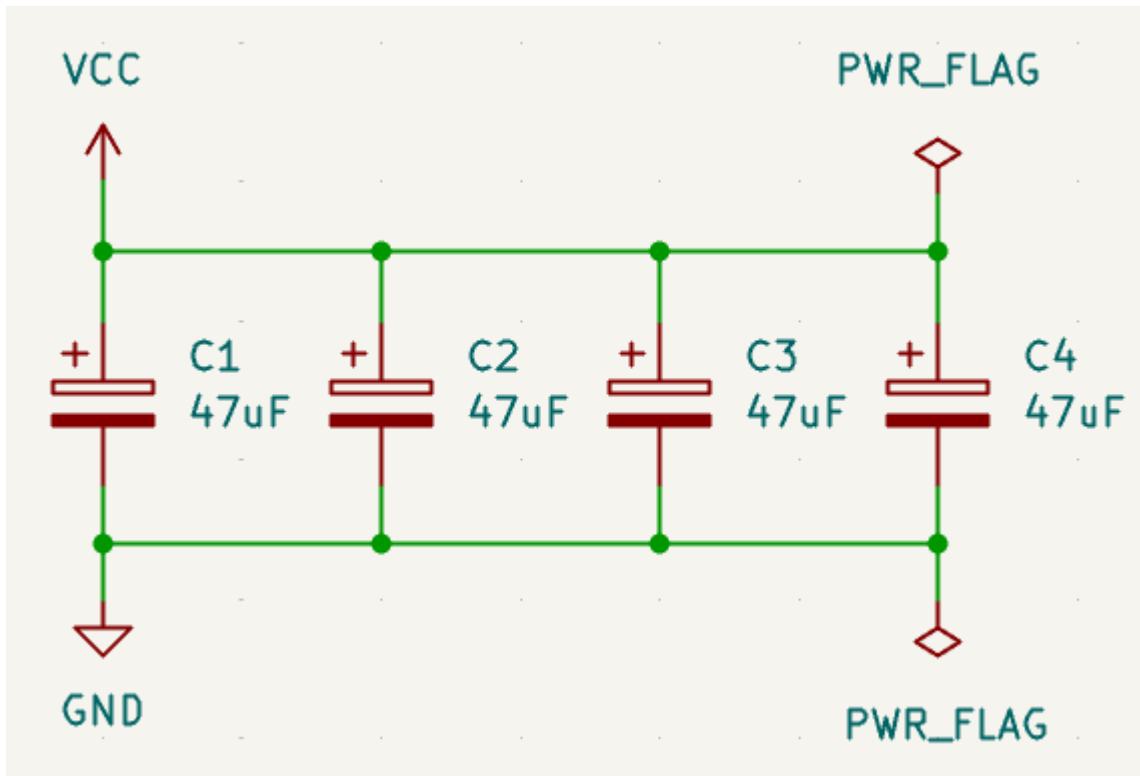
For each set of bus wires that has more than one label, you must choose the label to keep. The drop-down name box lets you choose between the labels that exist in the design, or you can choose a different name by manually entering it into the new name field.

## Power Ports

Power port symbols are conventionally used to connect pins to power nets. Power port symbols have a single pin which is invisible and marked as a power input. As described in the [hidden power pins section](#), any wire connected to the pin of a power port is therefore automatically connected to the power net with the same name as the port's pin.

In the KiCad standard library, power ports are found in the `power` library, but power port symbols can be created in any library. To create a custom power port, make a new symbol with a hidden pin marked as a power input. Name the pin according to the desired power net.

La figure ci-dessous montre un exemple de connexion de sources d'alimentation.



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

Power port symbols are found in the `power` symbol library. They can also be created by drawing a symbol with a hidden "power input" pin that has the name of the desired power net.

## PWR\_FLAG

Two `PWR_FLAG` symbols are visible in the screenshot above. They indicate to ERC that the two power nets `VCC` and `GND` are actually connected to a power source, as there is no explicit power source such as a voltage regulator output attached to either net.

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.

## No-connection flag

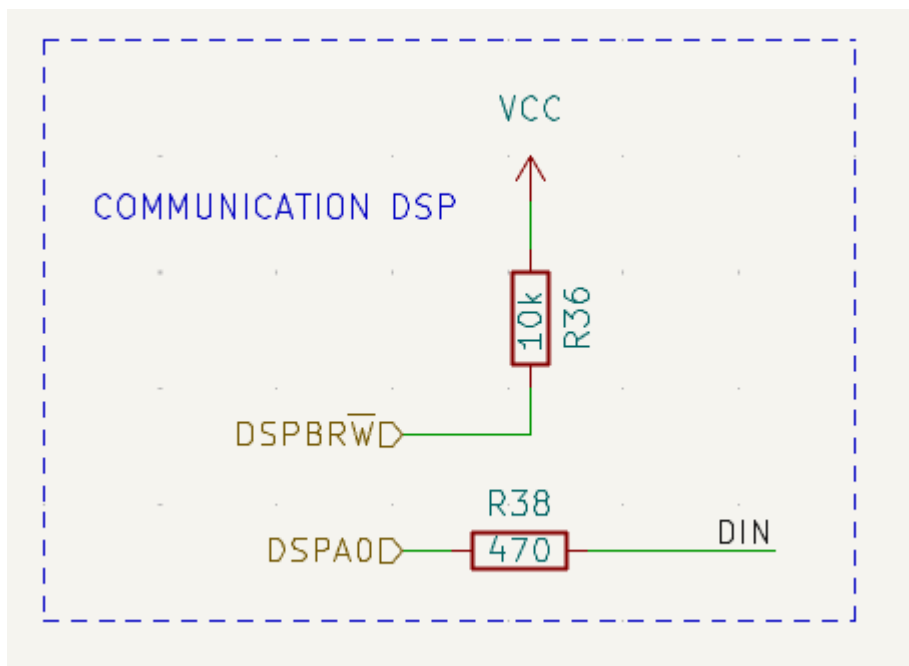
No-connection flags (→X) are used to indicate that a pin is intentionally unconnected. These flags do not have any effect on the schematic's connectivity, but they prevent "unconnected pin" ERC warnings for pins that are intentionally unconnected.

## Compléments Graphiques

### Text comments and graphic lines

It can be useful to place annotations such as text fields and frames to aid in understanding the schematic. Text fields (T) and graphic lines (→) are intended for this use, as opposed to labels and wires, which are connection elements.

The image below shows graphic lines and text in addition to wires, local labels, and hierarchical labels.



## Cartouche

The title block is edited with the Page Settings tool ()

**Ajustage opt Page**

**Page**

Taille: A3 297x420mm

Orientation: Paysage

Taille Utilisateur:

Hauteur: 279,40 Largeur: 431,80

Prévisualisation de la Page

**Paramètres du Cartouche**

Nombre de feuilles: 1 Numéro feuille: 1

Date de Publication: 22 Mars 2015 <<< 30/12/2015

Révision: 28

Titre: UNIVERSAL INTERFACE

Société: KICAD

Commentaire1: Commentaire 1

Commentaire2: Commentaire 2

Commentaire3: Commentaire 3

Commentaire4: Commentaire 4

Fichier de Description de Page

Exporter vers autres feuilles

Annuler Valider Examiner

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.



A drawing sheet template file can also be selected.

Comment 4		
Comment 3		
Comment 2		
Comment 1		
<b>KICAD</b>		
Sheet: /		
File: title_block_example.kicad_sch		
<b>Title: EXAMPLE</b>		
Size: A4	Date: 2022-02-22	<b>Rev: 1</b>
KiCad E.D.A. kicad (6.0.1)		Id: 1/1
4	5	6

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

## Rescuing cached symbols

By default, KiCad loads symbols from the project libraries according to the set paths and library order. This can cause a problem when loading a very old project: if the symbols in the library have changed or have been removed or the library no longer exists since they were used in the project, the ones in the project would be automatically replaced with the new versions. The new versions might not line up correctly or might be oriented differently leading to a broken schematic.

When a project is saved, a cache library with the contents of the current library symbols is saved along with the schematic. This allows the project to be distributed without the full libraries. If you load a project where symbols are present both in its cache and in the system libraries, KiCad will scan the libraries for conflicts. Any conflicts found will be listed in the following dialog:

This project uses symbols that no longer match the ones in the system libraries. Using this tool, you can rescue these cached symbols into a new library.

Choose "Rescue" for any parts you would like to save from this project's cache, or press "Cancel" to allow the symbols to be updated to the new versions.

All rescued components will be renamed with a new suffix of "-RESCUE-kicad\_test" to avoid naming conflicts.

**Symbols with cache/library conflicts:**

scue symbol	Symbol name
<input checked="" type="checkbox"/>	DIODE

**Instances of this symbol:**

Reference	Value
D1	DIODE
D2	DIODE
D3	DIODE


**Cached Part:**



**Library Part:**



Never Show Again

 Cancel

 OK

You can see in this example that the project originally used a diode with the cathode facing up, but the library now contains one with the cathode facing down. This change would break the schematic! Pressing OK here will cause the symbol cache library to be saved into a special ``rescue" library and all the symbols are renamed to avoid naming conflicts.

If you press Cancel, no rescues will be made, so KiCad will load all the new components by default. If you save the schematic at this point, your cache will be overwritten and the old symbols will not be recoverable. If you have saved the schematic, you can still go back and run the rescue function again by selecting "Rescue Cached Components" in the "Tools" menu to call up the rescue dialog again.

If you would prefer not to see this dialog, you can press "Never Show Again". The default will be to do nothing and allow the new components to be loaded. This option can be changed back in the Libraries preferences.


# Schématiques hiérarchiques

## Introduction

Une représentation hiérarchique est généralement une bonne solution pour des projets dépassant quelques feuilles. Si vous voulez gérer ce type de projet, il vous faudra :

- Utiliser de grande feuilles, ce qui pourrait conduire à des problèmes d'impression ou de manipulation.
- Utiliser plusieurs feuilles, ce qui vous amène à une structure hiérarchique.

La schématique complète consiste alors en une feuille principale, appelée feuille racine, et des sous-feuilles constituant la hiérarchie. En outre, une habile subdivision du schéma en plusieurs feuilles augmentera souvent sa lisibilité.

From the root sheet, you must be able to find all sub-sheets. Hierarchical schematics management is very easy with KiCad, thanks to an integrated "hierarchy navigator" accessible via the icon  of the top toolbar.

There are two types of hierarchy that can exist simultaneously: the first one has just been evoked and is of general use. The second consists in creating symbols in the library that appear like traditional symbols in the schematic, but which actually correspond to a schematic which describes their internal structure.

Le second type est utilisé pour concevoir des circuits intégrés, car dans ce cas vous devez utiliser des bibliothèques de fonctions dans le schéma que vous êtes en train de dessiner.

KiCad currently doesn't treat this second case.

Une hiérarchie peut être :

- simple : une feuille donnée n'est utilisée qu'une seule fois.
- complexe : une feuille donnée sera utilisée plusieurs fois (instances multiples).
- à plat : c'est un hiérarchie simple, mais les liaisons entre feuilles ne sont pas dessinées.


KiCad can deal with all these hierarchies.

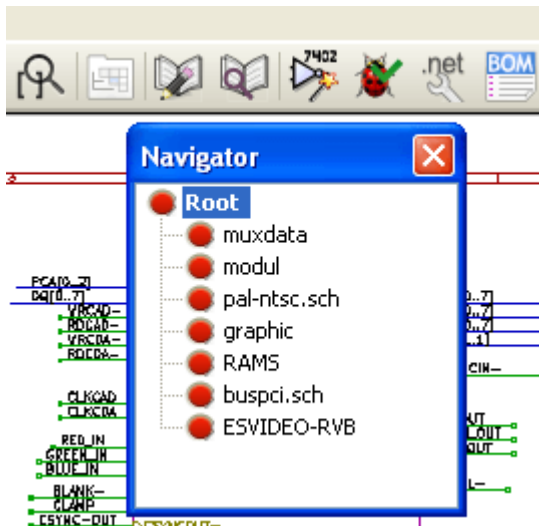
La création d'une schématique hiérarchique est facile, la hiérarchie étant manipulée à partir de la feuille racine, comme si vous n'aviez qu'un seul schéma.

Les deux étapes importantes à comprendre sont :

- Comment créer une sous-feuille.
- How to build electrical connections between sub-sheets.

## Navigation dans la hiérarchie

Navigation among sub-sheets is achieved by using the navigator tool accessible via the button  on the top toolbar.






Each sheet is reachable by clicking on its name. For quick access, right click on a sheet name, and choose to Enter Sheet or double click within the bounds of the sheet.

In order to exit the current sheet to the parent sheet, right click anywhere in the schematic where there is no object and select "Leave Sheet" in the context menu or press Alt+Backspace.

## Labels locaux, hiérarchiques et globaux

### Propriétés

Local labels, tool , are connecting signals only within a sheet. Hierarchical labels (tool ) are connecting signals only within a sheet and to a hierarchical pin placed in the parent sheet.

Global labels (tool ) are connecting signals across all the hierarchy. Power pins (type *power in* and *power out*) invisible are like global labels because they are seen as connected between them across all the hierarchy.

#### NOTE

À l'intérieur d'une hiérarchie, on peut utiliser à la fois des labels globaux ou hiérarchiques.

## Summary of hierarchy creation


Vous devez :

- Placer dans la feuille racine un symbole appelé "Feuille hiérarchique".
- Accéder à cette nouvelle feuille schématique (sous-feuille) par le navigateur, et la dessiner, comme n'importe quel schéma.
- Draw the electric connections between the two schematics by placing Global Labels (HLabels) in the new schematic (sub-sheet), and labels having the same name in the root sheet, known as SheetLabels. These SheetLabels will be connected to the sheet symbol of the root sheet to the other elements of the schematic like standard symbol pins.

## Symbole de feuille hiérarchique

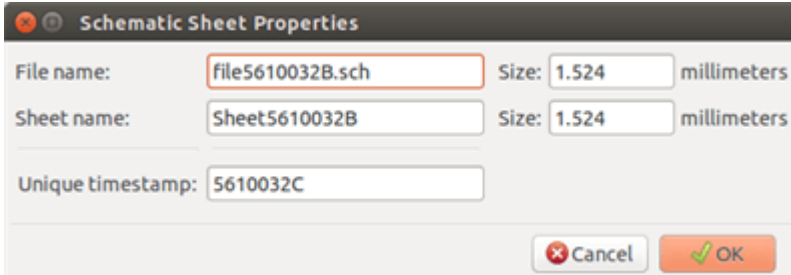
Tracez un rectangle symbolisant la sous-feuille, en plaçant deux points sur une diagonale.

La taille de ce rectangle vous permettra d'ajouter plus tard des labels particuliers, des pins de hiérarchie, correspondant aux labels globaux (Hlabels) de la sous-feuille.

These labels are similar to usual symbol pins. Select the tool .

Cliquez pour placer le coin supérieur gauche du rectangle. Cliquez à nouveau pour positionner le coin inférieur droit, afin d'avoir un rectangle suffisamment grand.

On vous demandera alors de donner un nom de fichier et un nom de feuille pour cette sous-feuille, pour vous permettre de l'atteindre par le navigateur de hiérarchie.




Vous devez au moins spécifier un nom de fichier. En l'absence de nom de feuille, c'est le nom de fichier qui sera utilisé comme nom de feuille (c'est la méthode habituelle).

## Connexions - Pins hiérarchiques

Vous allez maintenant créer des points de connexion (pins hiérarchiques) pour le symbole de feuille qui vient d'être créé.


These points of connection are similar to normal symbol pins, with however the possibility to connect a complete bus with only one point of connection.

## Importing Hierarchical Sheet Pins

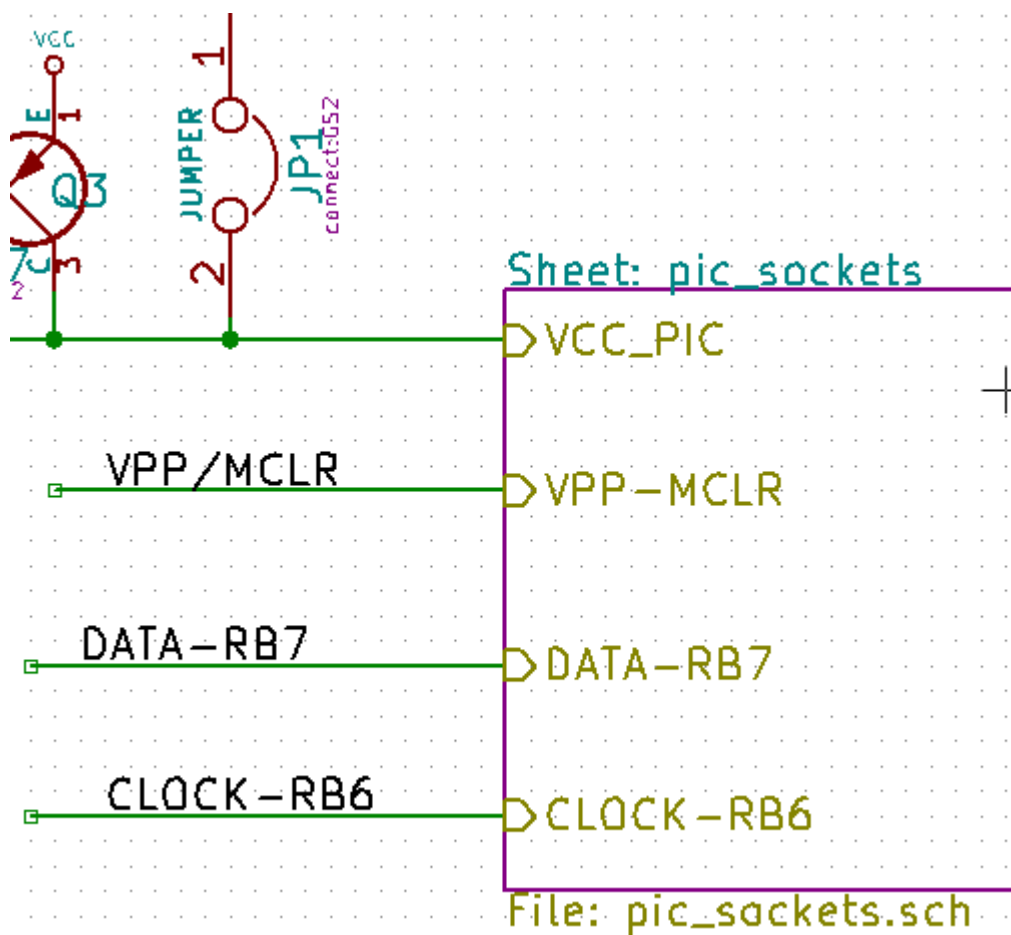
- Select the tool .
- Click on the hierarchical sheet from where you want to import the pins corresponding to hierarchical labels placed in the corresponding schematic. A hierarchical pin appears, if a new hierarchical label exists, i.e. not corresponding to an already placed pin.
- Cliquez où vous souhaitez placer la pin.

All necessary pins can thus be placed quickly and without error. Their aspect is in accordance with corresponding hierarchical labels.

## Connexions - Labels hiérarchiques

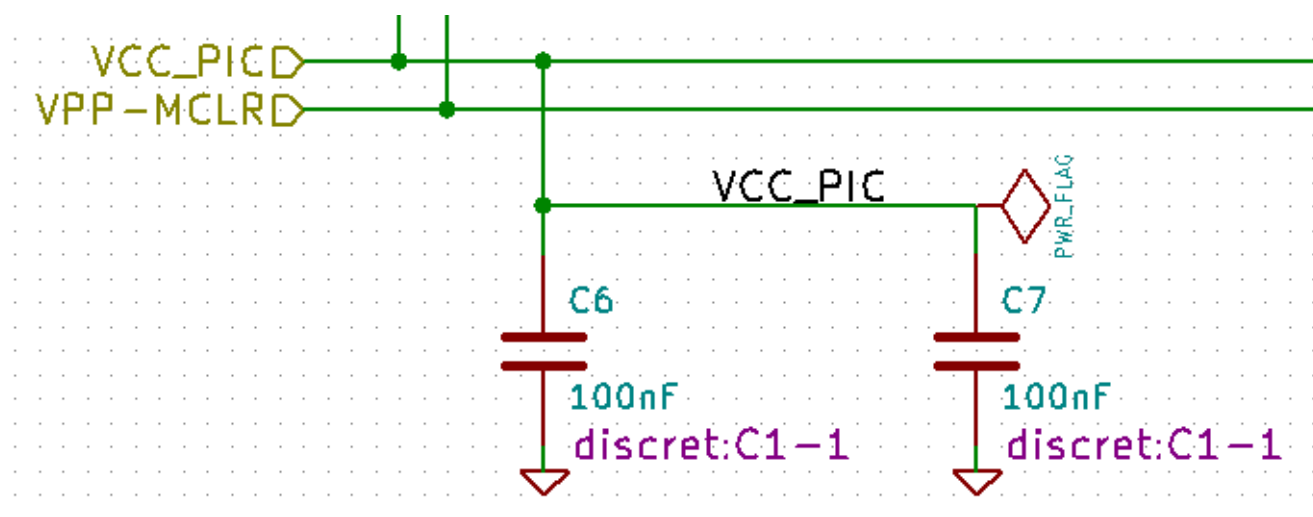
Each pin of the sheet symbol just created, must correspond to a label called hierarchical Label in the sub-sheet. Hierarchical labels are similar to labels, but they provide connections between sub-sheet and root sheet. The graphical representation of the two complementary labels (pin and hierarchical labels) is similar. Hierarchical labels are made with the tool .

Ci-dessous un exemple de feuille racine :



Remarquez la pin hiérarchique VCC-PIC, reliée au connecteur JP1.

Voici les connexions correspondantes dans la sous-feuille :



Nous retrouvons les deux labels hiérarchiques correspondants, qui établissent la connexion entre les deux feuilles hiérarchiques.

#### NOTE

Vous pouvez utiliser des pins et des labels hiérarchiques pour relier deux bus, en utilisant la syntaxe décrite précédemment (Bus [N..m]).

## Labels, labels hiérarchiques, labels globaux et pins d'alimentation invisibles

Quelques remarques sur les différentes façons d'établir des connexions autrement qu'avec des fils.

### Labels simples

Les labels simples n'ont qu'une portée locale de connexion, limitée à la feuille de schéma dans laquelle ils sont placés. Ceci est du au fait que :

- Chaque feuille a un numéro de feuille.
- Ce numéro de feuille est associé à l'étiquette.

Ainsi, quand vous placez un label "TOTO" dans la feuille n°3, le vrai nom de ce label est "TOTO\_3". Si vous avez aussi un label "TOTO" dans la feuille n°1 (feuille racine), c'est en fait un label "TOTO\_1" différent de "TOTO\_3". Ceci est toujours vrai, même si vous n'avez qu'une seule feuille.

### Labels hiérarchiques

Ce que nous avons dit pour les labels simple est vrai aussi pour les labels hiérarchiques.

Thus in the same sheet, a hierarchical label "TOTO" is considered to be connected to a local label "TOTO", but not connected to a hierarchical label or label called "TOTO" in another sheet.

A hierarchical label is considered to be connected to the corresponding sheet pin symbol in the hierarchical symbol placed in the parent sheet.

### Pins d'alimentations invisibles

It was seen that invisible power pins were connected together if they have the same name. Thus all the power pins declared "Invisible Power Pins" and named VCC are connected all symbol invisible power pins named VCC only within the sheet they are placed.

En revanche, si vous placez un label VCC dans une sous-feuille, il ne sera pas relié aux pins VCC, parce que ce label est en fait VCC\_n, où n est le numéro de la feuille.

If you want this label VCC to be really connected to the VCC for the entire schematic, it will have to be explicitly connected to an invisible power pin via a VCC power symbol.

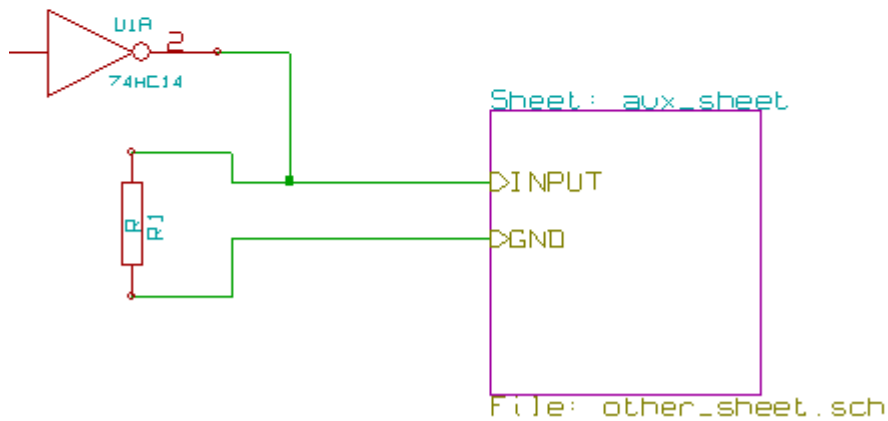
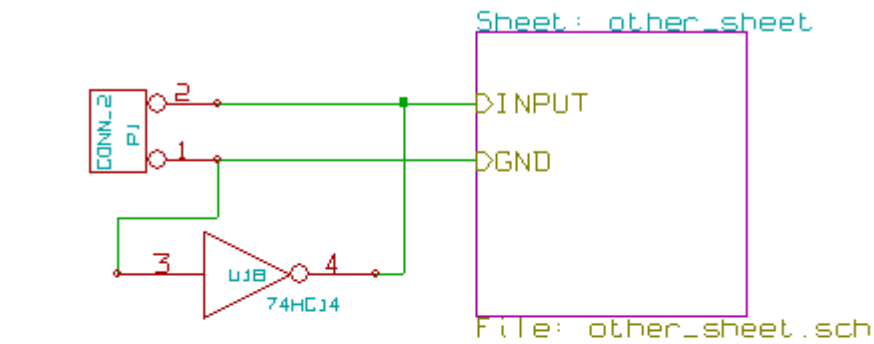
### Labels globaux

Les labels globaux qui portent le même nom sont connectés à travers toute la hiérarchie.

(les labels d'alimentation comme vcc ... sont des labels globaux)

### Hiérarchie complexe

Here is an example. The same schematic is used twice (two instances). The two sheets share the same schematic because the file name is the same for the two sheets (``other\_sheet.sch"). The sheet names must be unique.

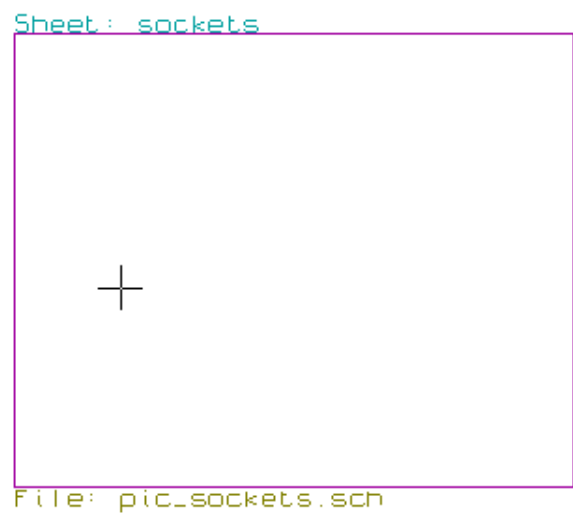


## Hiérarchie à plat

You can create a project using many sheets without creating connections between these sheets (flat hierarchy) if the following rules are observed:

- Create a root sheet containing the other sheets which acts as a link between others sheets.
- Aucune connexion explicite n'est nécessaire.
- Use global labels instead of hierarchical labels in all sheets.

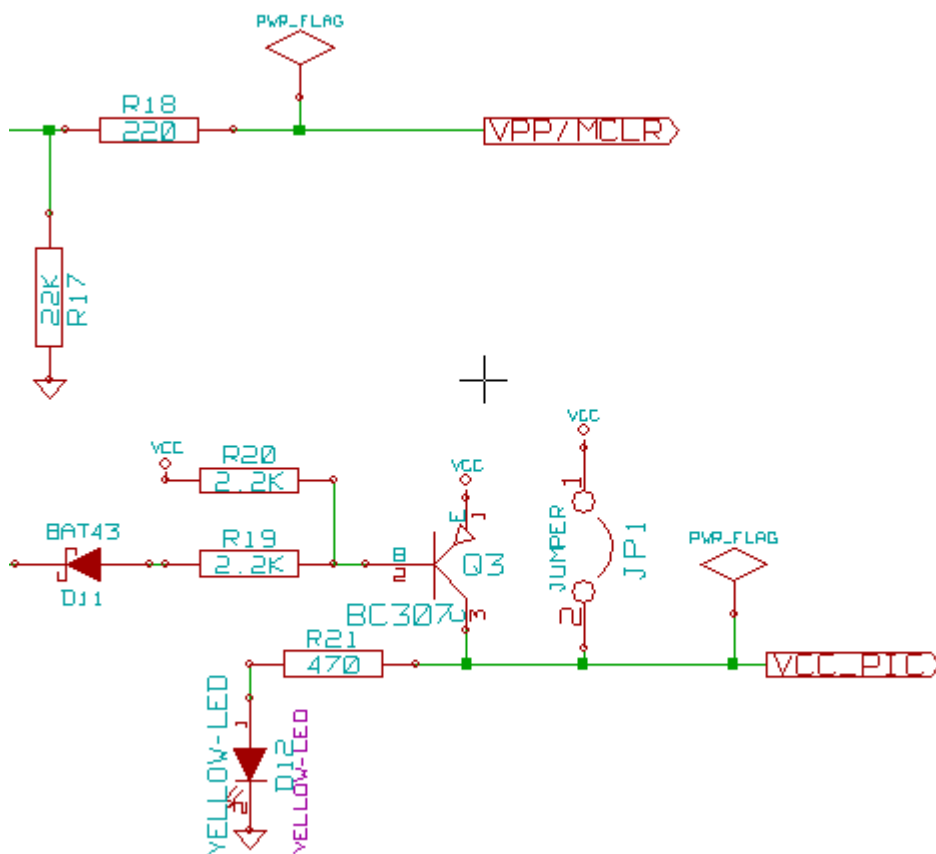
Voici un exemple de feuille racine :



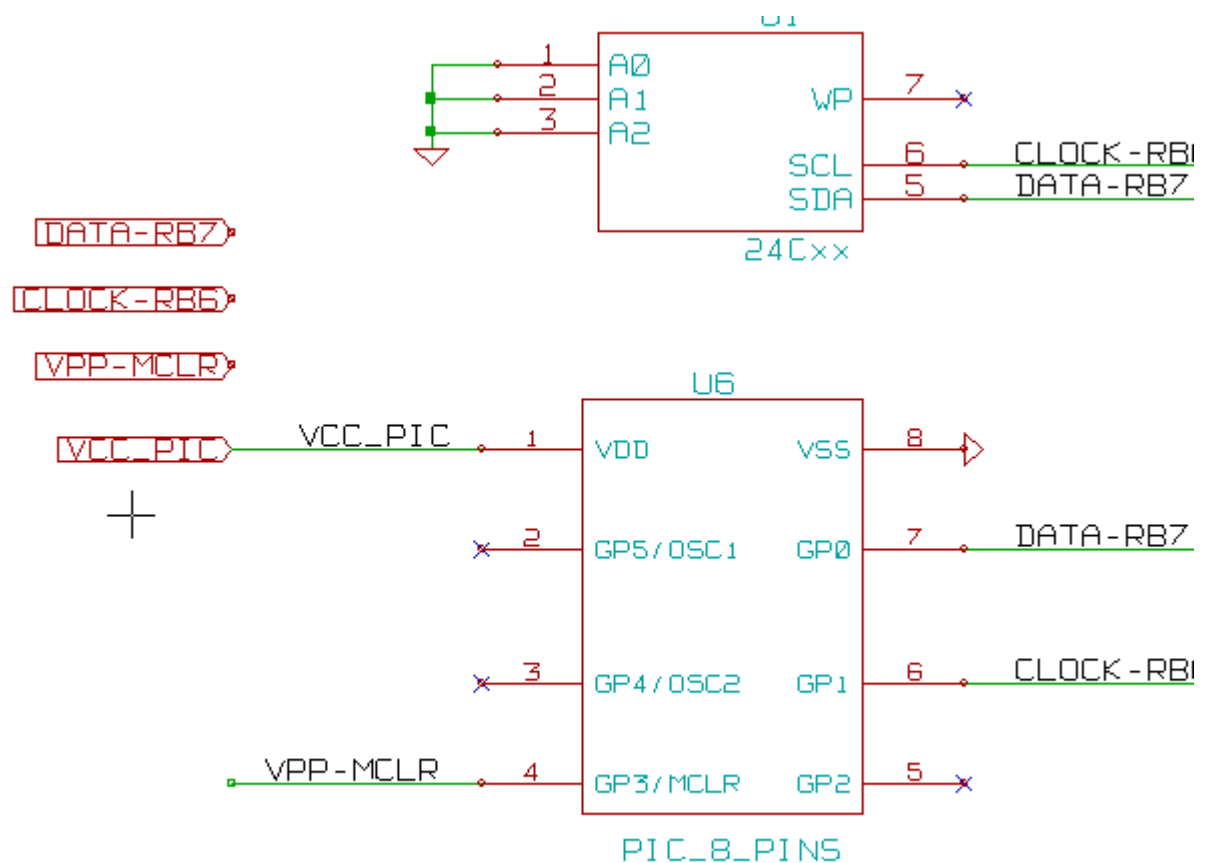


Voici les deux feuilles, connectées par des labels globaux.

Voici la feuille pic\_programmer.sch.



Voici la feuille pic\_sockets.sch.



Regardez les labels globaux.


DATA-RB7

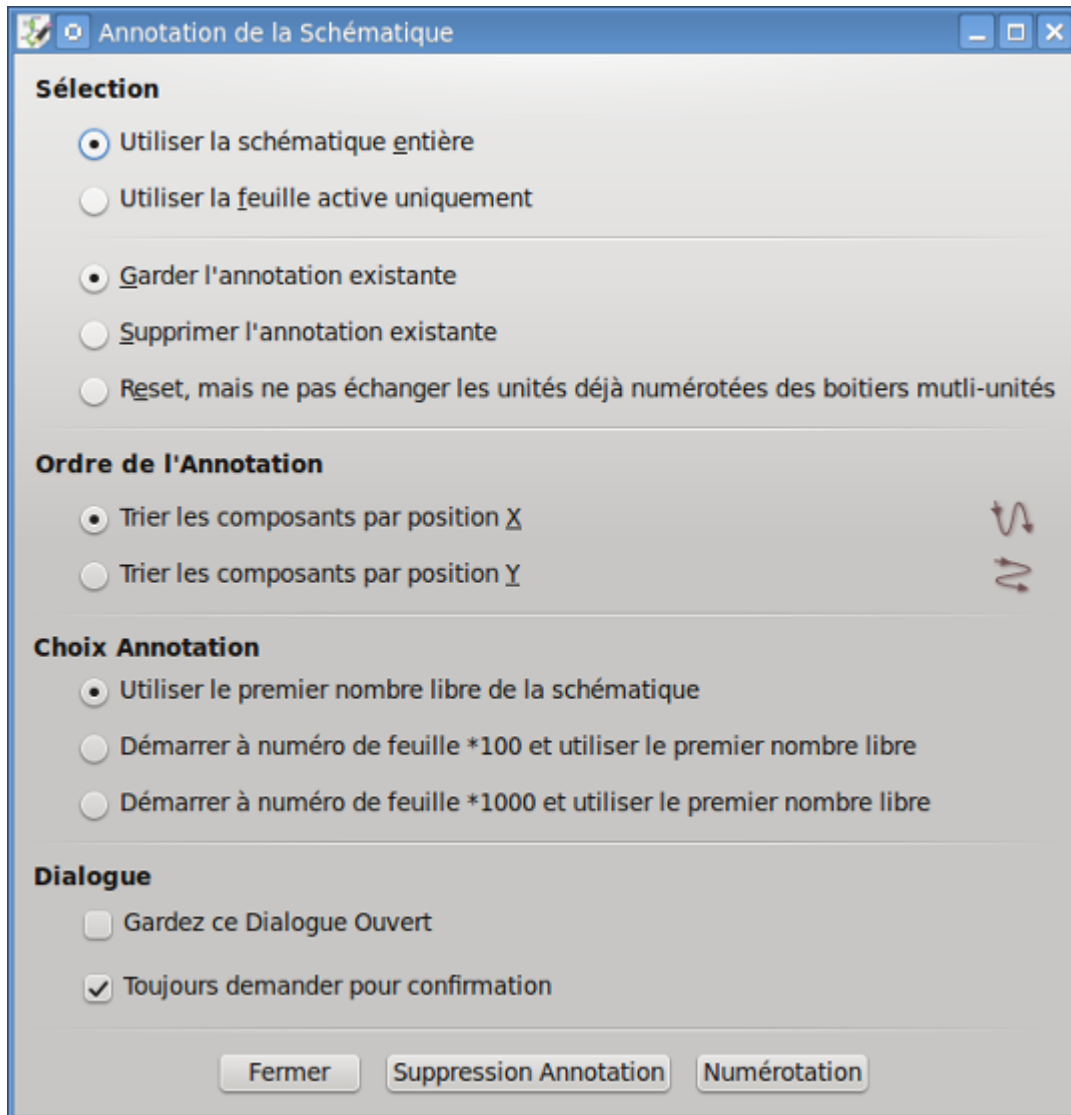
CLOCK-RB6

VPP-MCLR

# Symbol Annotation Tool

## Introduction

The annotation tool allows you to automatically assign a designator to symbols in your schematic. Annotation of symbols with multiple units will assign a unique suffix to minimize the number of these symbols. The annotation tool is accessible via the icon . Here you find its main window.



Available annotation schemes:

- Annotate all the symbols (reset existing annotation option)
- Annotate all the symbols, but do not swap any previously annotated multi-unit parts.
- Annotate only symbols that are currently not annotated. Symbols that are not annotated will have a designator which ends with a '?' character.
- Annoter toute la hiérarchie (Utiliser la schématique entière).
- Annoter seulement le schéma en cours (Utiliser la feuille active uniquement).

The ``Reset, but do not swap any annotated multi-unit parts" option keeps all existing associations between symbols with multiple units. For example, U2A and U2B may be reannotated to U1A and U1B respectively but they will never be reannotated to U1A and U2A, nor to U2B and U2A. This is useful if you want to ensure that pin groupings are maintained.

Le choix de l'ordre de l'annotation fixe la méthode utilisée pour affecter les numéros de référence sur chaque feuille de la hiérarchie.

Sauf exception, l'annotation automatique s'applique au projet entier (toutes les feuilles) et aux nouveaux composants, si on ne veut pas modifier les annotations précédentes.

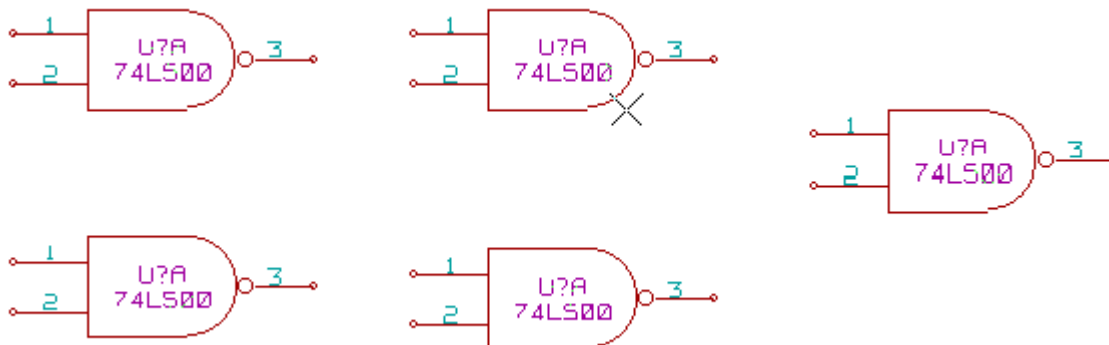
The Annotation Choice gives the method used to calculate reference:

- Use first free number in schematic: components are annotated from 1 (for each reference prefix). If a previous annotation exists, only unused numbers will be used.
- Démarrer à numéro de feuille \*100 et utiliser le premier nombre libre : l'annotation commence par 101 sur la feuille numéro 1, par 201 sur la feuille numéro 2, etc... S'il y a plus de 99 éléments avec le même préfixe de référence (U, R) sur la feuille 1, l'outil d'annotation utilisera le numéro 200 et suivants, et l'annotation de la feuille 2 commencera au prochain numéro libre.
- Démarrer à numéro de feuille \*1000 et utiliser le premier nombre libre : l'annotation commence par 1001 sur la feuille numéro 1, par 2001 sur la feuille numéro 2, etc...

## Quelques exemples

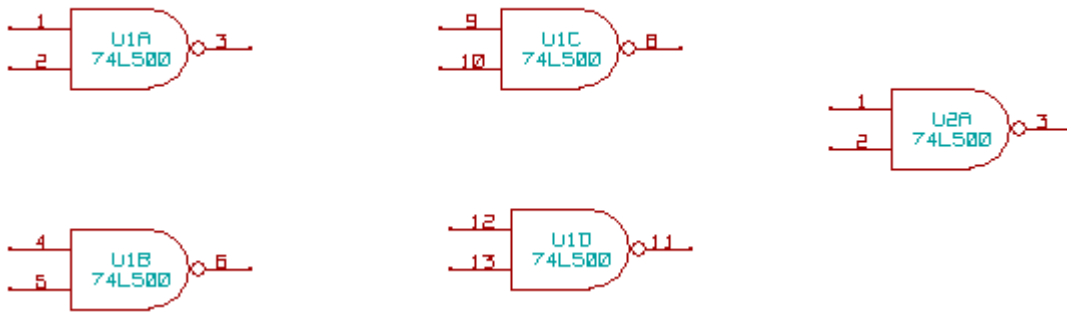
### Ordre d'annotation

Cet exemple montre 5 composants, non encore annotés.

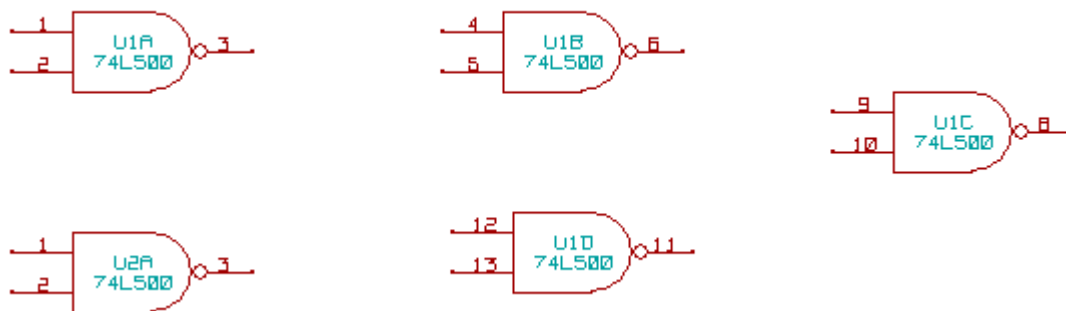


Après l'exécution de l'annotation automatique, on obtient le résultat suivant.

Composants triés par position X.



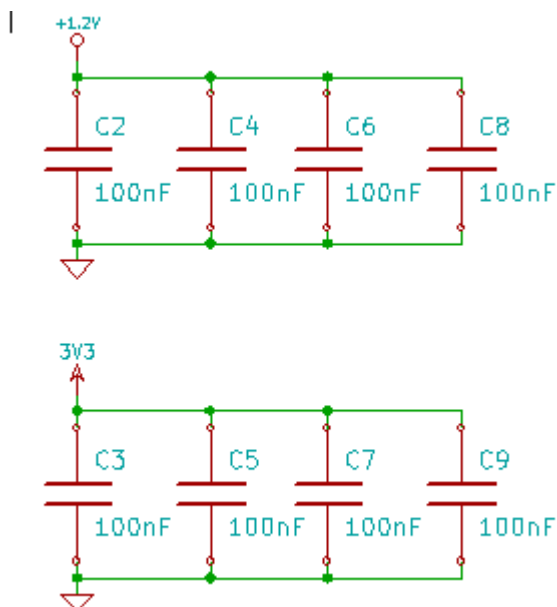
Composants triés par position Y.



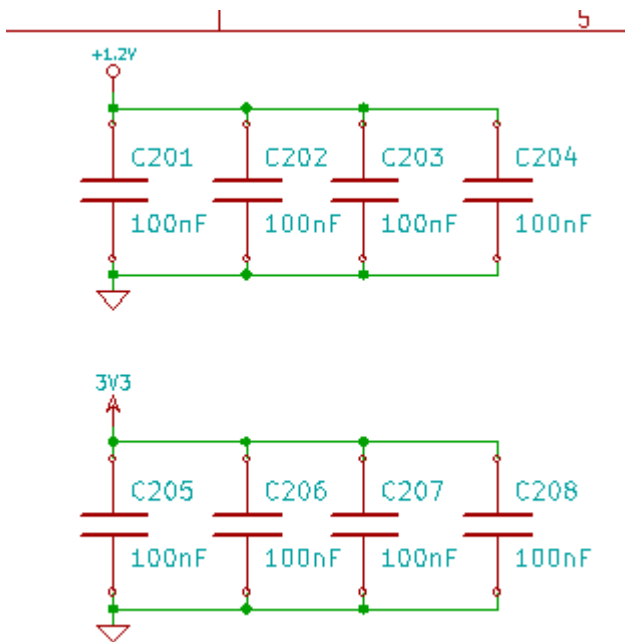
Vous pouvez voir que quatre portes 74LS00 ont été réparties dans le boîtier U1, et que la cinquième porte 74LS00 a été assignée au suivant, U2.

## Choix de l'annotation

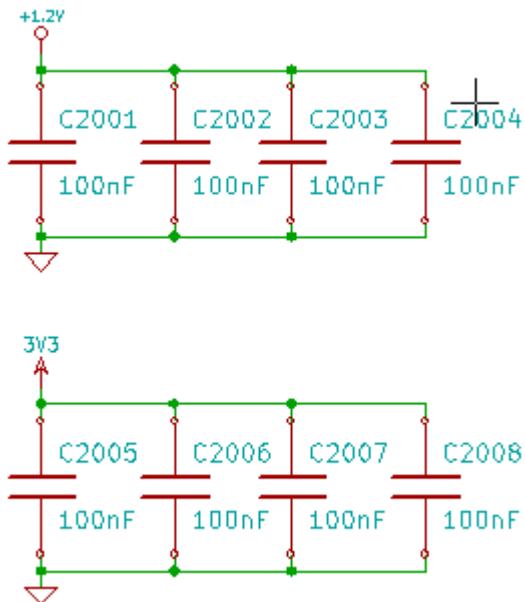
Voici une annotation de la feuille 2 avec l'option 'Utiliser le premier nombre libre de la schématique'.



L'option 'Démarrer à numéro de feuille \*100 et utiliser le premier nombre libre' donne le résultat suivant.



L'option 'Démarrer à numéro de feuille \*1000 et utiliser le premier nombre libre' donne le résultat suivant.

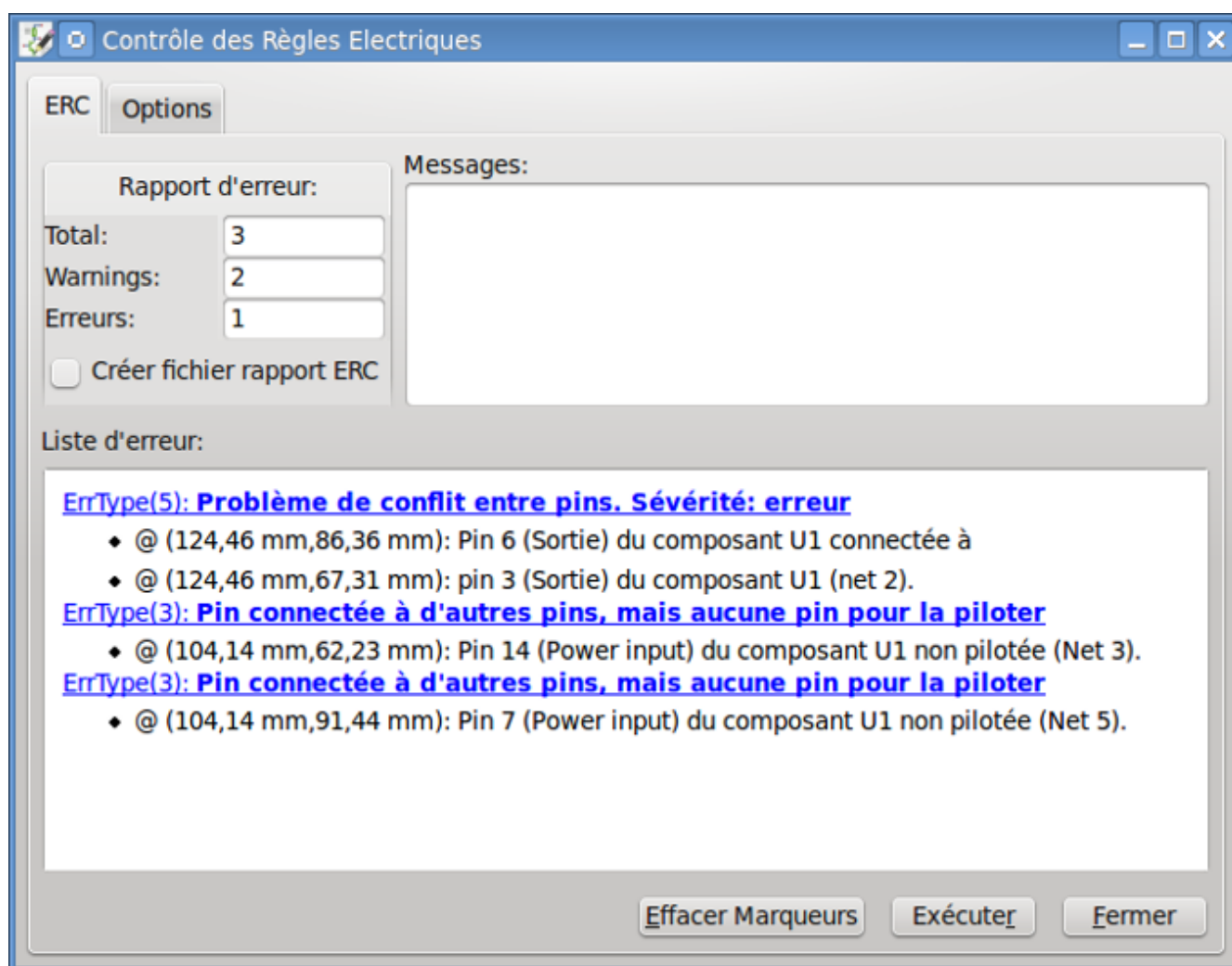


# Vérification des règles électriques (ERC)

## Introduction

L'outil de vérification des règles électriques, ou ERC (Electrical Rules Check), vérifie automatiquement votre schéma. Il détecte les erreurs dans la feuille, comme les pins ou les symboles hiérarchiques non connectés, les sorties en court-circuit, etc... Bien entendu une vérification automatique n'est pas infaillible, et le logiciel qui la réalise n'est pas encore terminé à 100%. Malgré tout, cette vérification est très utile, car elle détecte beaucoup d'omissions et de petites erreurs.

In fact all detected errors must be checked and then corrected before proceeding as normal. The quality of the ERC is directly related to the care taken in declaring electrical pin properties during symbol library creation. ERC output is reported as `errors''` or `warnings''`.



## Utilisation de l'ERC

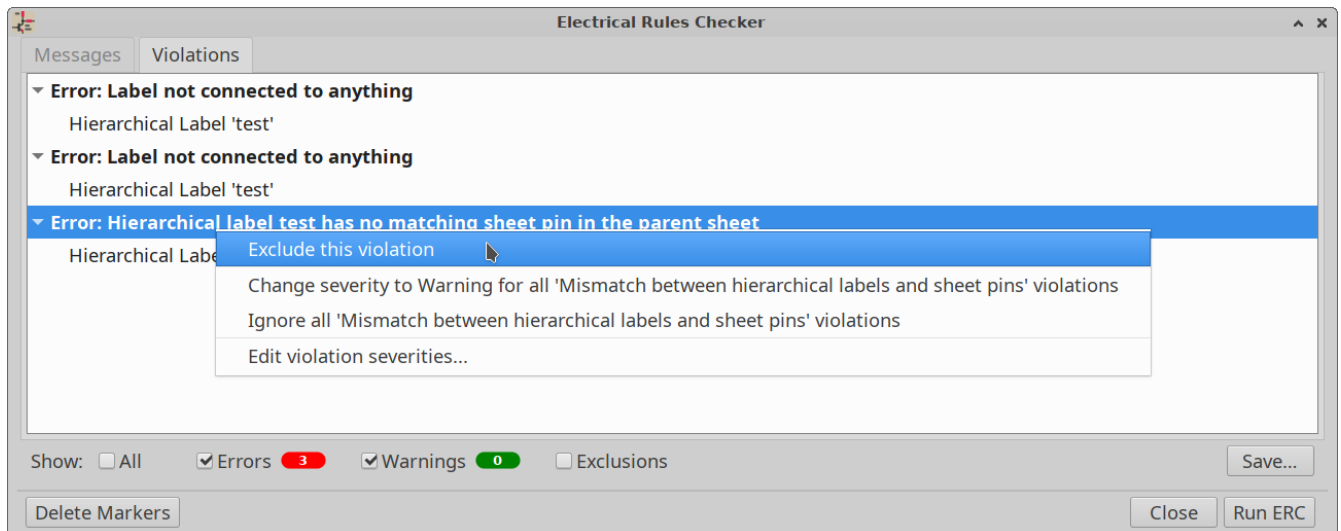
ERC can be started by clicking on the icon .

Des avertissements, sous forme de petites flèches de marquage, seront placés sur les éléments schématiques générant une erreur ERC (pins ou labels).

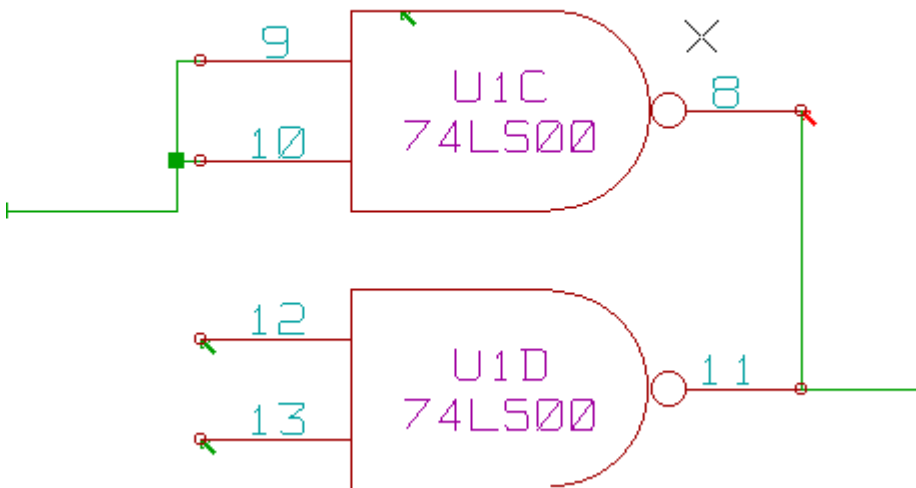
## NOTE

- Dans cette boîte de dialogue, en cliquant sur un message d'erreur, vous allez au marqueur d'erreur correspondant dans le schéma.
- Dans le schéma, faites un clic droit sur un marqueur pour accéder au message de diagnostic correspondant.

You can also delete error markers from the dialog and set specific ERC messages to be suppressed by using the right-click context menu.



## Exemple d'ERC



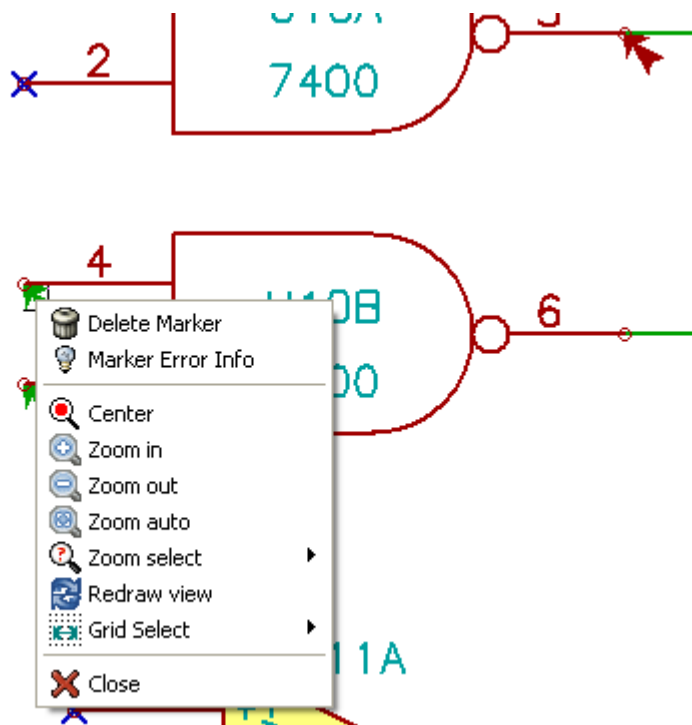
Ici, vous pouvez voir quatre erreurs :

- Deux sorties logiques ont été reliées ensemble (flèche rouge).
- Deux entrées ne sont pas connectées (flèches vertes du bas).
- Une erreur sur une source d'alimentation invisible, dont il manque le symbole d'alimentation (flèche verte du haut).

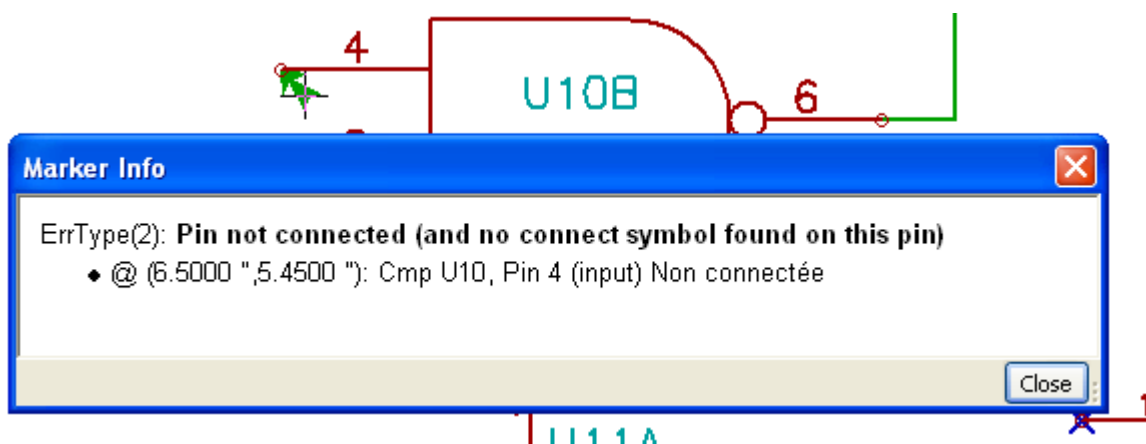


## Affichage du diagnostic

Un clic droit sur un marqueur vous affiche le menu contextuel permettant d'accéder à la fenêtre d'informations de diagnostic de l'ERC.



et en cliquant sur un marqueur, vous obtenez une description de l'erreur.

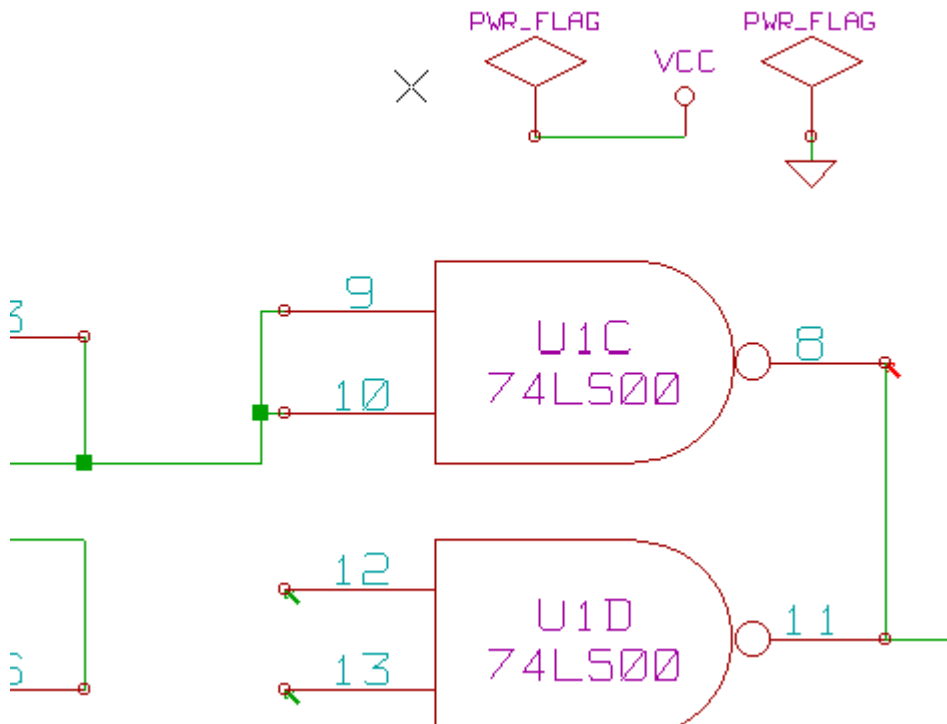


## Pins d'alimentation et symboles d'alimentation (Power Flag)

Il est fréquent d'avoir une erreur ou un avertissement sur les pins d'alimentation, même si tout semble normal. Voir l'exemple ci-dessus. Cela arrive parce que, dans la plupart des dessins, l'alimentation est fournie par des connecteurs qui ne sont pas identifiés comme des sources d'énergie (au contraire d'une sortie de régulateur qui, elle, est déclarée en tant que sortie d'alimentation).

Ainsi l'ERC ne détectera pas une pin de sortie d'alimentation pour ce fil et le déclarera non-connecté à une source d'alimentation.

Pour éviter ceci, il faut placer un symbole d'alimentation, "PWR\_FLAG", sur ce connecteur d'alim. Comme dans l'exemple suivant :

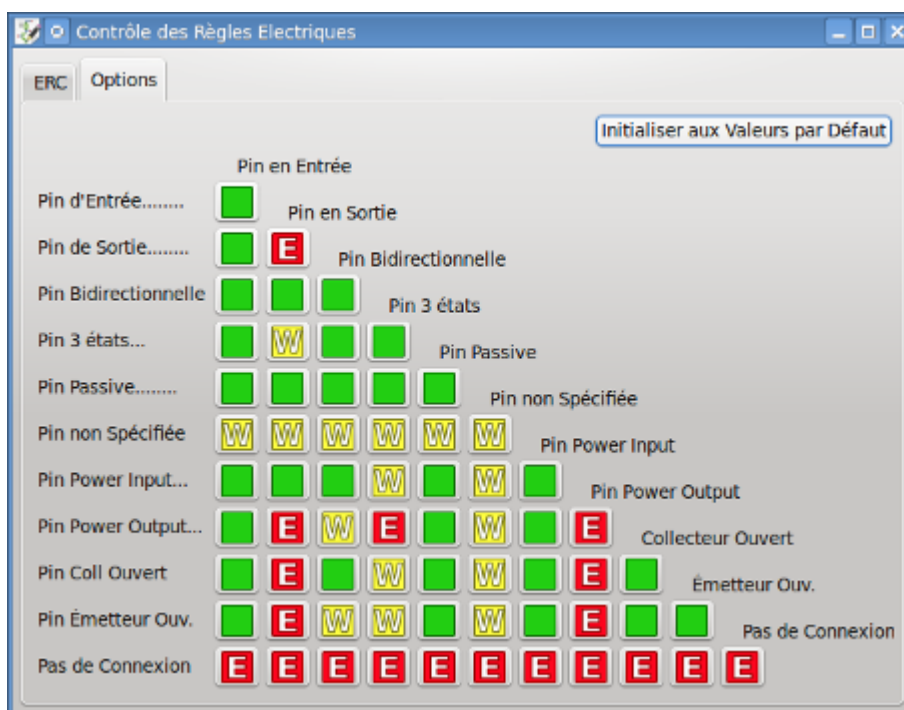


Et ainsi le marqueur disparaît.

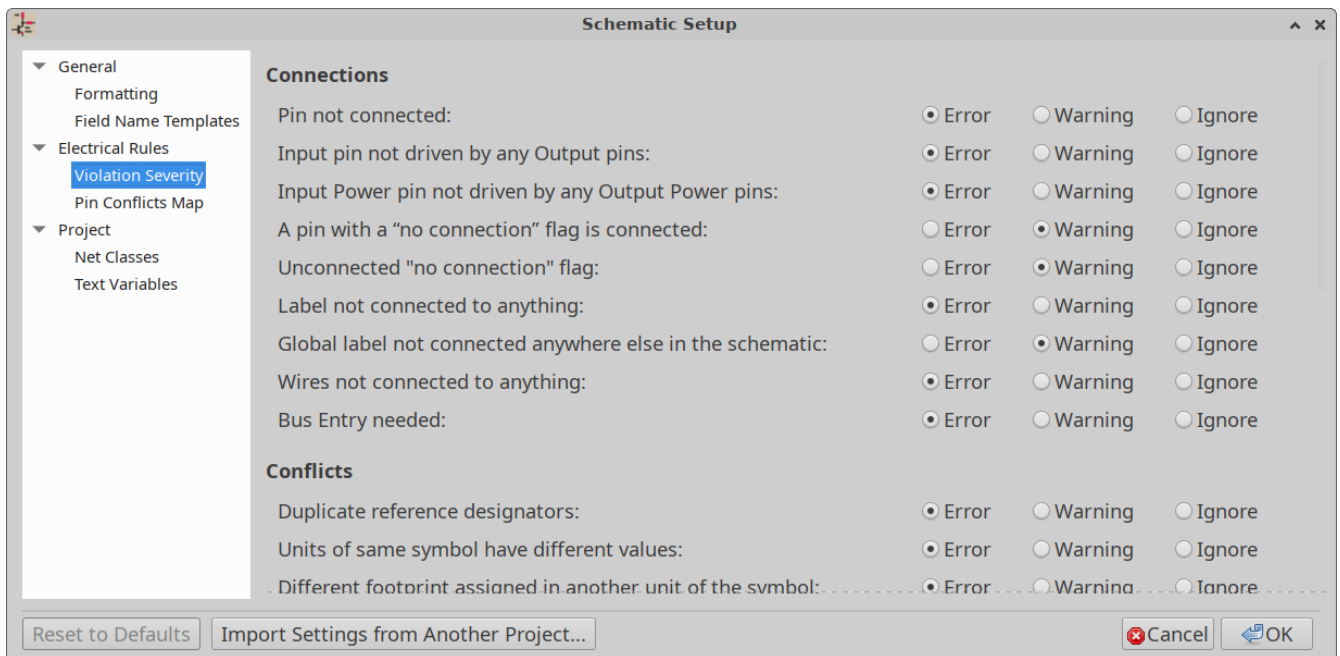
Most of the time, a PWR\_FLAG must be connected to GND, because regulators have outputs declared as power out, but ground pins are never power out (the normal attribute is power in), so grounds never appear connected to a power source without a power flag symbol.

## Configuration

The *Pin Conflicts Map* panel in Schematic Setup 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



Les règles sont modifiées en cliquant plusieurs fois sur le bouton carré dans le tableau pour faire défiler les différents choix : normal [vert], avertissement [W jaune], erreur [E rouge].



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

## Fichier de rapport d'ERC

An ERC report file can be generated and saved by checking the option Write ERC report. The file extension for ERC report files is .erc. Here is an example ERC report file.


```
ERC control (4/1/1997-14:16:4)

***** Sheet 1 (INTERFACE UNIVERSAL)
ERC: Warning Pin input Unconnected @ 8.450, 2.350
ERC: Warning passive Pin Unconnected @ 8.450, 1.950
ERC: Warning: BiDir Pin connected to power Pin (Net 6) @ 10.100, 3.300
ERC: Warning: Power Pin connected to BiDir Pin (Net 6) @ 4.950, 1.400

>> Errors ERC: 4
```

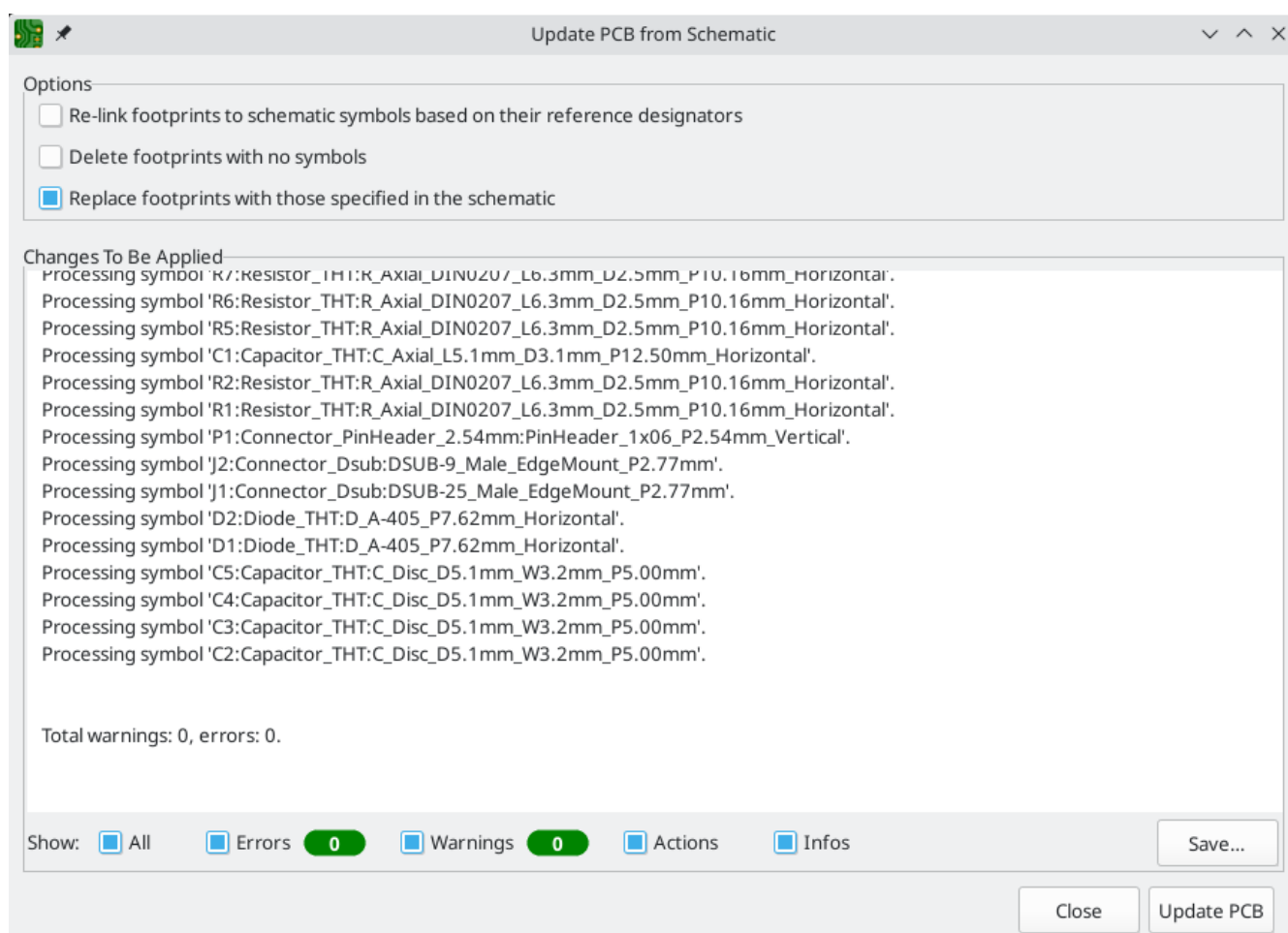
# Transfer Schematic to PCB

## Généralités

Use the Update PCB from Schematic tool to sync design information from the Schematic Editor to the Board Editor. The tool can be accessed with **Tools** → **Update PCB from Schematic** (**F8**) in both the schematic and board editors. You can also use the  icon in the top toolbar of the Board Editor.

### NOTE

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



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

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

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

## Options

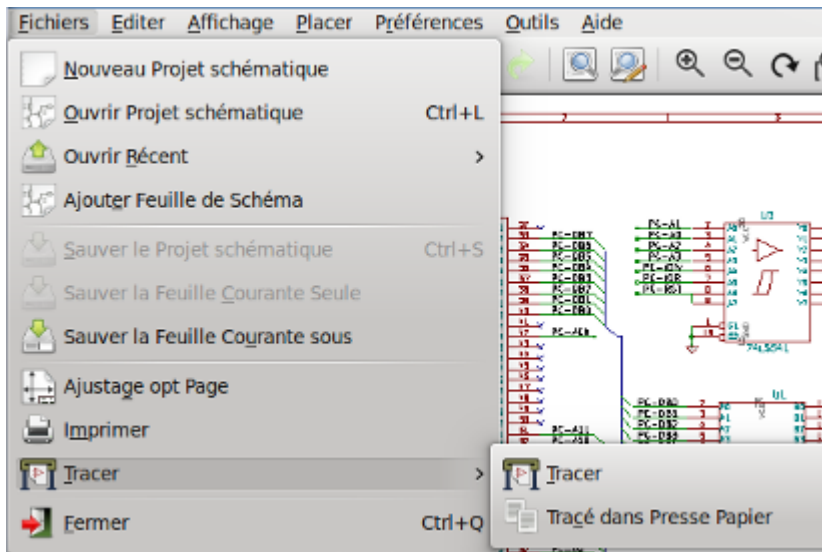
The tool has several options to control its behavior.

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

# Tracer / Imprimer

## Introduction

Les commandes 'Imprimer' et 'Tracer' sont accessibles par le menu 'Fichiers'.



Les formats de sortie peuvent être : Postscript, PDF, SVG, DXF ou HPGL. Vous pouvez aussi imprimer directement sur votre imprimante.

## Commandes de tracé communes

### Tracer Page Courante

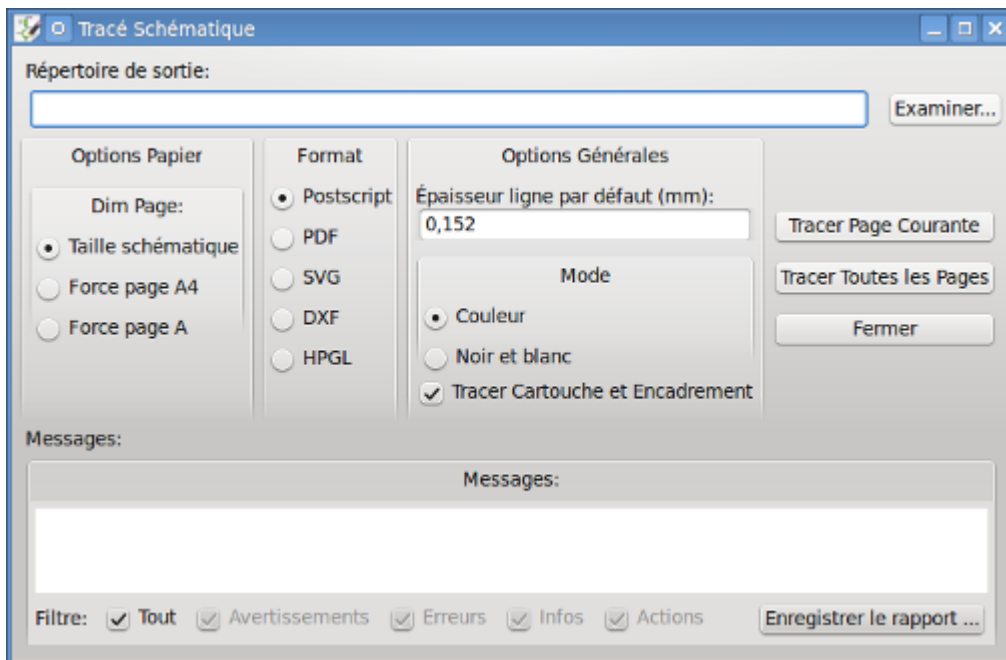
génère un fichier pour la feuille courante seulement.

### Tracer Toutes les Pages

vous permet de tracer toute la hiérarchie (un fichier est généré pour chaque feuille).

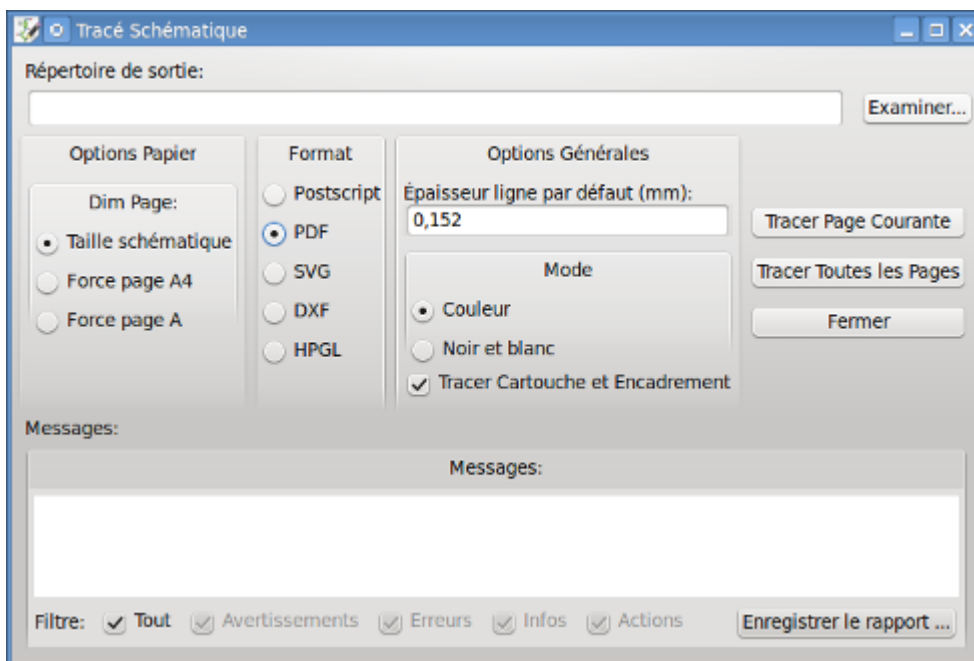
## Tracer en Postscript

Cette commande vous permet de générer des fichiers au format PostScript.



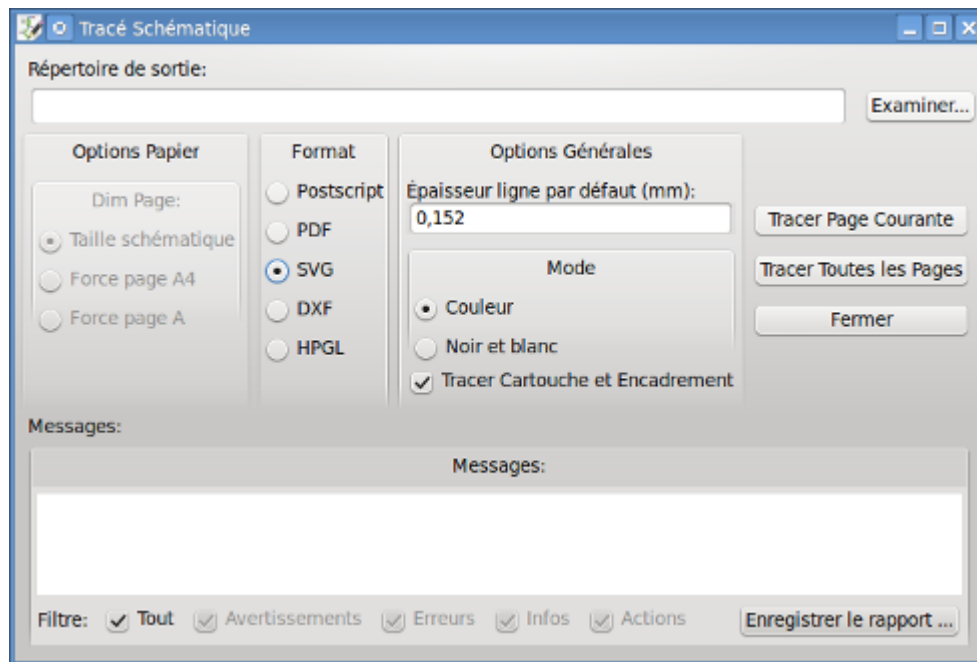
Le nom du fichier généré est le nom de la feuille avec l'extension .ps. Vous pouvez désactiver l'option "Tracer cartouche et encadrement". Ceci est utile quand vous voulez créer un fichier PostScript pour l'encapsulation (format .eps), utilisé pour insérer une figure dans un logiciel de traitement de texte. La fenêtre de message affiche le chemin et le nom des fichiers créés.

## Tracer en PDF



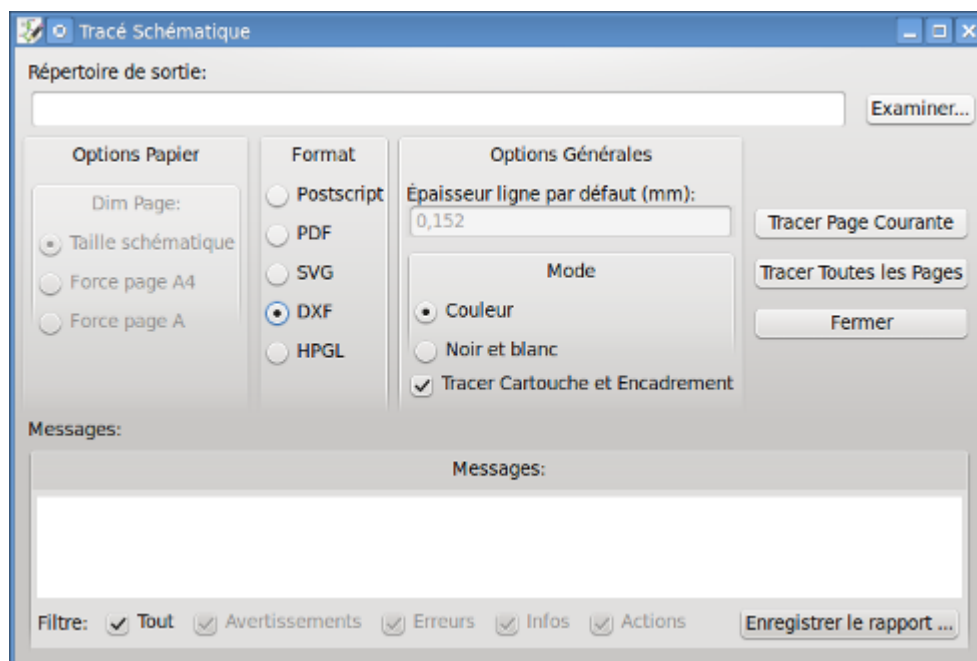
Vous permet de générer un tracé au format PDF. Le nom du fichier généré est le nom de la feuille avec l'extension .pdf.

## Tracer en SVG



Vous permet de générer un tracé au format vectoriel SVG. Le nom du fichier généré est le nom de la feuille avec l'extension .svg.

## Tracer en DXF



Vous permet de générer un tracé au format DXF. Le nom du fichier généré est le nom de la feuille avec l'extension .dxf.

## Tracer en HPGL

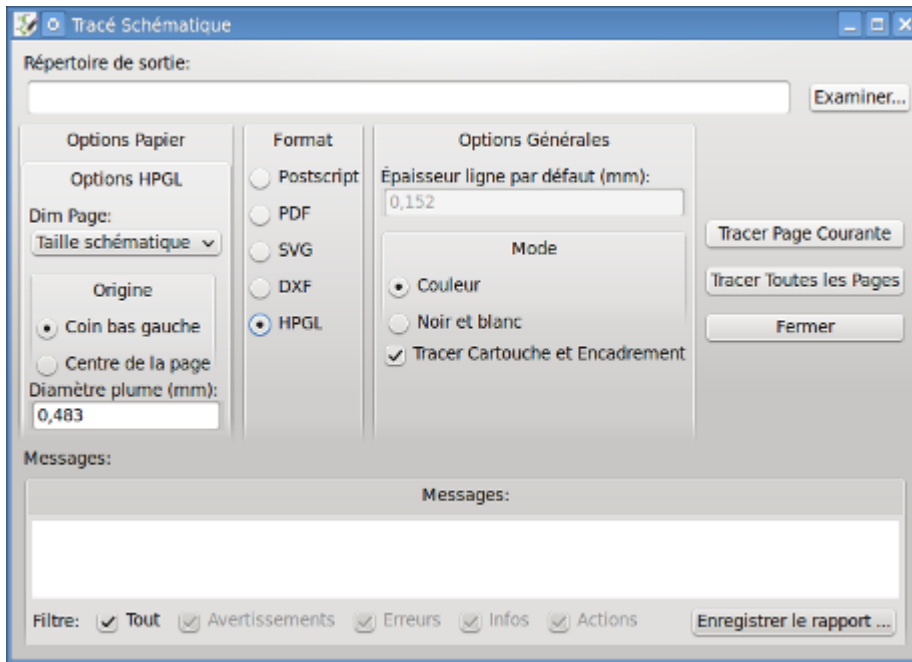
Vous permet de générer un tracé au format HPGL. Pour ce format, vous pouvez définir :

- La taille de page.
- L'origine.



- La taille du pinceau (en mm).

La fenêtre de configuration du tracé ressemble à ceci :



Le nom du fichier généré sera le nom de la feuille avec l'extension .plt.

## Sélection de la taille de la feuille schématique

La case 'Taille Shématique' est normalement cochée. Dans ce cas, la taille de la feuille définie dans les options de la page sera utilisée, et l'échelle choisie sera de 1. Si une autre taille de feuille est sélectionnée (de A4 à A0, de A à E, etc.), l'échelle sera automatiquement ajustée pour remplir la page.

## Ajustement des décalages

Pour toutes les dimensions standards, vous pouvez ajuster les décalages pour centrer le dessin aussi précisément que possible. Certains traceurs ayant un point d'origine au centre, et d'autres au coin inférieur droit, il est nécessaire de pouvoir introduire un décalage pour tracer correctement.


Généralement :

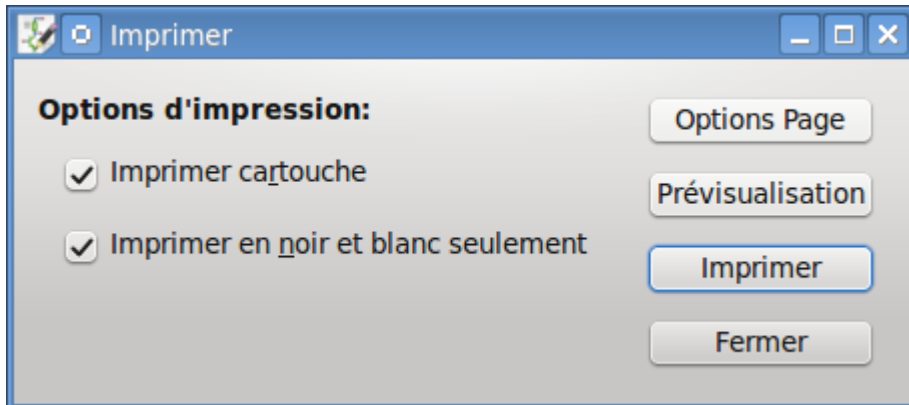
- Pour des traceurs ayant leur point d'origine au centre de la feuille, le décalage doit être négatif et fixé à la moitié de la dimension de la feuille.
- Pour des traceurs ayant leur point d'origine dans le coin inférieur gauche de la feuille, le décalage doit être réglé à 0.

Pour fixer un décalage :

- Sélectionnez la taille de la feuille.
- Fixez les décalages X et Y.
- Cliquez sur accepter les décalages.

## Imprimer sur papier

This command, available via the icon , allows you to visualize and generate design files for the standard printer.



L'option "Imprimer cartouche" active ou désactive l'impression du cartouche.

L'option "Imprimer en noir et blanc seulement" force l'impression en monochrome. Cette option est généralement nécessaire si vous avez une imprimante laser noir et blanc, parce que les couleurs, imprimées en demi-tons, ne sont souvent pas très lisibles.

# Symbol Editor

## General Information About Symbol Libraries

A symbol is a schematic element which contains a graphical representation, electrical connections, and text fields describing the symbol. Symbols used in a schematic are stored in symbol libraries. KiCad provides a symbol editing tool that allows you to create libraries, add, delete or transfer symbols between libraries, export symbols to files, and import symbols from files. The symbol editing tool provides a simple way to manage symbols and symbol libraries.

## Symbol Library Overview

A symbol library is composed of one or more symbols. Generally the symbols are logically grouped by function, type, and/or manufacturer.

A symbol is composed of:

- Graphical items (lines, circles, arcs, text, etc.) that determine how symbol looks in a schematic.
- Pins which have both graphic properties (line, clock, inverted, low level active, etc.) and electrical properties (input, output, bidirectional, etc.) used by the Electrical Rules Check (ERC) tool.
- Des champs : référence, valeur, empreintes correspondantes pour le dessin du circuit imprimé, etc...

Symbols can be derived from another symbol in the same library. Derived symbols share the base symbol's graphical shape and pin definitions, but can override the base symbol's property fields (value, footprint, footprint filters, datasheet, description, etc.). Derived symbols can be used to define symbols that are similar to a base part. For example, 74LS00, 74HC00, and 7437 symbols could all be derived from a 7400 symbol. In previous versions of KiCad, derived symbols were referred to as aliases.

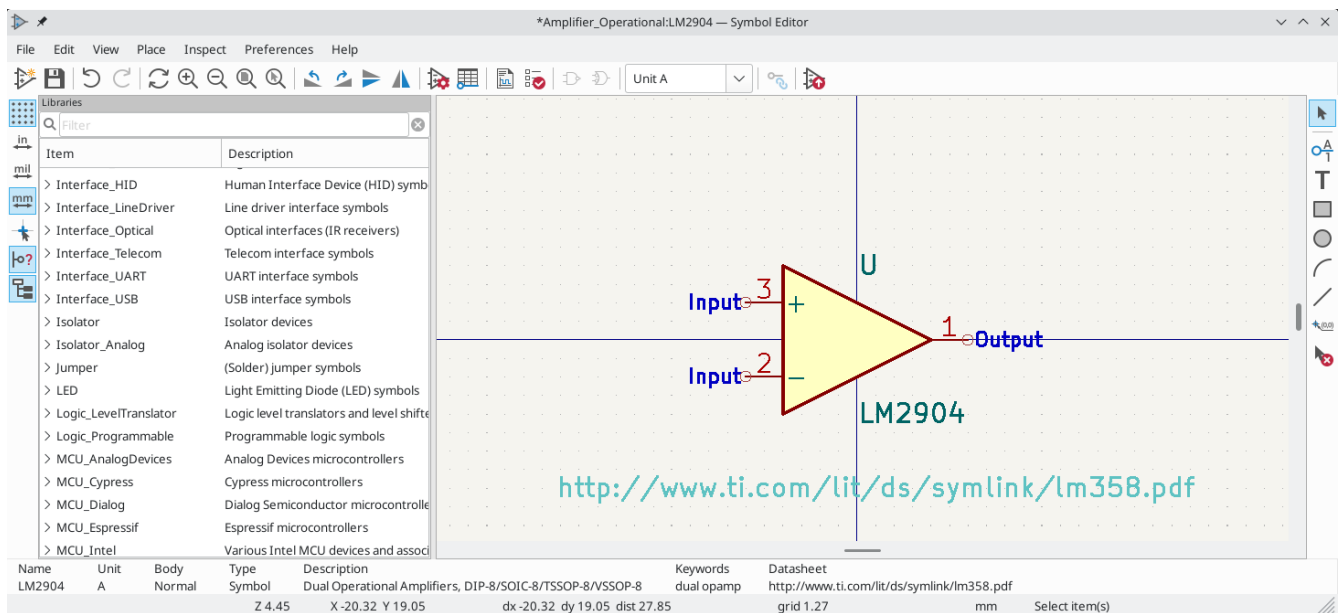
Proper symbol designing requires:

- Defining if the symbol is made up of one or more units.
- Defining if the symbol has an alternate body style (also known as a De Morgan representation).
- De dessiner sa représentation symbolique, au moyen de lignes, rectangles, cercles, polygones, et de texte.
- D'ajouter des pins, en définissant leurs éléments graphiques, leurs noms, leurs numéros et leurs propriétés électriques (entrées, sorties, trois-états, alimentations, etc.).
- Determining if the symbol should be derived from another symbol with the same graphical design and pin definition.
- D'ajouter des champs supplémentaires, comme le nom de l'empreinte utilisée par le logiciel de dessin du circuit imprimé, et de définir leur visibilité.
- Documenting the symbol by adding a description string and links to data sheets, etc.
- De le sauvegarder dans la librairie désirée.

## Symbol Library Editor Overview

The symbol library editor main window is shown below. It consists of three tool bars for quick access to common features and a symbol viewing/editing area. Not all commands are available on the tool bars but






















can be accessed using the menus.



## Barre d'outils principale










The main tool bar is located at the top of the main window. It consists of the undo/redo commands, zoom commands, symbol properties dialogs, and unit/representation management controls.



	Create a new symbol in the selected library.
	Save the currently selected library. All modified symbols in the library will be saved.
	Undo last edit.
	Redo last undo.
	Refresh display.
	Zoom in.
	Zoom out.
	Zoom to fit symbol in display.
	Zoom to fit selection.
	Rotate counter-clockwise.
	Rotate clockwise.
	Mirror horizontally.
	Mirror vertically.
	Edit the current symbol properties.
	Edit the symbol's pins in a tabular interface.
	Open the symbol's datasheet. The button will be disabled if no datasheet is defined for the current symbol.
	Test the current symbol for design errors.
	Select the normal body style. The button is disabled if the current symbol does not have an alternate body style.
	Select the alternate body style. The button is disabled if the current symbol does not have an alternate body style.
	Select the unit to display. The drop down control will be disabled if the current symbol is not derived from a symbol with multiple units.
	Enable synchronized pins edit mode. When this mode is enabled, any pin modifications are propagated to all other symbol units. Pin number changes are not propagated. This mode is automatically enabled for symbols with multiple interchangeable units and cannot be enabled for symbols with only one unit.








## Barre d'outils des éléments

The vertical toolbar located on the right hand side of the main window allows you to place all of the elements required to design a symbol.


	Select tool. Right-clicking with the select tool opens the context menu for the object under the cursor. Left-clicking with the select tool displays the attributes of the object under the cursor in the message panel at the bottom of the main window. Double-left-clicking with the select tool will open the properties dialog for the object under the cursor.
	Pin tool. Left-click to add a new pin.
	Graphical text tool. Left-click to add a new graphical text item.
	Rectangle tool. Left-click to begin drawing the first corner of a graphical rectangle. Left-click again to place the opposite corner of the rectangle.
	Circle tool. Left-click to begin drawing a new graphical circle from the center. Left-click again to define the radius of the circle.
	Arc tool. Left-click to begin drawing a new graphical arc item from the first arc end point. Left-click again to define the second arc end point. Adjust the radius by dragging the arc center point.
	Connected line tool. Left-click to begin drawing a new graphical line item in the current symbol. Left-click for each additional connected line. Double-left-click to complete the line.
	Anchor tool. Left-click to set the anchor position of the symbol.
	Delete tool. Left-click to delete an object from the current symbol.

## Barre d'outils des options

The vertical tool bar located on the left hand side of the main window allows you to set some of the editor drawing options.


	Toggle grid visibility on and off.
	Set units to inches.
	Set units to mils (0.001 inch).
	Set units to millimeters.
	Toggle full screen cursor on and off.
	Toggle display of pin electrical types.
	Toggle display of libraries and symbols.

## Sélection et gestion des bibliothèques

The selection of the current library is possible via the  icon which shows you all available libraries and allows you to select one. When a symbol is loaded or saved, it will be put in this library. The library name of a symbol is the contents of its `Value` field.

### Select and Save a Symbol

#### Symbol Selection

Clicking the  icon on the left tool bar toggles the treeview of libraries and symbols. Clicking on a symbol opens that symbol.

#### NOTE

Some symbols are derived from other symbols. Derived symbol names are displayed in *italics* in the treeview. If a derived symbol is opened, its symbol graphics will not be editable. Its symbol fields will be editable as normal. To edit the graphics of a base symbol and all of its derived symbols, open the base symbol.

#### Save a Symbol

After modification, a symbol can be saved in the current library or a different library.

To save the modified symbol in the current library, click the  icon. The modifications will be written to the existing symbol.

#### NOTE


Saving a modified symbol also saves all other modified symbols in the same library.

To save the symbol changes to a new symbol, click **File** → **Save As....** The symbol can be saved in the current library or a different library. A new name can be set for the symbol.

To create a new file containing only the current symbol, click **File** → **Export** → **Symbol....** This file will be a standard library file which will contain only one symbol.

## Creating Library Symbols

### Create a New Symbol

A new symbol can be created by clicking the  icon. You will be asked for a number of symbol properties.

- A symbol name (this name is used as the default value for the `Value` field in the schematic editor)
- An optional base symbol to derive the new symbol from. The new symbol will use the base symbol's graphical shape and pin configuration, but other symbol information can be modified in the derived symbol. The base symbol must be in the same library as the new derived symbol.
- The reference designator prefix ( U , C , R ...).
- The number of units per package, and whether those units are interchangeable (for example a 7400 is made of 4 units per package).
- If an alternate body style (sometimes referred to as a "De Morgan equivalent") is desired.
- Whether the symbol is a power symbol. Power symbols appear in the "Add Power Port" dialog in the Schematic editor, their `Value` fields are not editable in the schematic, they cannot be assigned a

footprint and they are not added to the PCB, and they are not included in the bill of materials.

- Whether the symbol should be excluded from the bill of materials.
- Whether the symbol should be excluded from the PCB.

There are also several graphical options.

- The offset between the end of each pin and its pin name.
- Whether the pin number and pin name should be displayed.
- Whether the pin names should be displayed alongside the pins or at the ends of the pins inside the symbol body.

These properties can also be changed later in the [Symbol Properties window](#).

**New Symbol**

Symbol name:

Derive from existing symbol:

Default reference designator:

Number of units per package:

☐ Units are not interchangeable

☐ Create symbol with alternate body style (De Morgan)

☐ Create symbol as power symbol

☐ Exclude from schematic bill of materials

☐ Exclude from board

Pin name position offset:  mm

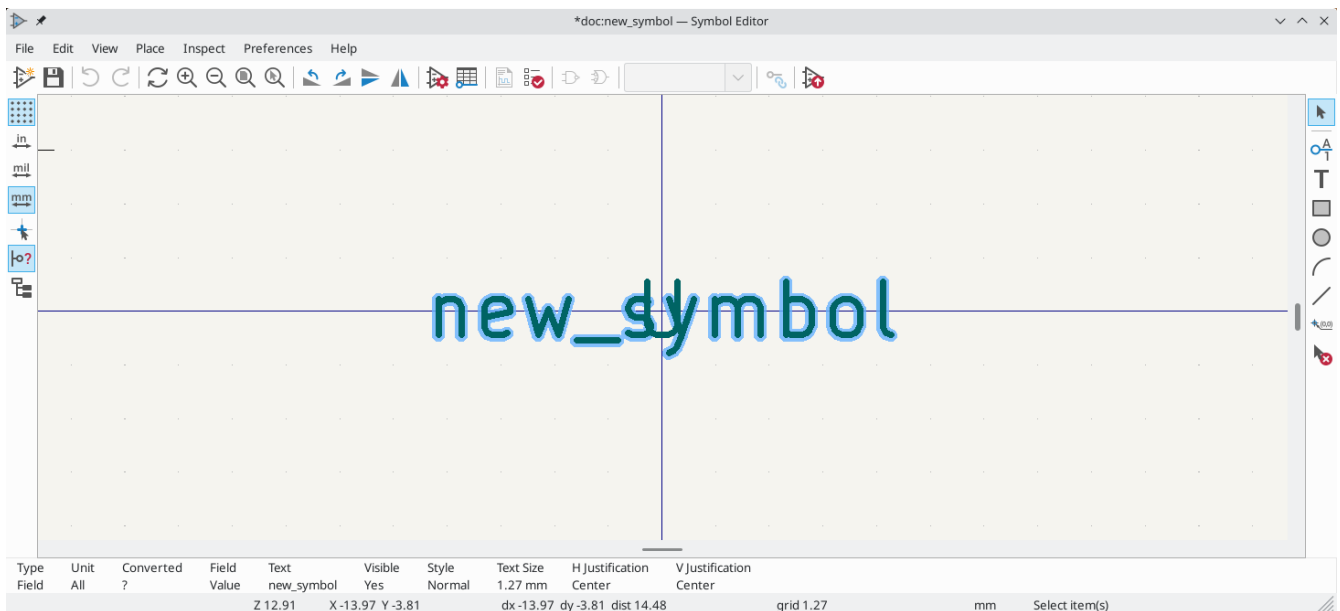
☒ Show pin number text


☒ Show pin name text

☒ Pin name inside

A new symbol will be created using the properties above and will appear in the editor as shown below.






The blue cross in the center is the symbol anchor, which specifies the symbol origin i.e. the coordinates (0, 0). The anchor can be repositioned by selecting the  icon and clicking on the new desired anchor position.

## Create a Symbol from Another Symbol

Often, the symbol that you want to make is similar to one already in a symbol library. In this case it is easy to load and modify an existing symbol.

- Load the symbol which will be used as a starting point.
- Save a new copy of the symbol using **File** → **Save As....** The Save As dialog will prompt for a name for the new symbol and the library to save it in.
- Edit the new symbol as required.
- Save the modified symbol.

## Symbol Properties

Symbol properties are set when the symbol is created but they can be modified at any point. To change the symbol properties, click on the  icon to show the dialog below.

Library Symbol Properties

General **Footprint Filters**

Fields

Name	Value	Show	H Align	V Align	Italic	Bold	Text Size
Reference	U	<input type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.27 mm
Value	LM2904	<input checked="" type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.27 mm
Footprint		<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.27 mm
Datasheet	<a href="http://www.ti.com/lit/ds/symlink/lm358.pdf">http://www.ti.com/lit/ds/symlink/lm358.pdf</a>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.27 mm

+ ↑ ↓

Symbol name: LM2904

Description: Dual Operational Amplifiers, DIP-8/SOIC-8/TSSOP-8/VSSOP-8

Keywords: dual opamp

Derive from symbol:

Symbol

☐ Has alternate body style (De Morgan)

☐ Define as power symbol

☐ Exclude from schematic bill of materials

☐ Exclude from board

Number of Units: 3 — +

☐ All units are interchangeable

Pin Text Options

☒ Show pin number

☒ Show pin name

☒ Place pin names inside

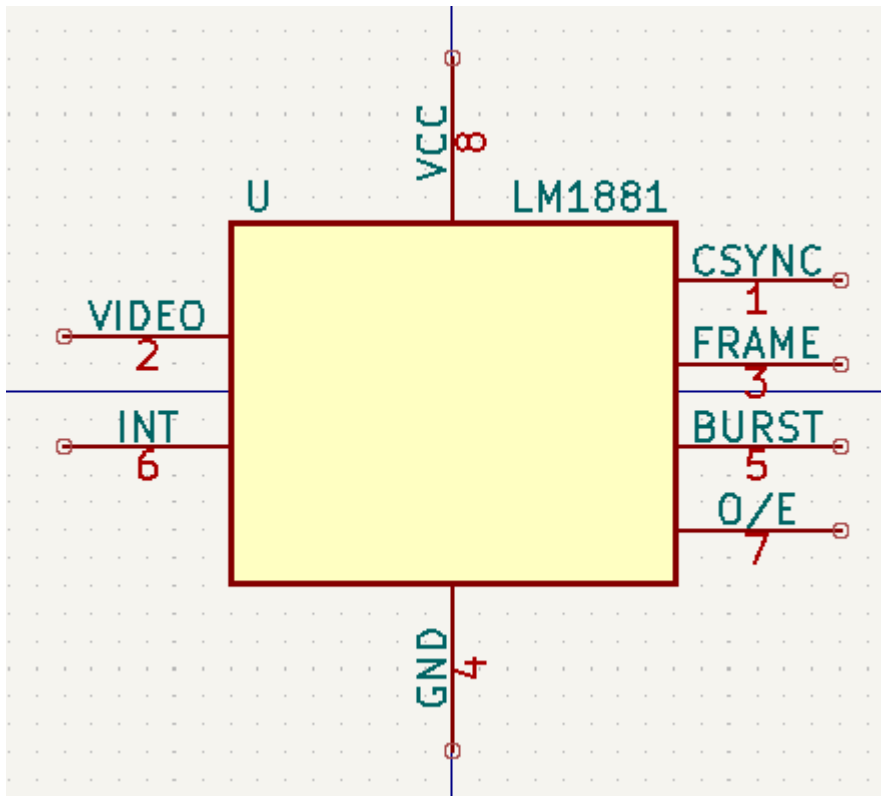
Position offset: 0.127 mm

Edit Spice Model... Cancel OK

It is important to correctly set the number of units per package and the alternate symbolic representation, if enabled, because when pins are edited or created the corresponding pins for each unit will be affected. If you change the number of units per package after pin creation and editing, there will be additional work to specify the pins and graphics for the new unit. Nevertheless, it is possible to modify these properties at any time.

The graphic options "Show pin number" and "Show pin name" define the visibility of the pin number and pin name text. The option "Place pin names inside" defines the pin name position relative to the pin body. The pin names will be displayed inside the symbol outline if the option is checked. In this case the "Pin Name Position Offset" property defines the shift of the text away from the body end of the pin. A value from 0.02 to 0.05 inches is usually reasonable.

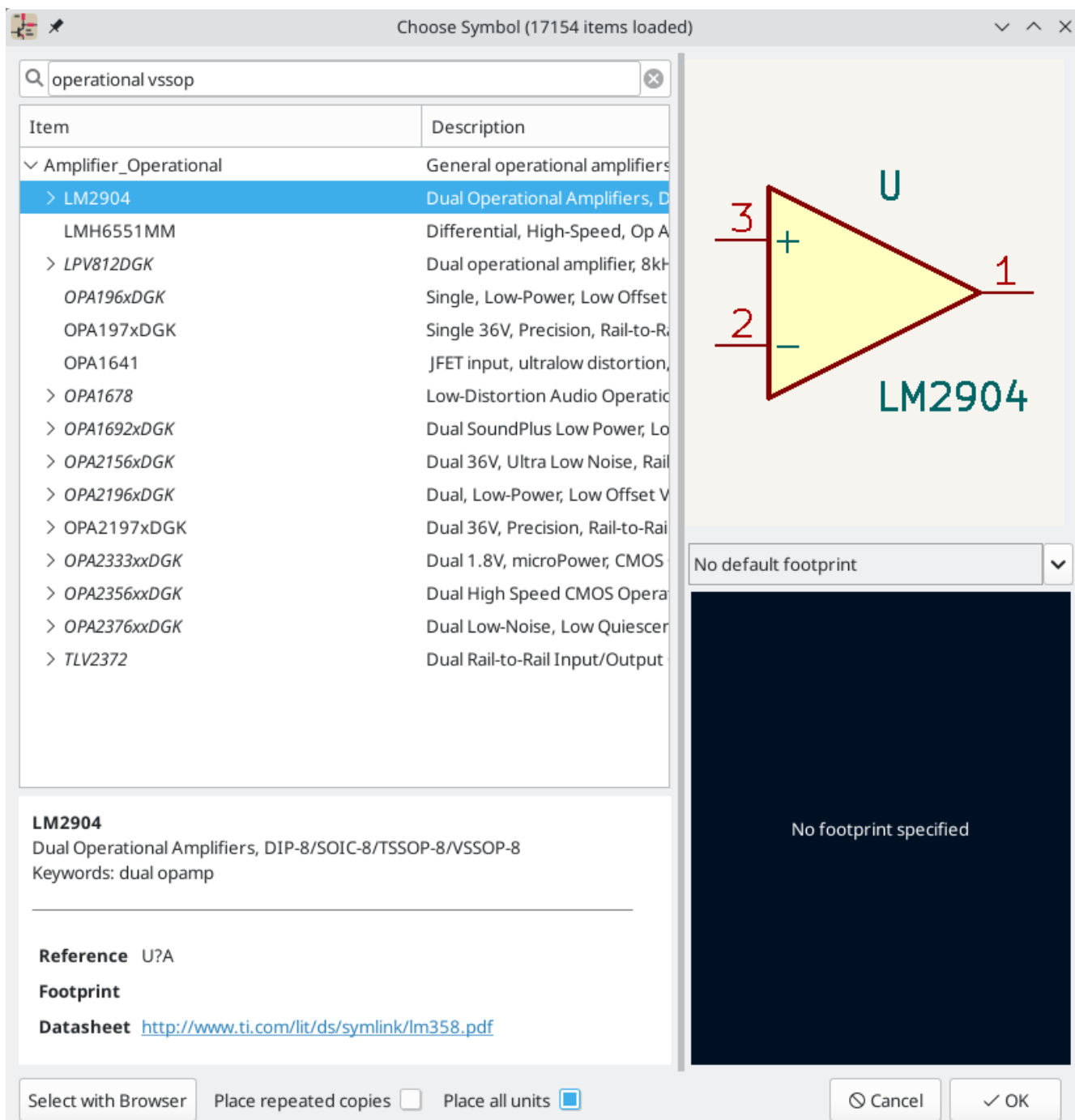
The example below shows a symbol with the "Place pin name inside" option unchecked. Notice the position of the names and pin numbers.



### Symbol Name, Description, and Keywords

The symbol's name is the same as the `Value` field. When the symbol name is changed the value also changes, and vice versa. The symbol's name in the library also changes accordingly.

The symbol description should contain a brief description of the component, such as the component function, distinguishing features, and package options. The keywords should contain additional terms related to the component. Keywords are used primarily to assist in searching for the symbol.



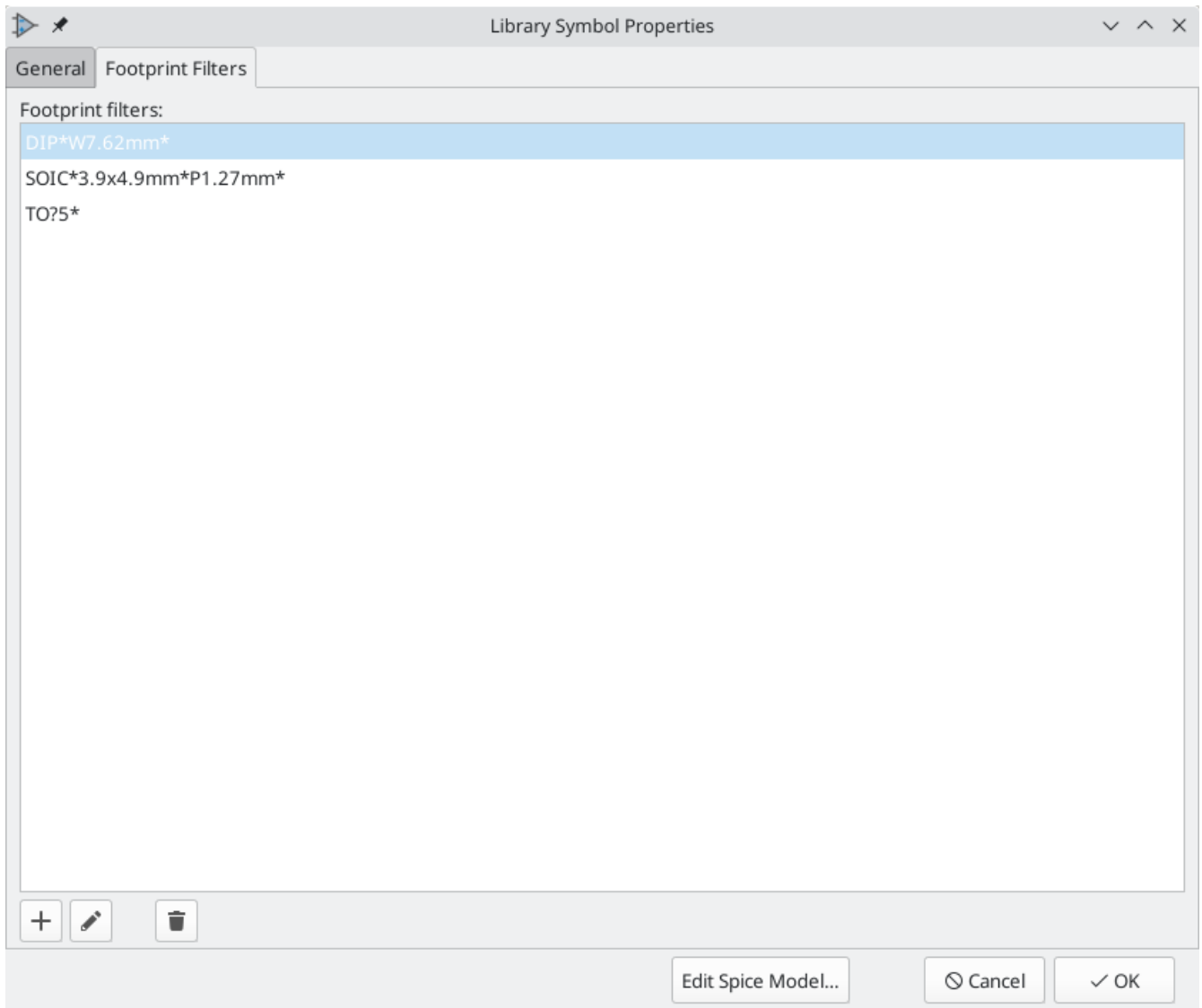
A symbol's name, description, and keywords are all used when searching for symbols in the Symbol Editor and Add a Symbol dialog. The description and keywords are displayed in the Symbol Library Browser and Add a Symbol dialog.

## Footprint Filters


The footprint filters tab is used to define which footprints are appropriate to use with the symbol. The filters can be applied in the Footprint Assignment tool so that only appropriate footprints are displayed for each symbol.

Multiple footprint filters can be defined. Footprints that match any of the filters will be displayed; if no filters are defined, then all footprints will be displayed.

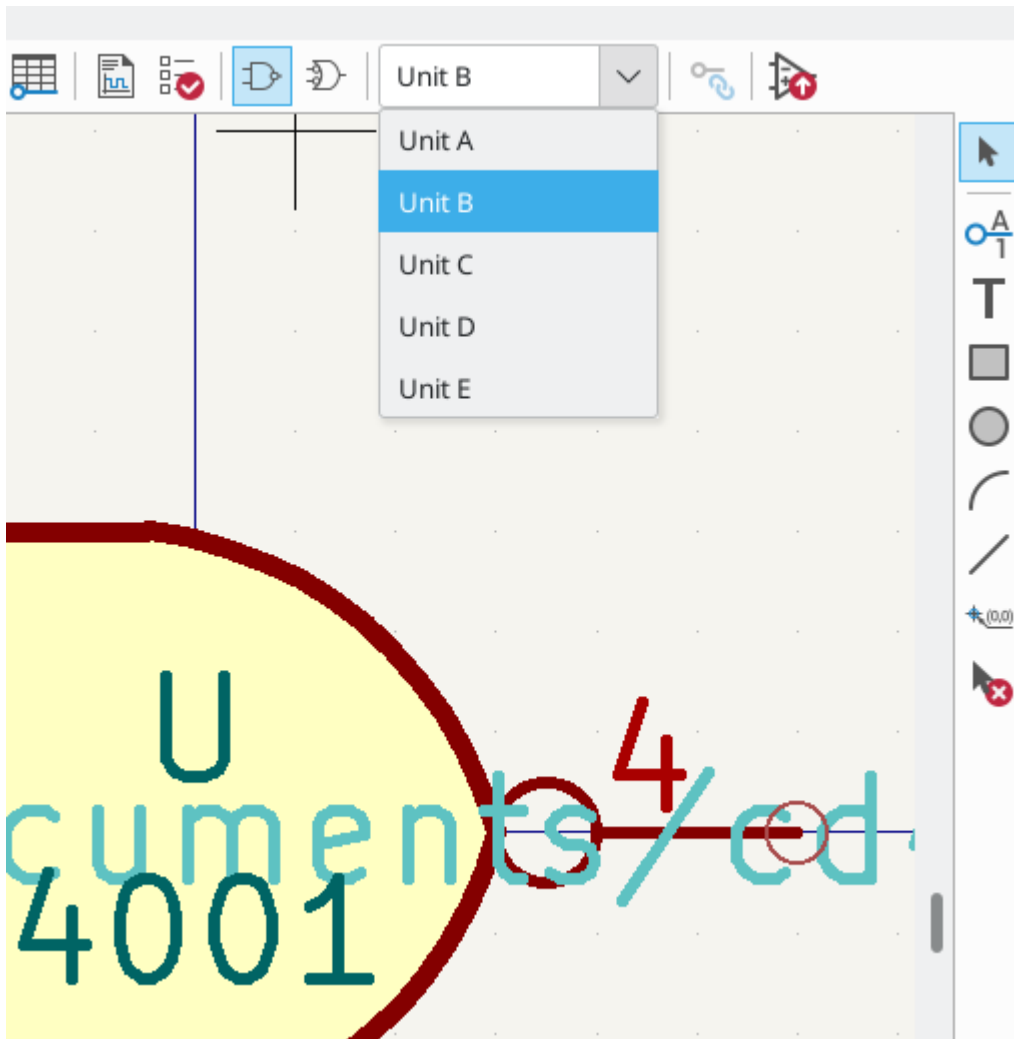
Filters can use wildcards: `*` matches any number of characters, including zero, and `?` matches zero or one characters. For example, `SOIC-*` would match the `SOIC-8_3.9x4.9mm_P1.27mm` footprint as well as any other footprint beginning with `SOIC-`. The filter `SOT?23` matches `SOT23` as well as `SOT-23`.



## Symbols with Alternate Symbolic Representation

If the symbol has an alternate body style defined, one body style must be selected for editing at a time. To edit the normal representation, click the  icon.

To edit the alternate representation, click on the  icon. Use the  dropdown shown below to select the unit you wish to edit.



## Éléments graphiques

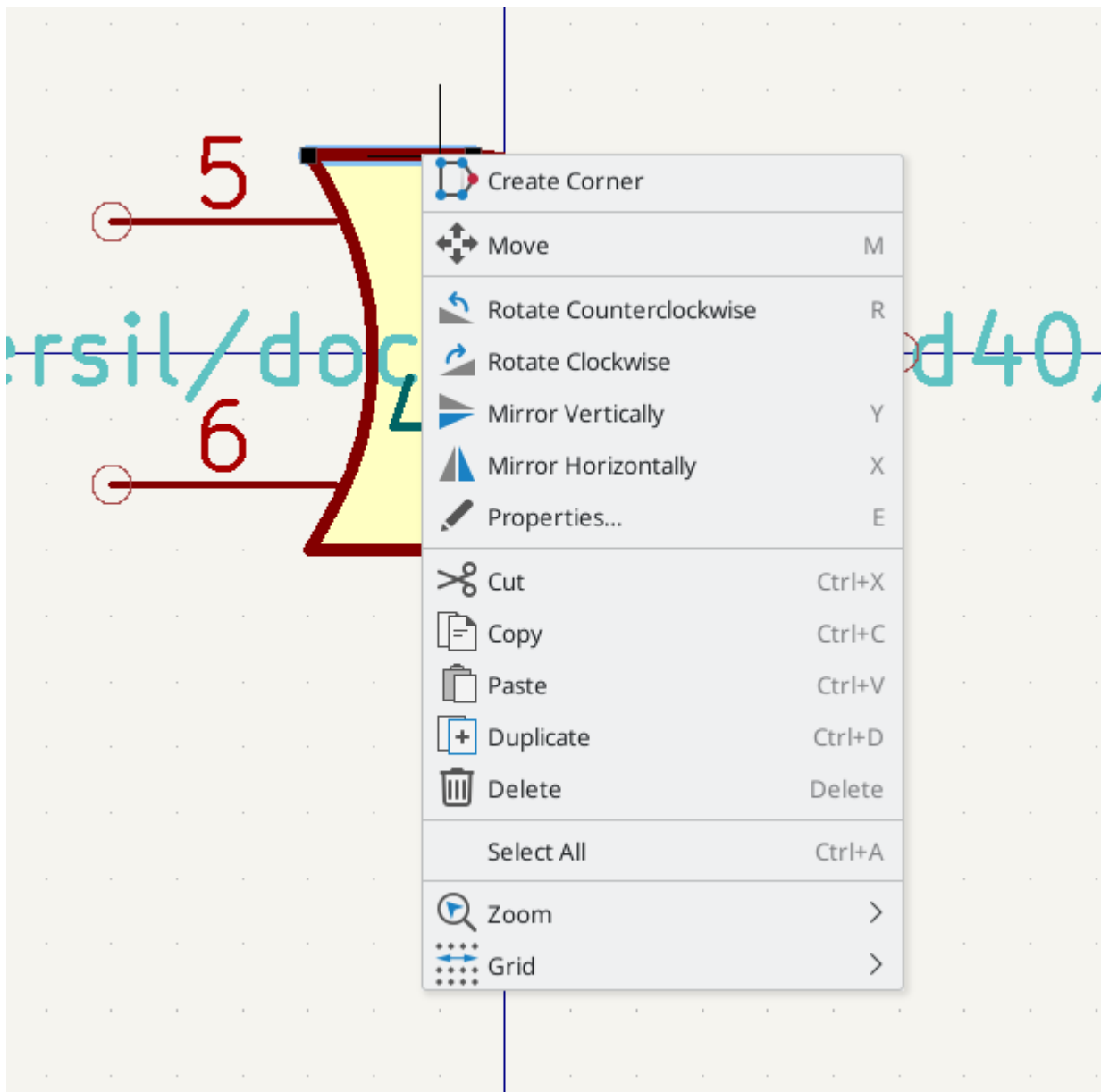
Graphical elements create the visual representation of a symbol and contain no electrical connection information. Graphical elements are created with the following tools:

- Lignes et polygones définis par des points d'origine et des points de fin.
- Rectangles définis par leurs deux coins opposés sur la diagonale.
- Cercles définis par leur centre et leur rayon.
- Arcs de cercles définis par leur centre et leurs points de départ et de fin. Un arc peut aller de 0 à 180°.

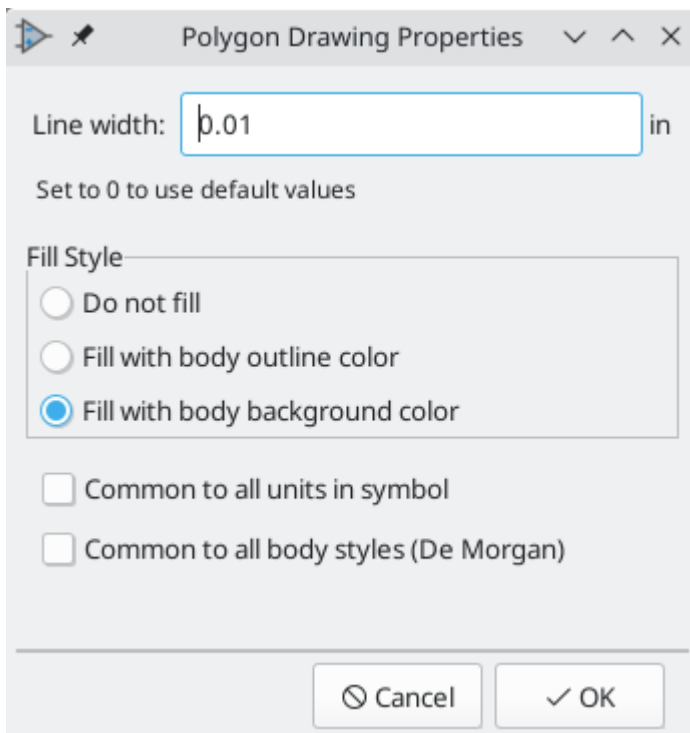
The vertical toolbar on the right hand side of the main window allows you to place all of the graphical elements required to design the representation of a symbol.

## Appartenance des éléments graphiques

Chaque élément graphique, (ligne, arc, cercle, etc...), peut être défini comme commun à toutes les unités et/ou représentations, ou spécifique à une unité donnée et/ou une représentation. Les options des éléments sont accessibles rapidement par le menu contextuel : clic droit sur l'élément à modifier. Ci-dessous, le menu contextuel pour un élément de type ligne.



Vous pouvez aussi double-cliquer sur un élément et modifier ses propriétés. Ci-dessous, la fenêtre des propriétés pour un élément de type polygone.



Les propriétés d'un élément graphique sont :

- "Line width" defines the width of the element's line in the current drawing units.
- "Fill Style" determines if the shape defined by the graphical element is to be drawn unfilled, background filled, or foreground filled.
- "Common to all units in symbol" determines if the graphical element is drawn for each unit in symbol with more than one unit per package or if the graphical element is only drawn for the current unit.
- "Common to all body styles (De Morgan)" determines if the graphical element is drawn for each symbolic representation in symbols with an alternate body style or if the graphical element is only drawn for the current body style.

## Éléments Graphiques Textes

The **T** icon allows for the creation of graphical text. Graphical text is automatically oriented to be readable, even when the symbol is mirrored. Please note that graphical text items are not the same as symbol fields.

## Multiple Units per Symbol and Alternate Body Styles

Symbols can have up to two body styles (a standard symbol and an alternate symbol often referred to as a "De Morgan equivalent") and/or have more than one unit per package (logic gates for example). Some symbols can have more than one unit per package each with different symbols and pin configurations.

Consider for instance a relay with two switches, which can be designed as a symbol with three different units: a coil, switch 1, and switch 2. Designing a symbol with multiple units per package and/or alternate body styles is very flexible. A pin or a body symbol item can be common to all units or specific to a given unit or they can be common to both symbolic representation so are specific to a given symbol representation.

By default, pins are specific to a unit and body style. When a pin is common to all units or all body styles, it only needs to be created once. This is also the case for the body style graphic shapes and text, which may be



common to each unit, but typically are specific to each body style).

## Example of a Symbol With Multiple Noninterchangeable Units

For an example of a symbol with multiple units that are not interchangeable, consider a relay with 3 units per package: a coil, switch 1, and switch 2.

The three units are not all the same, so "All units are interchangeable" should be deselected in the Symbol Properties dialog. Alternatively, this option could have been specified when the symbol was initially created.

The image shows the 'Library Symbol Properties' dialog box for a symbol named 'G5V-2\_Split'. The 'Footprint Filters' tab is active. The 'Fields' section contains a table with four rows: Reference, Value, Footprint, and Datasheet. The 'Reference' row is selected. Below the table are buttons for adding, moving up, moving down, and deleting fields. The 'Symbol name' is 'G5V-2\_Split', the 'Description' is 'Relay Miniature Omron DPDT', and the 'Keywords' are 'Miniature Relay Dual Pole DPDT Omron'. The 'Derive from symbol' dropdown is empty. The 'Symbol' section has four checkboxes: 'Has alternate body style (De Morgan)', 'Define as power symbol', 'Exclude from schematic bill of materials', and 'Exclude from board', all of which are unchecked. The 'Number of Units' is set to 3. The 'All units are interchangeable' checkbox is unchecked. The 'Pin Text Options' section has three checkboxes: 'Show pin number', 'Show pin name', and 'Place pin names inside', all of which are checked. The 'Position offset' is set to 0.02. At the bottom are buttons for 'Edit Spice Model...', 'Cancel', and 'OK'.

Name	Value	Show	H Align	V Align	Italic	Bold	Text Size
Reference	K	<input type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	0.05 in
Value	G5V-2_Split	<input checked="" type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	0.05 in
Footprint	Relay_THT:Relay_DPDT_Omron_G5V-2	<input type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	0.05 in
Datasheet	<a href="http://omronfs.omron.com/en_US/ecb/products/pdf/en-g5v">http://omronfs.omron.com/en_US/ecb/products/pdf/en-g5v</a>	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	0.05 in

Symbol name: G5V-2\_Split

Description: Relay Miniature Omron DPDT

Keywords: Miniature Relay Dual Pole DPDT Omron

Derive from symbol: ▼

Symbol

- ☐ Has alternate body style (De Morgan)
- ☐ Define as power symbol
- ☐ Exclude from schematic bill of materials
- ☐ Exclude from board

Number of Units: 3 — +

☐ All units are interchangeable

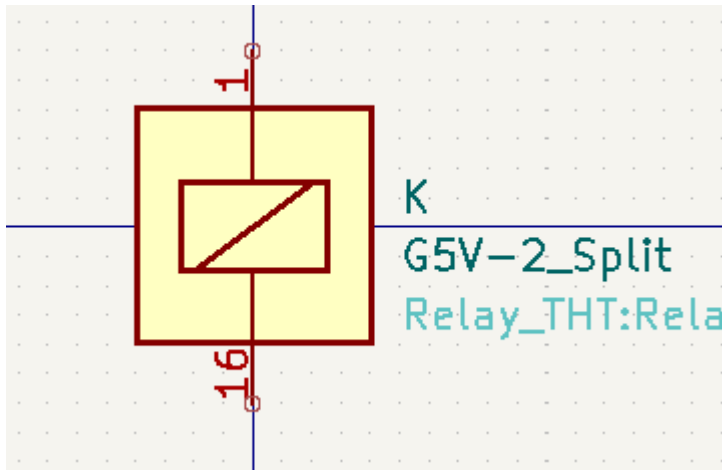
Pin Text Options

- ☒ Show pin number
- ☒ Show pin name
- ☒ Place pin names inside

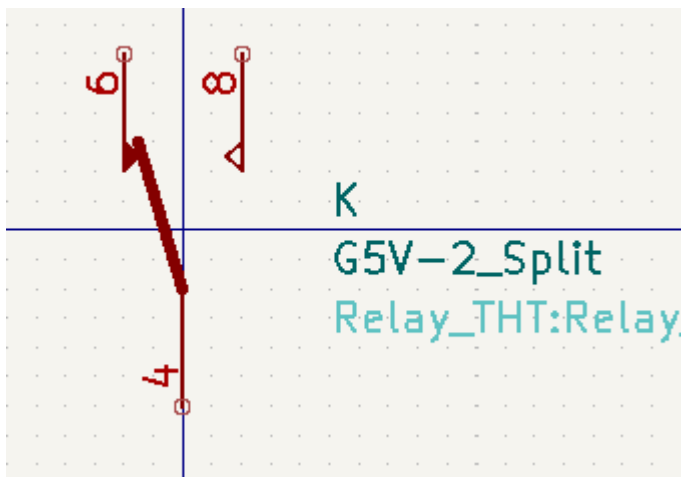
Position offset: 0.02 in

Edit Spice Model... Cancel OK

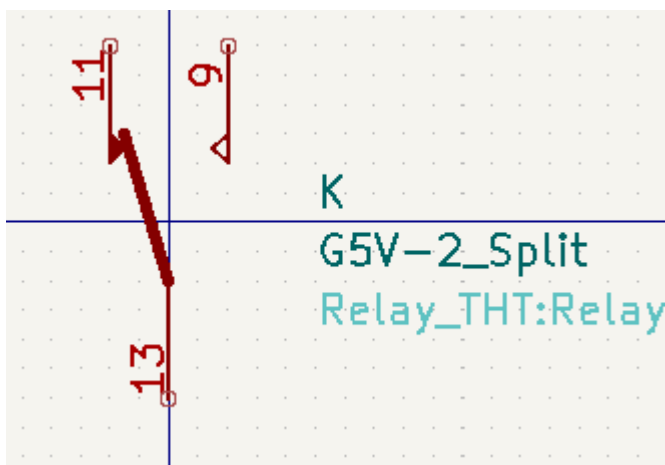
## Unit A



## Unit B




## Unit C



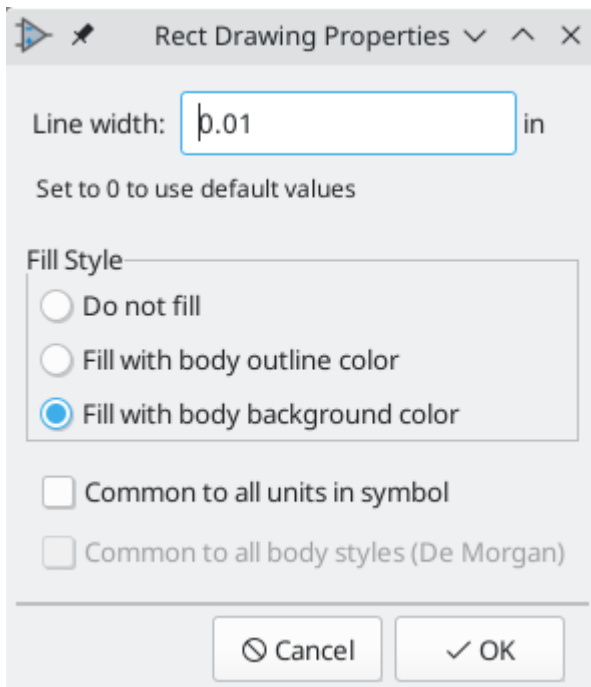
Unit A does not have the same symbol and pin layout as Units B and C, so the units are not interchangeable.

### NOTE

"Synchronized Pins Edit Mode" can be enabled by clicking the  icon. In this mode, pin modifications are propagated between symbol units; changes made in one unit will be reflected in the other units as well. When this mode is disabled, pin changes made in one unit do not affect other units. This mode is enabled automatically when "All units are interchangeable" is checked, but it can be disabled. The mode cannot be enabled when "All units are interchangeable" is unchecked or when the symbol only has one unit.

## Éléments graphiques symboliques

Shown below are properties for a graphic body element. In the relay example above, the three units have different symbolic representations. Therefore, each unit was created separately and the graphical body elements have the "Common to all units in symbol" setting disabled.



## Création et édition de pins

You can click on the  icon to create and insert a pin. The editing of all pin properties is done by double-clicking on the pin or right-clicking on the pin to open the pin context menu. Pins must be created carefully, because any error will have consequences on the PCB design. Any pin already placed can be edited, deleted, and/or moved.

## Généralités sur les pins

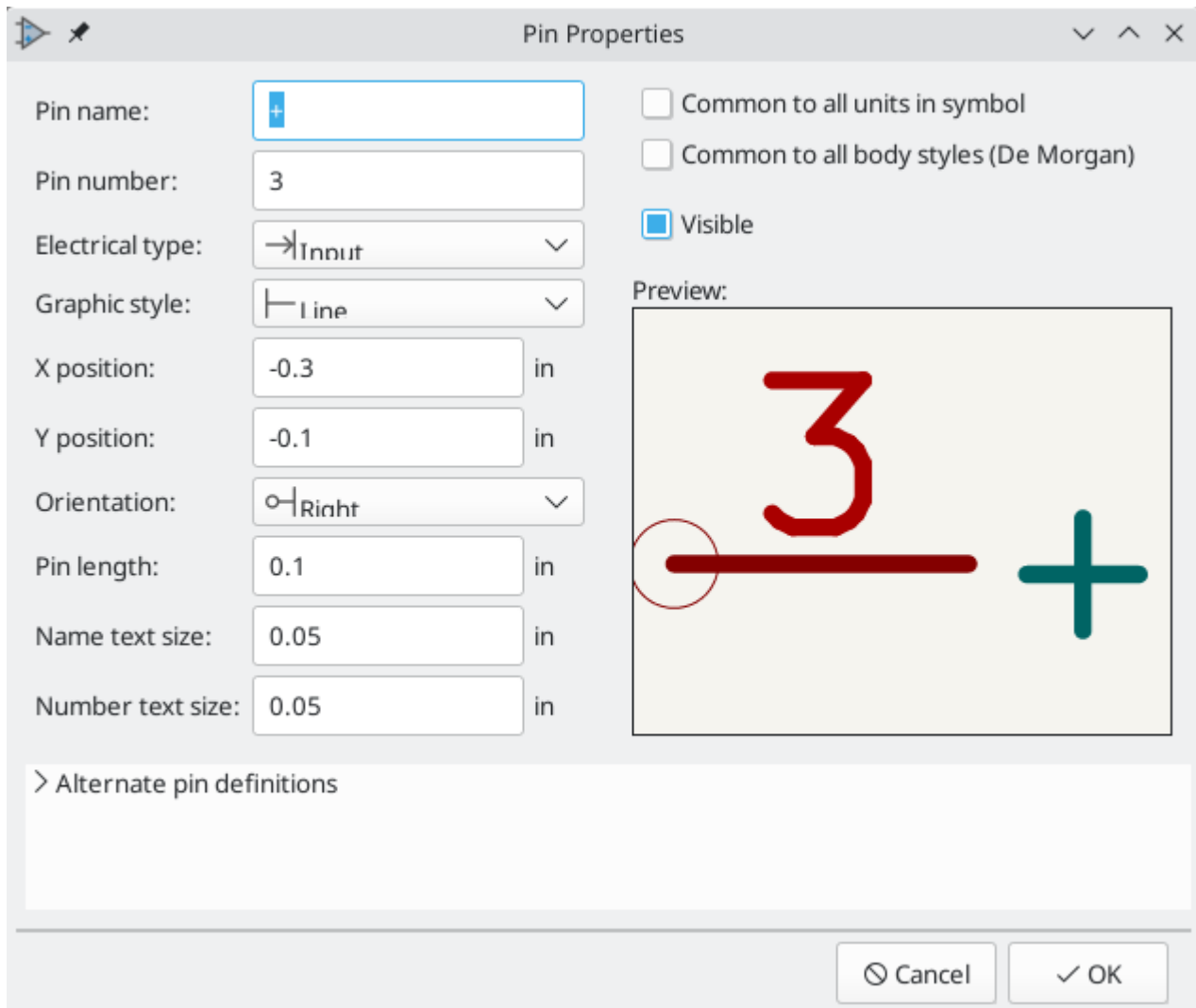
A pin is defined by its graphical representation, its name and its number. The pin's name and number can contain letters, numbers, and symbols, but not spaces. For the Electrical Rules Check (ERC) tool to be useful, the pin's electrical type (input, output, tri-state...) must also be defined correctly. If this type is not defined properly, the schematic ERC check results may be invalid.

Notes importantes :

- Symbol pins are matched to footprint pads by number. The pin number in the symbol must match the corresponding pad number in the footprint.
- Do not use spaces in pin names and numbers. Spaces will be automatically replaced with underscores (`_`).
- To define a pin name with an inverted signal (overline) use the `~` (tilde) character followed by the text to invert in braces. For example `~{FO}O` would display  $\overline{FO}$  O.
- If the pin name is empty, the pin is considered unnamed.
- Pin names can be repeated in a symbol.

Pin numbers must be unique in a symbol.

## Propriétés des pins



The image shows a 'Pin Properties' dialog box with various settings for a pin. The settings are as follows:

Property	Value
Pin name:	+
Pin number:	3
Electrical type:	Input
Graphic style:	Line
X position:	-0.3 in
Y position:	-0.1 in
Orientation:	Right
Pin length:	0.1 in
Name text size:	0.05 in
Number text size:	0.05 in

Additional options on the right:

- ☐ Common to all units in symbol
- ☐ Common to all body styles (De Morgan)
- ☒ Visible

Preview: A diagram showing a red horizontal line with a red circle at its left end. Above the line is a large red number '3'. To the right of the line is a large teal plus sign '+'. The red circle is highlighted with a red outline.

> Alternate pin definitions

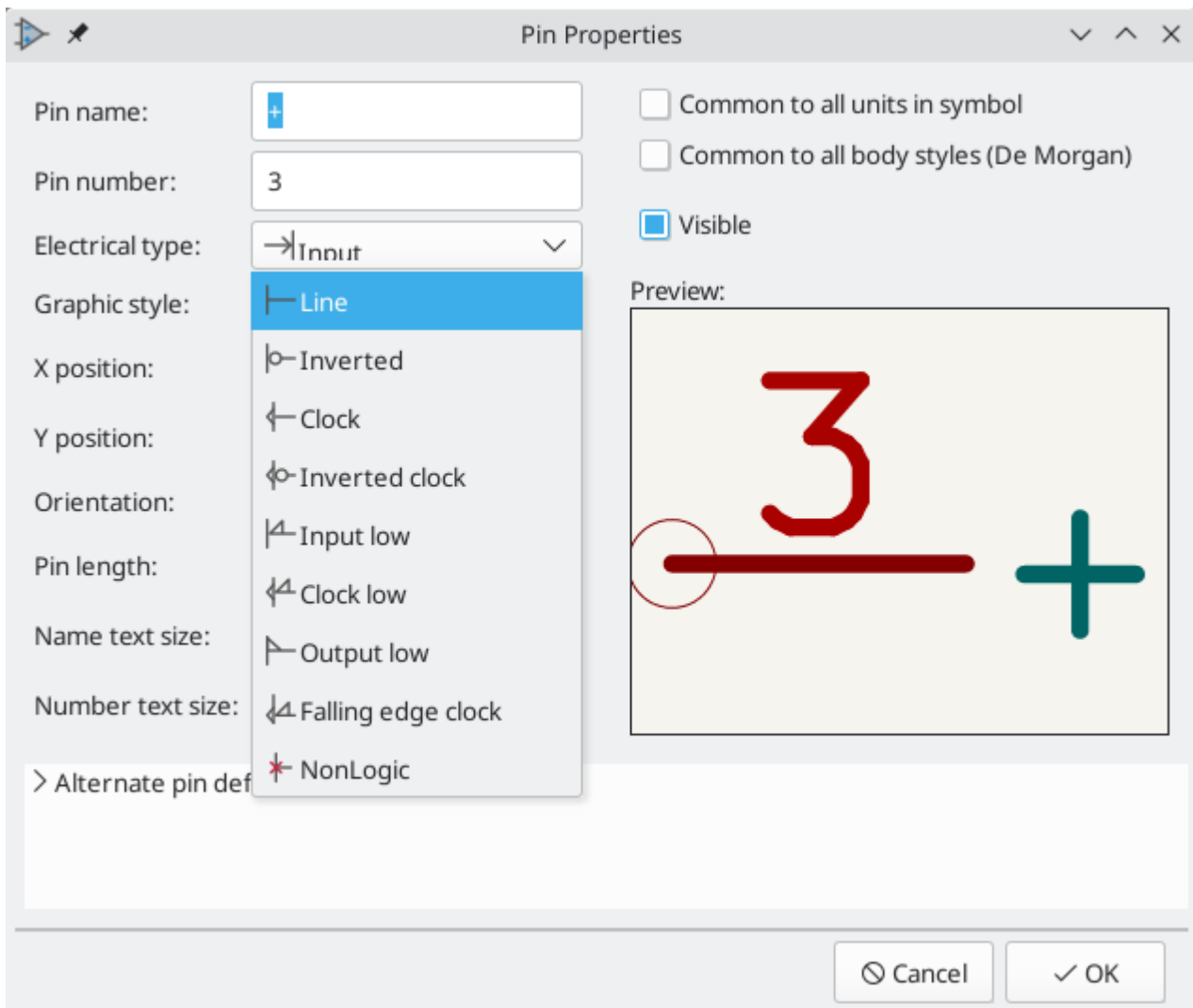
Buttons: Cancel, OK

La fenêtre des propriétés des pins vous permet de modifier toutes les caractéristiques d'une pin. Cette fenêtre apparaît automatiquement à la création de la pin ou quand vous double-cliquez sur une pin existante. Vous pouvez modifier :

- The pin name and text size.
- The pin number and text size.
- The pin length.
- The pin electrical type and graphical style.
- Son appartenance aux unités et aux représentations alternatives.
- Pin visibility.
- [Alternate pin definitions](#).

## Pin Graphic Styles

Shown in the figure below are the different pin graphic styles. The choice of graphic style does not have any influence on the pin's electrical type.



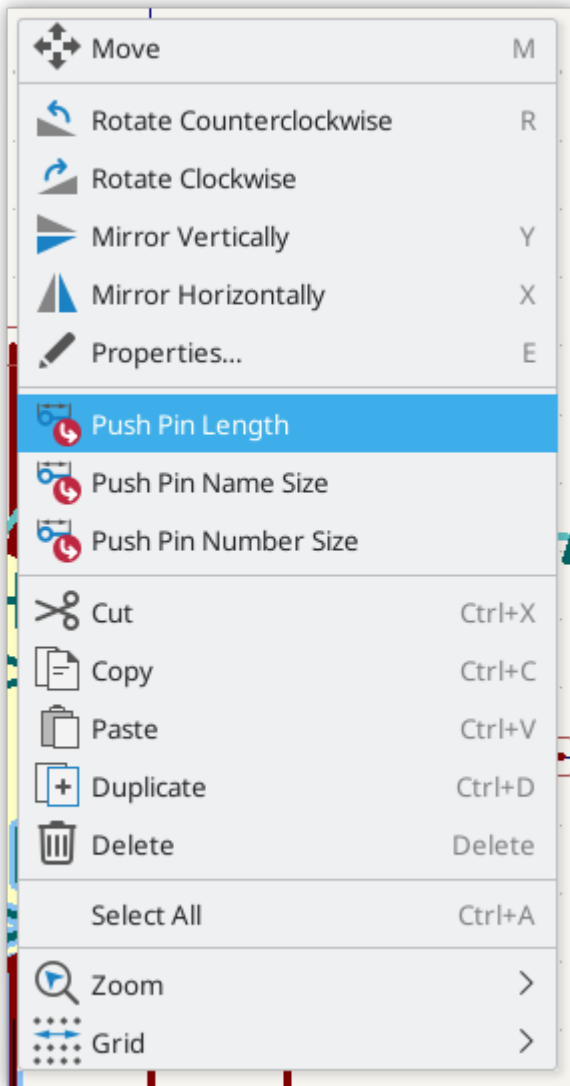
## Types électriques des pins

Choosing the correct electrical type is important for the schematic ERC tool. ERC will check that pins are connected appropriately, for example ensuring that input pins are driven and power inputs receive power from an appropriate source.

Pin Type	Description
Input	A pin which is exclusively an input.
Output	A pin which is exclusively an output.
Bidirectional	A pin that can be either an input or an output, such as a microcontroller data bus pin.
Tri-state	A three state output pin (high, low, or high impedance)
Passive	A passive symbol pin: resistors, connectors, etc.
Free	A pin that can be freely connected to any other pin without electrical concerns.
Unspecified	A pin for which the ERC check does not matter.
Power input	A symbol's power pin. As a special case, power input pins that are marked invisible are automatically connected to the net with the same name. See the <a href="#">Power Ports section</a> for more information.
Power output	A pin that provides power to other pins, such as a regulator output.
Open collector	An open collector logic output.
Open emitter	An open emitter logic output.
Unconnected	A pin that should not be connected to anything.


## Pushing Pin Properties to Other Pins

You can apply the length, name size, or number size of a pin to the other pins in the symbol by right clicking the pin and selecting **Push Pin Length**, **Push Pin Name Size**, or **Push Pin Number Size**, respectively.





## Définitions de pins pour unités multiples et représentations alternatives

Symbols with multiple units and/or graphical representations are particularly problematic when creating and editing pins. The majority of pins are specific to each symbol unit (because each unit has a different set of pins) and to each body style (because the form and position is different between the normal body style and the alternate form).


The symbol library editor allows the simultaneous creation of pins. By default, changes made to a pin are made for all units of a multiple unit symbol and to both representations for symbols with an alternate symbolic representation. The only exception to this is the pin's graphical type and name, which remain unlinked between symbol units and body styles. This dependency was established to allow for easier pin creation and editing in most cases. This dependency can be disabled by toggling the  icon on the main tool bar. This will allow you to create pins for each unit and representation completely independently.

Pins can be common or specific to different units. Pins can also be common to both symbolic representations or specific to each symbolic representation. When a pin is common to all units, it only has to be drawn once. Pins are set as common or specific in the pin properties dialog.

An example is the output pin in the 7400 quad dual input NAND gate. Since there are four units and two symbolic representations, there are eight separate output pins defined in the symbol definition. When creating a new 7400 symbol, unit A of the normal symbolic representation will be shown in the library

editor. To edit the pin style in the alternate symbolic representation, it must first be enabled by clicking the  button on the tool bar. To edit the pin number for each unit, select the appropriate unit using the  drop down control.

## Pin Table

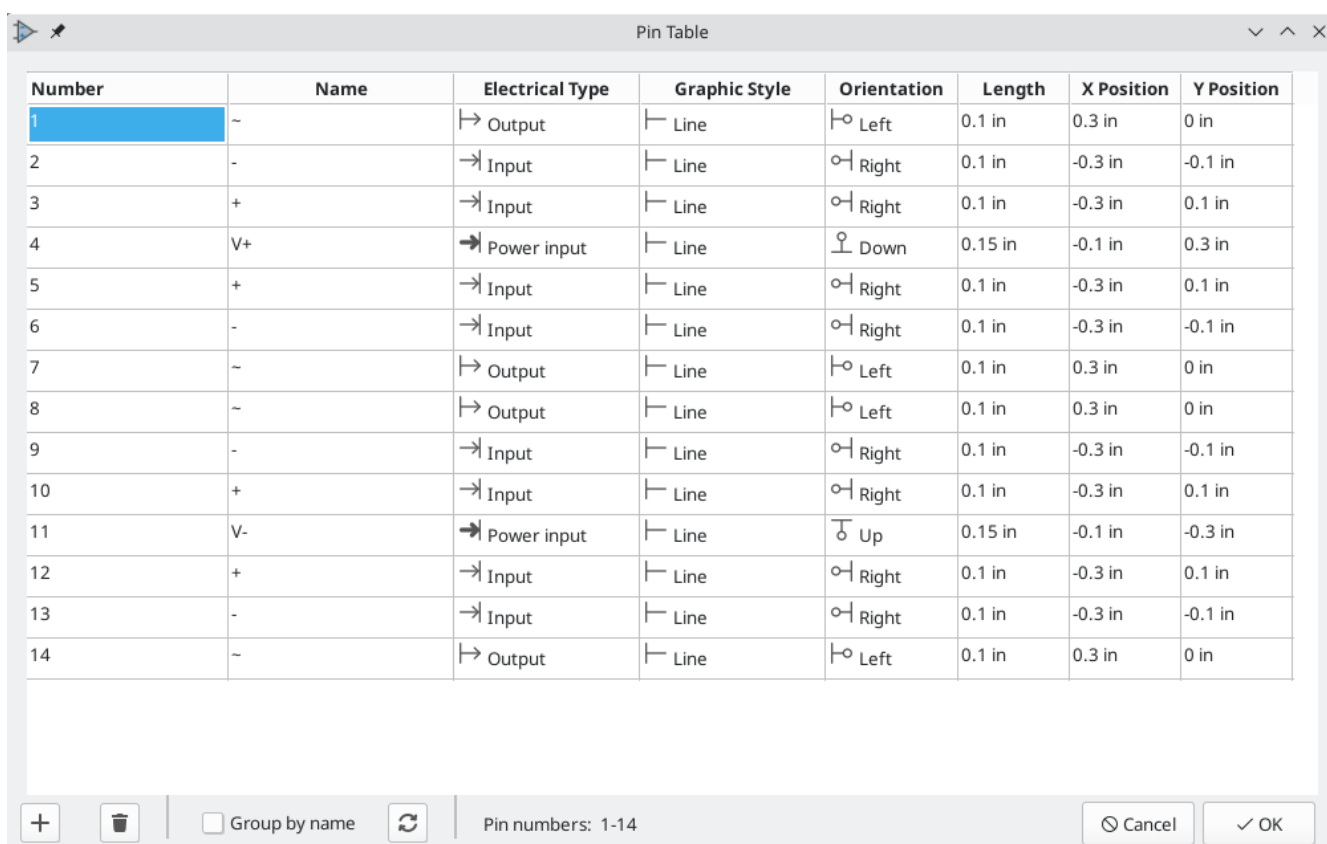
Another way to edit pins is to use the Pin Table, which is accessible via the  icon. The Pin Table displays all of the pins in the symbol and their properties in a table view, so it is useful for making bulk pin changes.

Any pin property can be edited by clicking on the appropriate cell. Pins can be added and removed with the  and  icons, respectively.

### NOTE

Columns of the pin table can be shown or hidden by right-clicking on the header row and checking or unchecking additional columns. Some columns are hidden by default.

The screenshot below shows the pin table for a quad opamp.



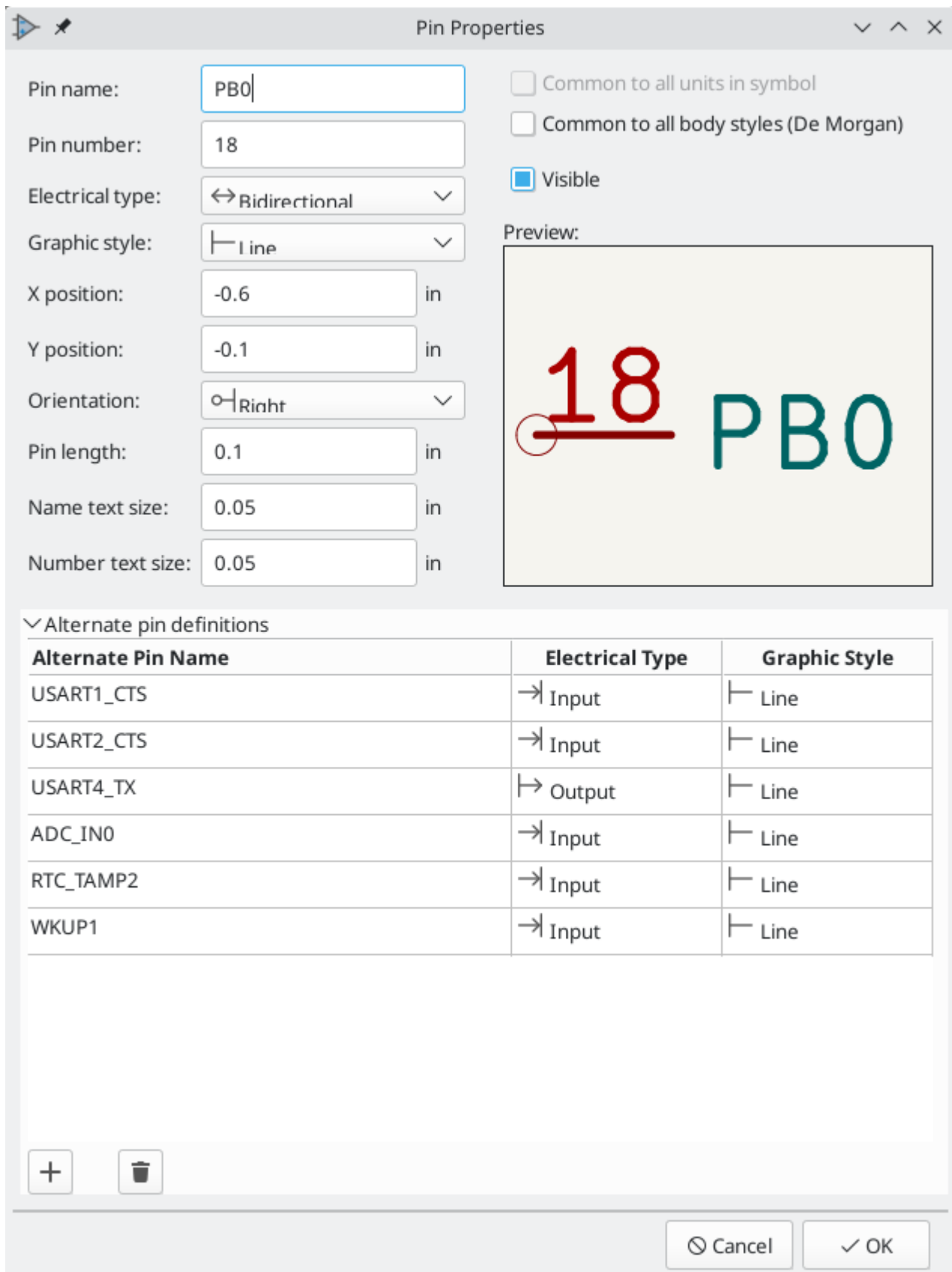
Number	Name	Electrical Type	Graphic Style	Orientation	Length	X Position	Y Position
1	~	Output	Line	Left	0.1 in	0.3 in	0 in
2	-	Input	Line	Right	0.1 in	-0.3 in	-0.1 in
3	+	Input	Line	Right	0.1 in	-0.3 in	0.1 in
4	V+	Power input	Line	Down	0.15 in	-0.1 in	0.3 in
5	+	Input	Line	Right	0.1 in	-0.3 in	0.1 in
6	-	Input	Line	Right	0.1 in	-0.3 in	-0.1 in
7	~	Output	Line	Left	0.1 in	0.3 in	0 in
8	~	Output	Line	Left	0.1 in	0.3 in	0 in
9	-	Input	Line	Right	0.1 in	-0.3 in	-0.1 in
10	+	Input	Line	Right	0.1 in	-0.3 in	0.1 in
11	V-	Power input	Line	Up	0.15 in	-0.1 in	-0.3 in
12	+	Input	Line	Right	0.1 in	-0.3 in	0.1 in
13	-	Input	Line	Right	0.1 in	-0.3 in	-0.1 in
14	~	Output	Line	Left	0.1 in	0.3 in	0 in

## Alternate Pin Definitions

Pins can have alternate pin definitions added to them. Alternate pin definitions allow a user to select a different name, electrical type, and graphical style for a pin when the symbol has been placed in the schematic. This can be used for pins that have multiple functions, such as microcontroller pins.

Alternate pin definitions are added in the Pin Properties dialog as shown below. Each alternate definition contains a pin name, electrical type, and graphic style. This microcontroller pin has all of its peripheral functions defined in the symbol as alternate pin names.





The Pin Properties dialog box is used to configure the properties of a pin symbol. It includes fields for Pin name, Pin number, Electrical type, Graphic style, X position, Y position, Orientation, Pin length, Name text size, and Number text size. It also has checkboxes for 'Common to all units in symbol', 'Common to all body styles (De Morgan)', and 'Visible'. A Preview window shows the resulting pin symbol. Below the main fields is a section for 'Alternate pin definitions' which contains a table of alternate pin names, electrical types, and graphic styles. At the bottom are buttons for '+', a trash icon, 'Cancel', and 'OK'.

Pin name: 
☐ Common to all units in symbol  
☐ Common to all body styles (De Morgan)  
☒ Visible

Pin number:

Electrical type:

Graphic style:

X position:  in


Y position:  in

Orientation:

Pin length:  in

Name text size:  in

Number text size:  in

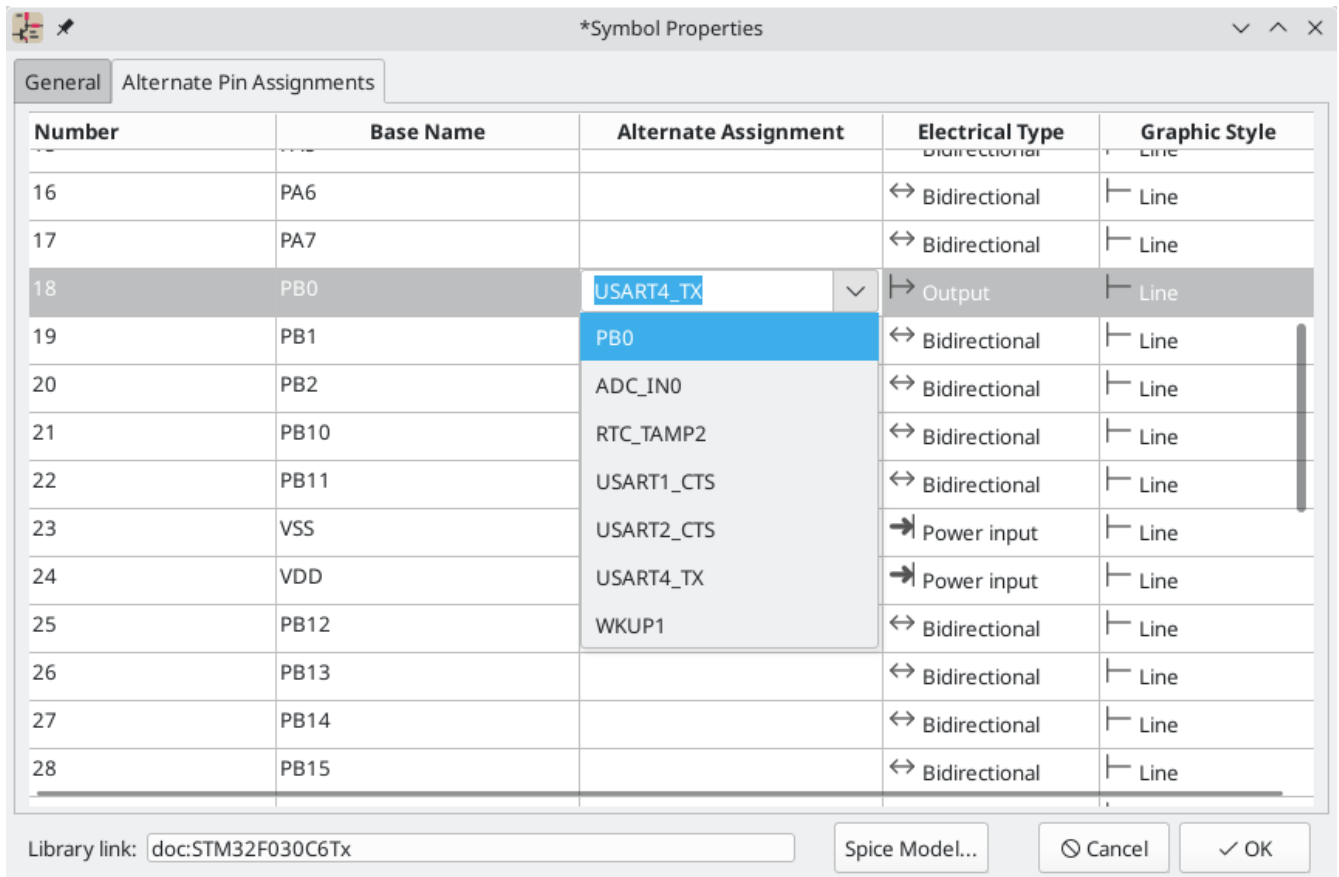
Preview:


The preview shows a pin symbol with the number '18' in red and the name 'PB0' in teal. The number is underlined with a red line, and a red circle is drawn around the pin symbol's body.

▼ Alternate pin definitions

Alternate Pin Name	Electrical Type	Graphic Style
USART1_CTS	→ Input	└─ Line
USART2_CTS	→ Input	└─ Line
USART4_TX	└─ Output	└─ Line
ADC_IN0	→ Input	└─ Line
RTC_TAMP2	→ Input	└─ Line
WKUP1	→ Input	└─ Line

Alternate pin definitions are selected in the Schematic Editor once the symbol has been placed in the schematic. The alternate pin is assigned in the Alternate Pin Assignments tab of the Symbol Properties dialog. Alternate definitions are selectable in the dropdown in the Alternate Assignment column.



## Symbol Fields

All library symbols are defined with four default fields. The reference designator, value, footprint assignment, and datasheet link fields are created whenever a symbol is created or copied. Only the reference designator and value fields are required.

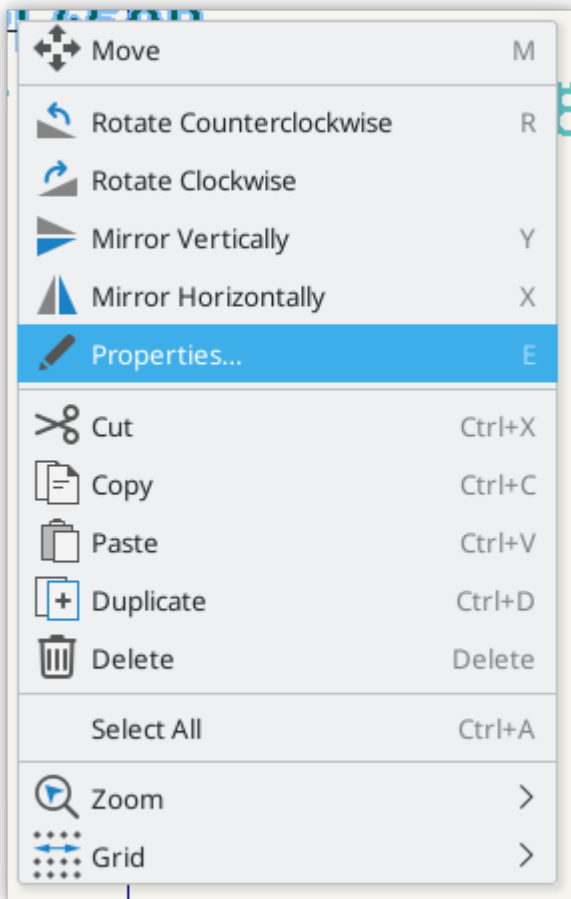
Symbols defined in libraries are typically defined with only these four default fields. Additional fields such as vendor, part number, unit cost, etc. can be added to library symbols but generally this is done in the schematic editor so the additional fields can be applied to all of the symbols in the schematic.


### NOTE

A convenient way to create additional empty symbol fields is to use define field name templates. Field name templates define empty fields that are added to each symbol when it is inserted into the schematic. Field name templates can be defined globally (for all schematics) in the Schematic Editor Preferences, or they can be defined locally (specific to each project) in the Schematic Setup dialog.

## Editing Symbol Fields

To edit an existing symbol field, right-click on the field text to show the field context menu shown below.



To add new fields, delete optional fields, or edit existing fields, use the  icon on the main tool bar to open the [Symbol Properties dialog](#).

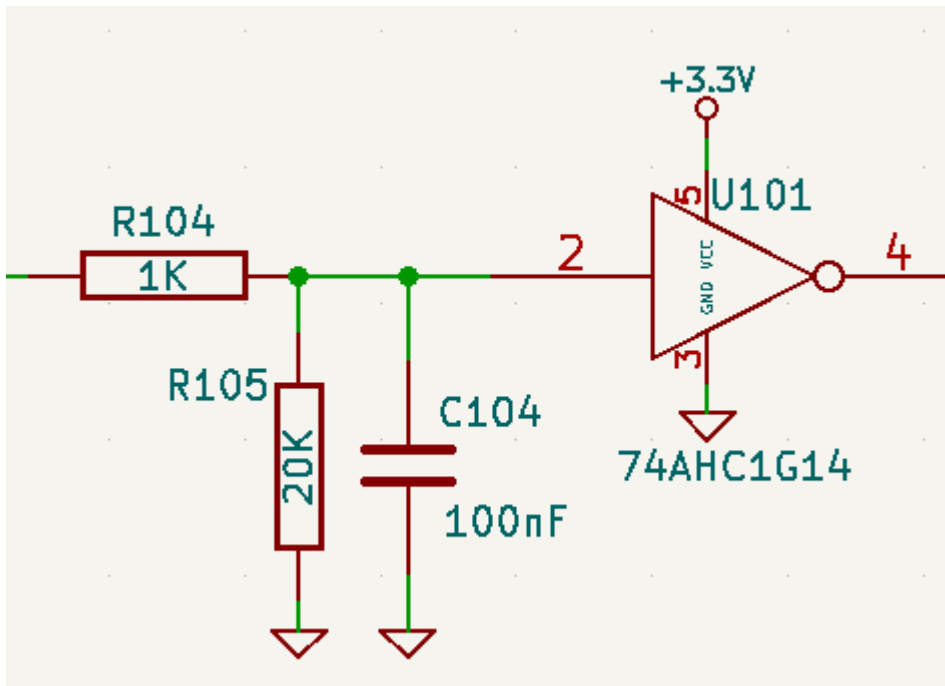
Fields are text information associated a the symbol. Do not confuse them with text in the graphic representation of a symbol.

Notes importantes :

- Modifying the `Value` field changes the name of the symbol. The symbol's name in the library will change when the symbol is saved.
- The Symbol Properties dialog must be used to edit a field that is empty or has the invisible attribute enabled because such fields cannot be clicked on.
- The footprint is defined as an absolute footprint using the `LIBNAME:FOOTPRINTNAME` format where `LIBNAME` is the name of the footprint library defined in the footprint library table (see the "Footprint Library Table" section in the PCB Editor manual) and `FOOTPRINTNAME` is the name of the footprint in the library `LIBNAME`.

## Power Ports

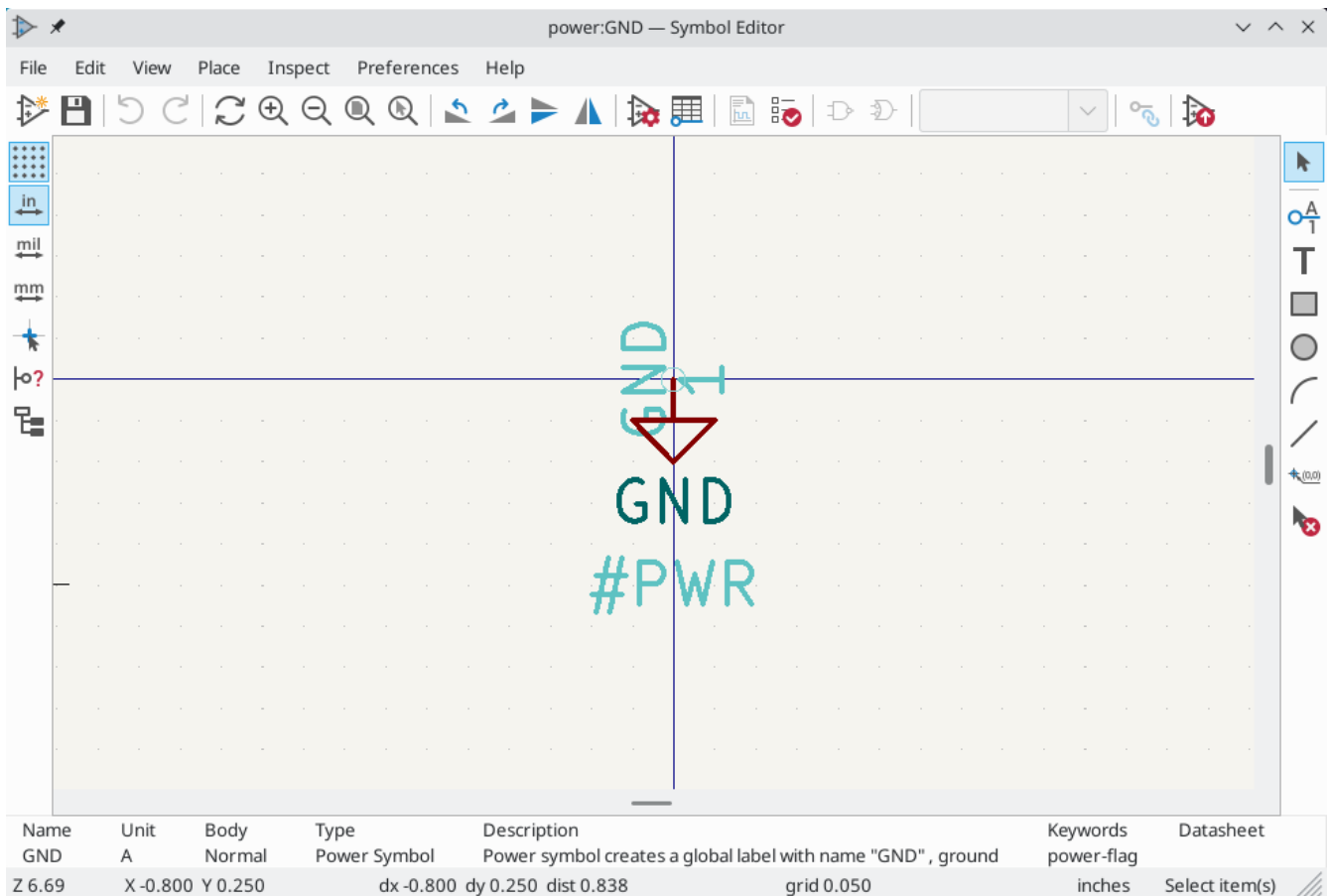
Power ports, or power symbols, are conventionally used to label a wire as part of a power net, like `VCC`, `+5V`, or `GND`. In the schematic below, the `+3.3V` and `GND` symbols are power ports. In addition to acting as a visual indicator that a net is a power rail, a power port will determine the name of the net it is attached to. This is true even if there is another net label attached to the net; the net name determined by the power symbol overrides any other net names.



It may be useful to place power symbols in a dedicated library. KiCad's symbol library places power symbols in the `power` library, and users may create libraries to store their own power symbols. If the "Define as power symbol" box is checked in a symbol's properties, that symbol will appear in the Schematic Editor's "Add Power Port" dialog for convenient access.

Power symbols are handled and created the same way as normal symbols, but there are several additional considerations described below. They consist of a graphical symbol and a pin of the type "Power input" that is marked hidden.

Below is an example of a `GND` power symbol.



## Creating a Power Port Symbol

Power Port symbols consist of a pin of type "Power input" that is marked invisible. Invisible power input pins have a special property of automatically connecting to a net with the same name as the pin name. A net that is wired to an invisible power input pin will therefore be named after the pin, even if there are other net labels on the net. This connection is global.

### NOTE

If the power symbol has the "Define as power symbol" property checked, the power input pin does not need to be marked invisible. However, the convention is to make these pins invisible anyway.

Pin Properties

Pin name:

Pin number:

Electrical type:

Graphic style:

X position:  in

Y position:  in

Orientation:

Pin length:  in

Name text size:  in

Number text size:  in

☐ Common to all units in symbol

☐ Common to all body styles (De Morgan)

☐ Visible

Preview:

> Alternate pin definitions

Cancel OK

Pour créer un symbole d'alimentation, utilisez les étapes suivantes :

- Add a pin of type "Power input", with "Visible" unchecked, and the pin named according to the desired net. Make the pin number 1, the length 0, and set the graphic style to "Line". The pin name establishes the connection to the net; in this case the pin will automatically connect to the net GND. The pin number, length, and line style do not matter electrically.
- Place the pin on the symbol anchor.
- Use the shape tools to draw the symbol graphics.
- Set the symbol value. The symbol value does not matter electrically, but it is displayed in the schematic. To eliminate confusion, it should match the pin name (which determines the connected net name).
- Check the "Define as power symbol" box in Symbol Properties window. This makes the symbol appear in the "Add Power Port" dialog, makes the Value field read-only in the schematic, prevents the symbol from being assigned a footprint, and excludes the symbol from the board, BOM, and netlists.
- Set the symbol reference and uncheck the "Show" box. The reference text is not important except for the first character, which should be #. For the power port shown above, the reference could be #GND. Symbols with references that begin with # are not added to the PCB, are not included in Bill of Materials exports or netlists, and they cannot be assigned a footprint in the footprint assignment tool. If a power port's reference does not begin with #, the character will be inserted automatically when the annotation or footprint assignment tools are run.


An easier method to create a new power port symbol is to use another symbol as a starting point, [as described earlier](#).

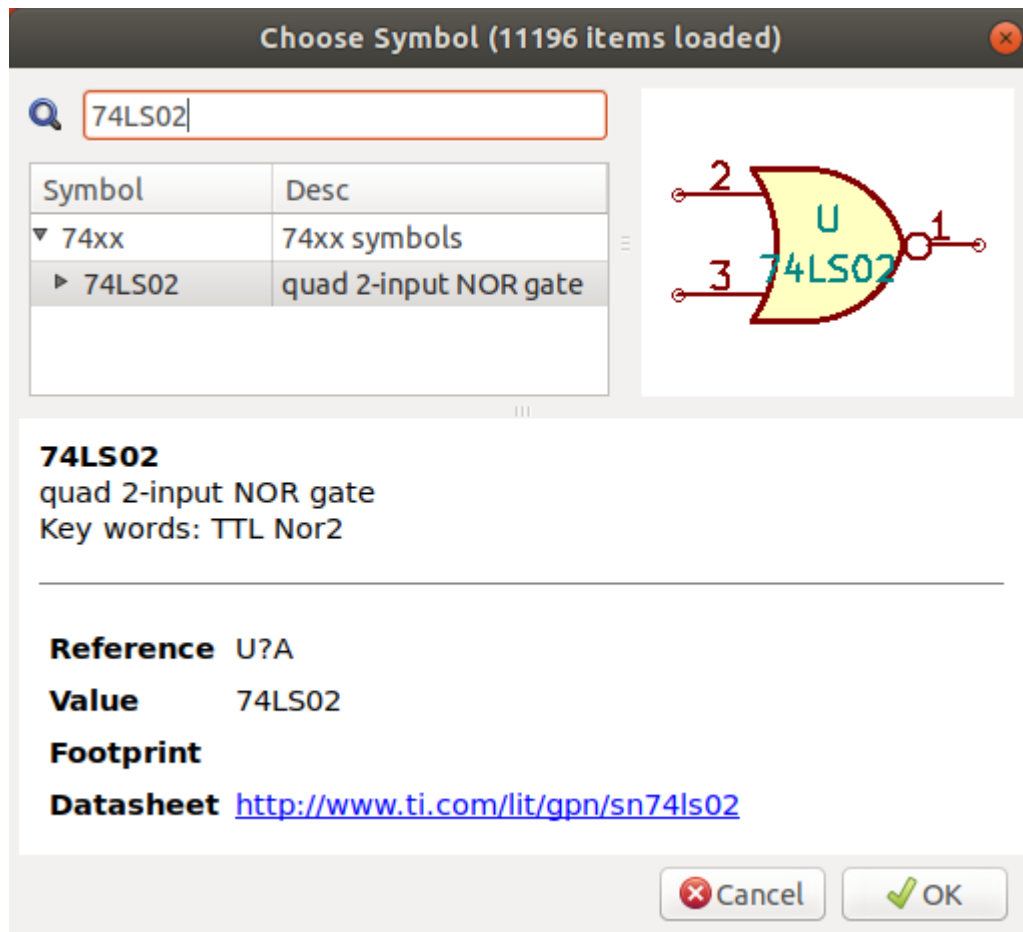
**NOTE**

When modifying an existing power port symbol, make sure to rename the pin name so that the new symbol connects to the appropriate power net.

# Symbol Library Browser

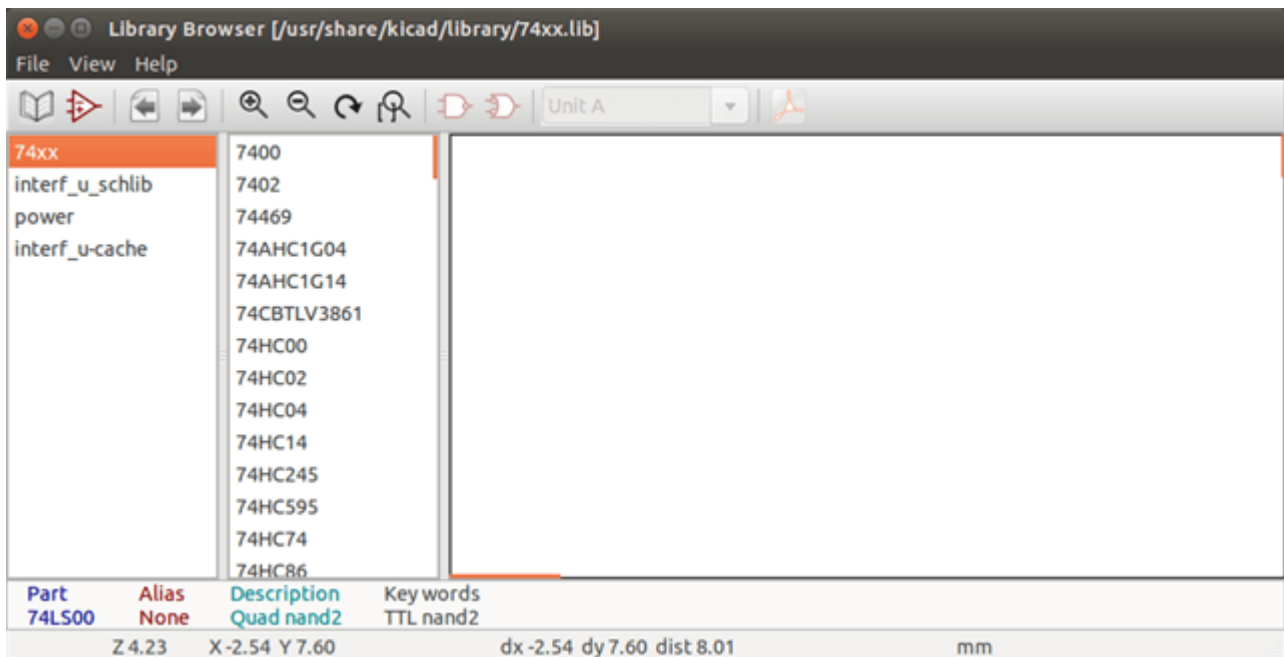
## Introduction

The Symbol Library Browser allows you to quickly examine the content of symbol libraries. The Symbol Library Viewer can be accessed by clicking  icon on the main toolbar, **View** → **Symbol Library Browser**..., or clicking **Select With Browser** in the "Choose Symbol" window.

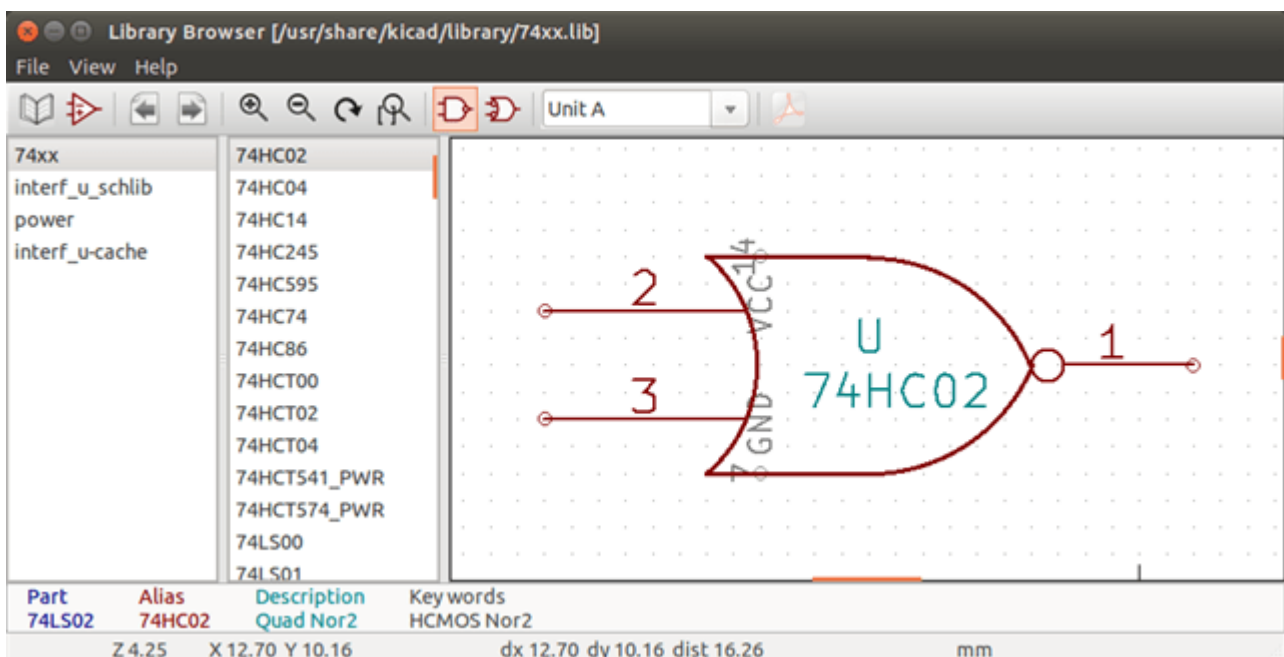




## Viewlib - fenêtre principale



To examine the contents of a library, select a library from the list on the left hand pane. All symbols in the selected library will appear in the second pane. Select a symbol name to view the symbol.





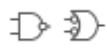
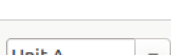




## Symbol Library Browser Top Toolbar

The top tool bar in Symbol Library Browser is shown below.



The available commands are:

	Selection of the symbol which can be also selected in the displayed list.
	Display previous symbol.
	Display next symbol.
	Zoom tools.
	Selection of the representation (normal or alternate) if an alternate representation exists.
	Selection of the unit for symbols that contain multiple units.
	If they exist, display the associated documents.
	Close the browser and place the selected symbol in the schematic.

# Création d'une Netliste

## Généralités

A netlist is a file which describes electrical connections between symbol pins. These connections are referred to as nets. Netlist files contain:

- A list of symbols and their pins.
- A list of connections (nets) between symbol pins.

Many different netlist formats exist. Sometimes the symbols list and the list of nets are two separate files. This netlist is fundamental in the use of schematic capture software, because the netlist is the link with other electronic CAD software, such as:

- PCB layout software.
- Schematic and electrical signal simulators.
- Programmable logic (FPGA, CPLD, etc.) compilers.

KiCad supports several netlist formats:

- KiCad format, which can be imported by the KiCad PCB Editor. However, the ["Update PCB from Schematic"](#) tool should be used instead of importing a KiCad netlist into the PCB editor.
- OrCAD PCB2 format, for designing PCBs with OrCAD.
- CADSTAR format, for designing PCBs with CADSTAR.
- Spice format, for use with various external circuit simulators.

### NOTE

In KiCad version 5.0 and later, it is not necessary to create a netlist for transferring a design from the schematic editor to the PCB editor. Instead, use the ["Update PCB from Schematic"](#) tool.

## Formats de Netliste

Netlists are exported with the Export Netlist dialog (**File** → **Export** → **Netlist...**).

Several netlist formats are available, and are selectable with the tabs at the top of the window. Some netlist formats have options.

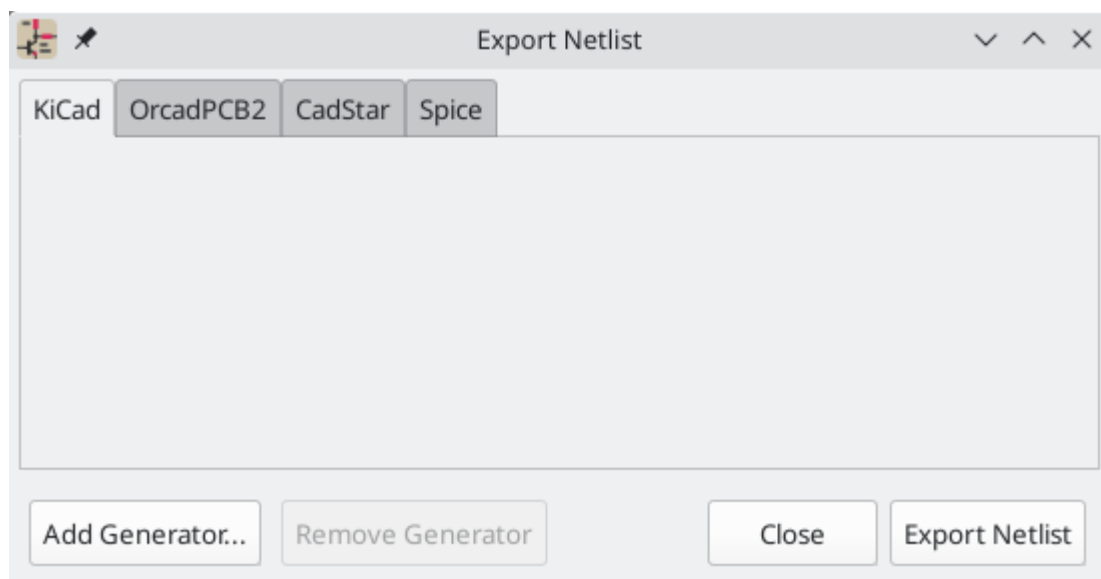
Clicking the **Export Netlist** button prompts for a netlist filename and saves the netlist.

### NOTE

Netlist generation can take up to several minutes for large schematics.

Custom generators can be added by clicking the **Add Generator...** button. Custom generators are external tools that are called by KiCad, for example Python scripts or XSLT stylesheets. For more information on custom netlist generators, see [the section on adding custom netlist generators](#).

## KiCad Netlist Format

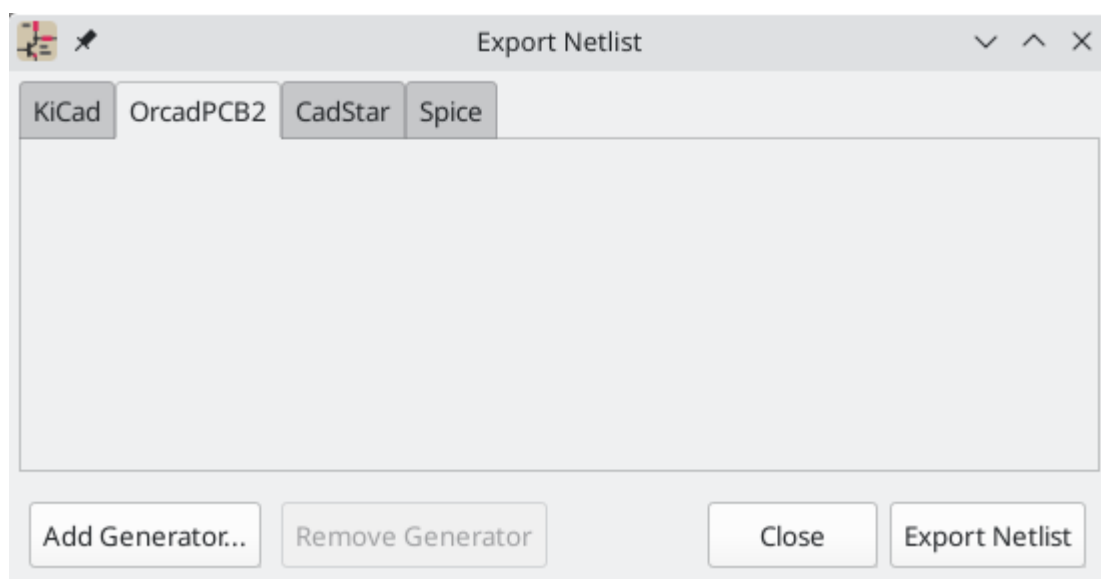


The KiCad netlist exporter does not have any options.

### NOTE

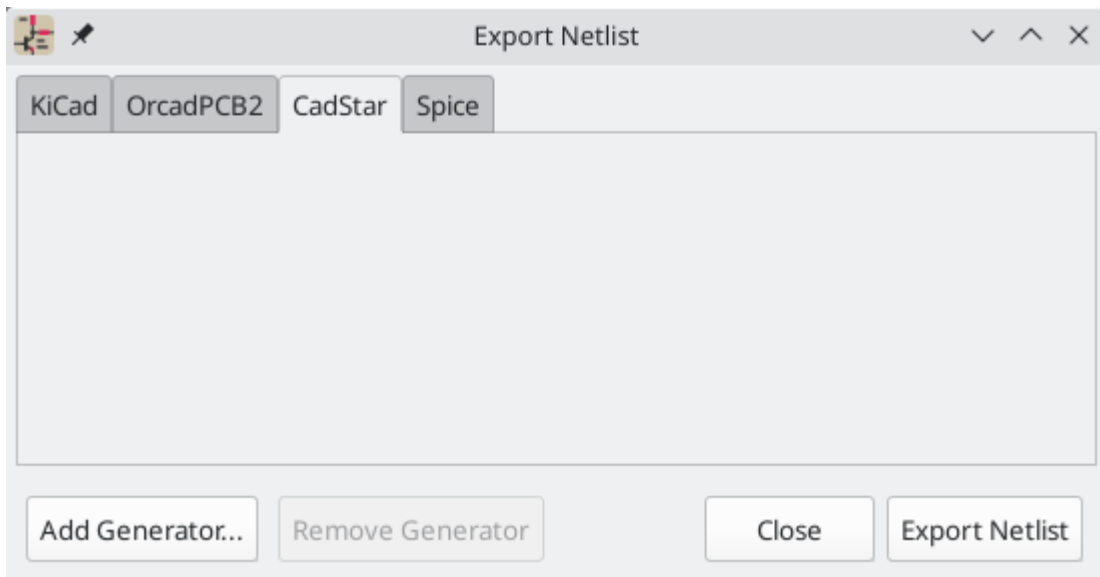
In KiCad version 5.0 and later, it is not necessary to create a netlist for transferring a design from the schematic editor to the PCB editor. Instead, use the ["Update PCB from Schematic"](#) tool.

## OrCAD PCB2 Netlist Format



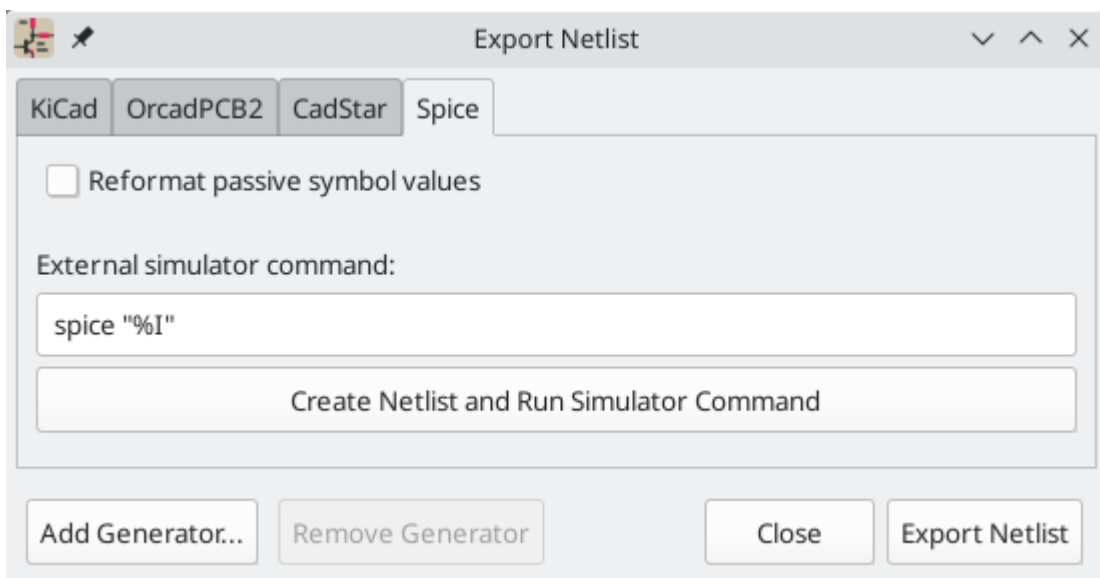
The OrCAD netlist exporter does not have any options.

## CADSTAR Netlist Format



The CADSTAR netlist exporter does not have any options.

## Spice Netlist Format



The Spice netlist format offers several options.

When the **Reformat passive symbol values** box is checked, passive symbol values will be adjusted to be compatible with Spice. Specifically:

- $\mu$  and M as unit prefixes are replaced with u and Meg, respectively
- Units are removed (e.g. 4.7k $\Omega$  is changed to 4.7k)
- Values in RKM format are rewritten to be Spice-compatible (e.g. 4u7 is changed to 4.7u)

The Spice netlist exporter also provides an easy way to simulate the generated netlist with an external simulator. This can be useful for running a simulation without using [KiCad's internal ngspice simulator](#), or for running an ngspice simulation with options that are not supported by KiCad's simulator tool.

Enter the path to the external simulator in the text box, with `%I` representing the generated netlist. Click the **Create Netlist and Run Simulator Command** button to generate the netlist and automatically run the simulator.

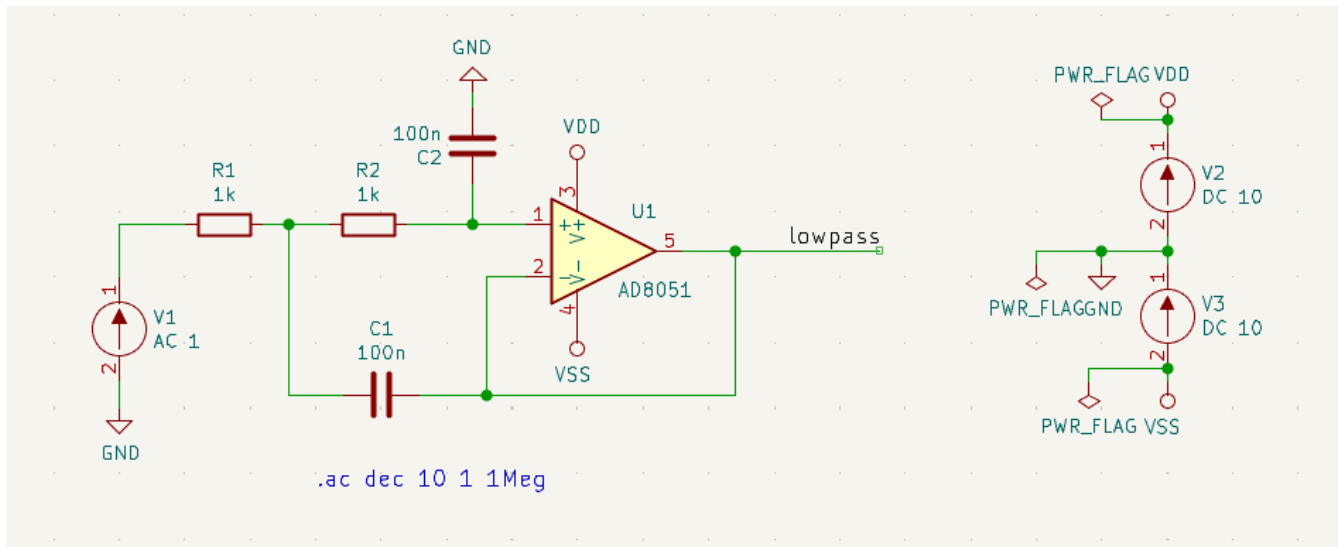
**NOTE**

The default simulator command ( `spice "%I"` ) must be adjusted to point to a simulator installed on your system.

For more information on the contents of Spice netlists, see the [Spice netlist section](#).

## Exemples de netlistes

Below is the schematic from the `sallen_key` project included in KiCad's simulation demos.



The KiCad format netlist for this schematic is as follows:

```

(export (version "E")
  (design
    (source "/usr/share/kicad/demos/simulation/sallen_key/sallen_key.kicad_sch")
    (date "Sun 01 May 2022 03:14:05 PM EDT")
    (tool "Eeschema (6.0.4)")
    (sheet (number "1") (name "/") (tstamps "/"))
    (title_block
      (title)
      (company)
      (rev)
      (date)
      (source "sallen_key.kicad_sch")
      (comment (number "1") (value ""))
      (comment (number "2") (value ""))
      (comment (number "3") (value ""))
      (comment (number "4") (value ""))
      (comment (number "5") (value ""))
      (comment (number "6") (value ""))
      (comment (number "7") (value ""))
      (comment (number "8") (value ""))
      (comment (number "9") (value ""))))
    (components
      (comp (ref "C1")
        (value "100n")
        (libsource (lib "sallen_key_schlib") (part "C") (description ""))
        (property (name "Sheetname") (value ""))
        (property (name "Sheetfile") (value "sallen_key.kicad_sch"))
        (sheetpath (names "/") (tstamps "/"))
        (tstamps "00000000-0000-0000-0000-00005789077d"))
      (comp (ref "C2")
        (value "100n")
        (fields
          (field (name "Fieldname") "Value")
          (field (name "SpiceMapping") "1 2")
          (field (name "Spice_Primitive") "C"))
        (libsource (lib "sallen_key_schlib") (part "C") (description ""))
        (property (name "Fieldname") (value "Value"))
        (property (name "Spice_Primitive") (value "C"))
        (property (name "SpiceMapping") (value "1 2"))
        (property (name "Sheetname") (value ""))
        (property (name "Sheetfile") (value "sallen_key.kicad_sch"))
        (sheetpath (names "/") (tstamps "/"))
        (tstamps "00000000-0000-0000-0000-00005789085b"))
      (comp (ref "R1")
        (value "1k")
        (fields
          (field (name "Fieldname") "Value")
          (field (name "SpiceMapping") "1 2")
          (field (name "Spice_Primitive") "R"))
        (libsource (lib "sallen_key_schlib") (part "R") (description ""))
        (property (name "Fieldname") (value "Value"))
        (property (name "SpiceMapping") (value "1 2"))
        (property (name "Spice_Primitive") (value "R"))
        (property (name "Sheetname") (value ""))
        (property (name "Sheetfile") (value "sallen_key.kicad_sch"))
        (sheetpath (names "/") (tstamps "/"))
        (tstamps "00000000-0000-0000-0000-0000578906ff"))
      (comp (ref "R2")
        (value "1k")
        (fields
          (field (name "Fieldname") "Value")
          (field (name "SpiceMapping") "1 2"))

```

In Spice format, the netlist is as follows:

```
.title KiCad schematic
.include "ad8051.lib"
XU1 Net-_C2-Pad1_ /lowpass VDD VSS /lowpass AD8051
C2 Net-_C2-Pad1_ GND 100n
C1 /lowpass Net-_C1-Pad2_ 100n
R2 Net-_C2-Pad1_ Net-_C1-Pad2_ 1k
R1 Net-_C1-Pad2_ Net-_R1-Pad2_ 1k
V1 Net-_R1-Pad2_ GND AC 1
V2 VDD GND DC 10
V3 GND VSS DC 10
.ac dec 10 1 1Meg
.end
```

## Notes sur les netlistes

### Précautions pour les noms de netlistes

Many software tools that use netlists do not accept spaces in component names, pins, nets, or other fields. Avoid using spaces in pins, labels, names, and value fields of components to ensure maximum compatibility.

In the same way, special characters other than letters and numbers can cause problems. Note that this limitation is not related to KiCad, but to the netlist formats that can then become untranslatable by other software that reads those netlist files.

### Spice netlists

Spice simulators expect simulation commands ( `.PROBE` , `.AC` , `.TRAN` , etc.) to be included in the netlist.

Any text line included in the schematic diagram starting with a period ( `.` ) will be included in the netlist. If a text object contains multiple lines, only the lines beginning with a period will be included.

`.include` directives for including model library files are automatically added to the netlist based on the Spice model settings for the symbols in the schematic.

## Autres formats

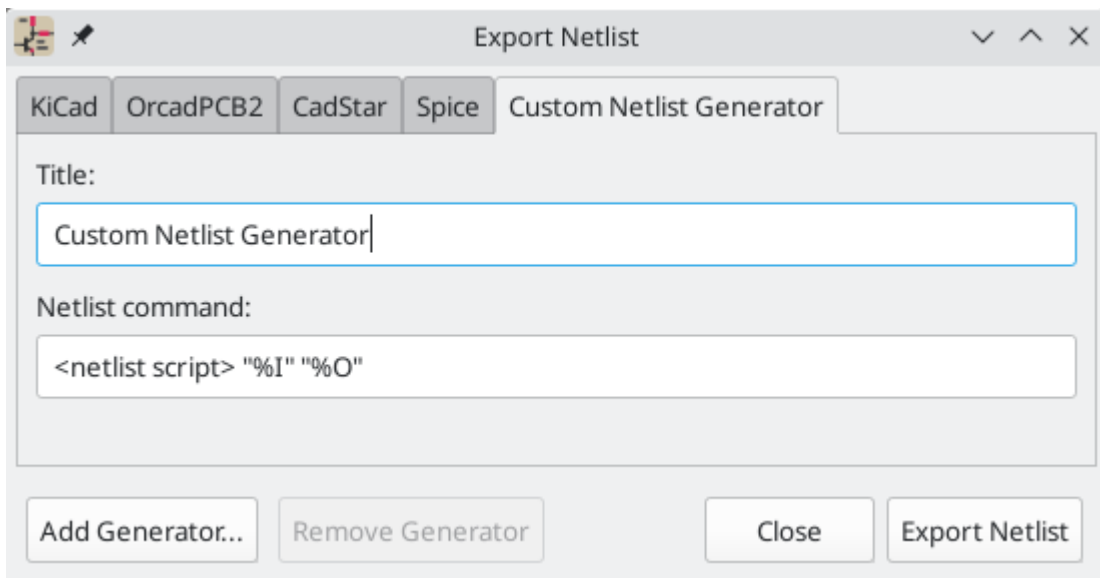
KiCad supports custom netlist generators for exporting netlists in other formats. Some examples of netlist generators are given in the [custom netlist generators section](#).

A netlist generator is a script or program that converts the intermediate netlist file created by KiCad into the desired netlist format. The intermediate netlist file contains all of the netlist information required to create an arbitrary netlist for the schematic. Python and XSLT are commonly used tools to create custom netlist generators.

### Adding new netlist generators

New netlist generators are added by clicking the **Add Generator...** button.





New generators require a name and a command. The name is shown in the tab label, and the command is run whenever the **Export Netlist** button is clicked.

When the netlist is generated, KiCad creates an intermediate XML file which contains all of the netlist information from the schematic. The generator command is then run in order to transform the intermediate netlist into the desired netlist format.

The netlist command must be set up properly so that the netlist generator script takes the intermediate netlist file as input and outputs the desired netlist file. The `%I` argument represents the input intermediate netlist filename and the `%O` argument represents the output netlist filename. The exact netlist command will depend on the generator script used.

## Format de la ligne de commande

Consider the following example which uses `xsltproc` to generate a netlist in PADS ASC format. `xsltproc` converts the intermediate netlist using the `netlist_form_pads-pcb.asc.xsl` stylesheet to define the output format:

```
xsltproc -o %O.net /usr/share/kicad/plugins/netlist_form_pads-pcb.asc.xsl %I
```

The purpose of each part of the command is as follows:

<code>xsltproc</code>	A tool to convert an XML file (the intermediate netlist) according to an XSLT stylesheet.
<code>-o %O.net</code>	Output filename. %O is replaced with the name of the intermediate netlist file, which is <code>&lt;schematic name&gt;.xml</code> . Therefore in this example the complete output filename is <code>&lt;schematic name&gt;.xml.net</code> . An arbitrary output filename can be specified if desired with <code>-o &lt;filename&gt;</code> .
<code>/usr/share/kicad/plugins/netlist_form_pads-pcb.asc.xsl</code>	XSLT stylesheet which determines how the output is formatted. This particular stylesheet is included with KiCad, but custom stylesheets can also be created.
<code>%I</code>	Input (intermediate netlist) filename. %I is replaced with the name of the intermediate netlist file, which is <code>&lt;schematic name&gt;.xml</code> .

For netlist generators that do not use `xsltproc`, the generator command will differ.

## Format du fichier intermédiaire de Netliste

See the [custom netlist generators section](#) for more information about netlist generators, a description of the intermediate netlist format, and some examples of netlist generators.

# Création de Netlistes et BOM personnalisés

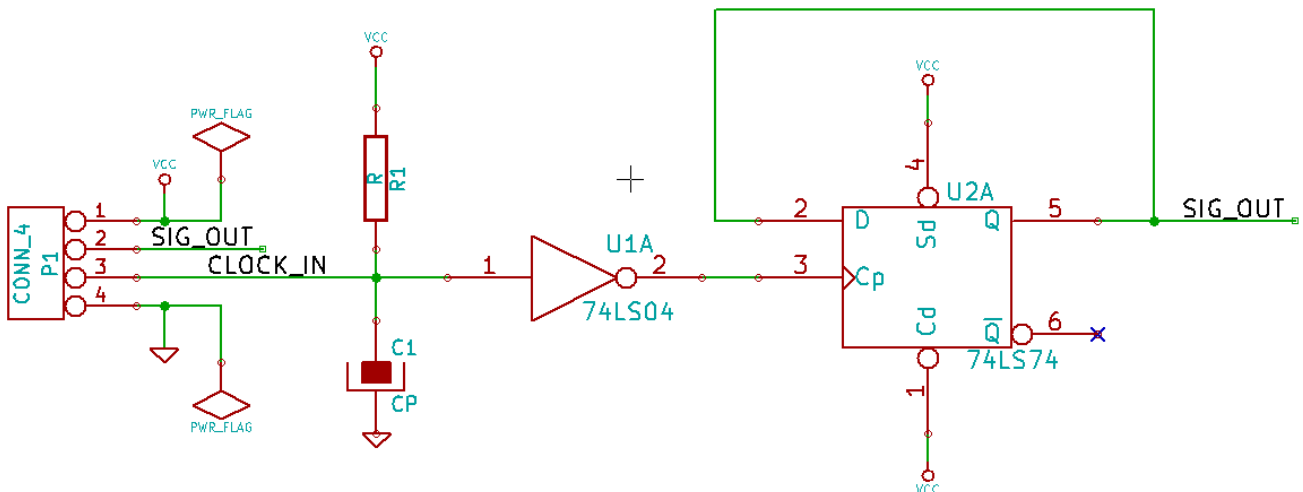
## Fichier intermédiaire de Netliste

BOM files and netlist files can be converted from an Intermediate netlist file created by KiCad.

Ce fichier utilise une syntaxe XML, et est appelé "netliste intermédiaire". Cette netliste intermédiaire inclue une grande quantité de données relatives au circuit, et, pour cette raison, il peut être utilisé par un post-traitement pour créer une liste de composants ou d'autres rapports.

Suivant le fichier de sortie (BOM ou netliste), différentes portions de la netliste intermédiaire seront utilisées dans le post-traitement.

## Exemple de schéma



## Exemple de fichier netliste intermédiaire

Le fichier netliste intermédiaire (utilisant une syntaxe XML) correspondant au schéma ci-dessus :

```

<?xml version="1.0" encoding="utf-8"?>
<export version="D">
  <design>
    <source>F:\kicad_aux\netlist_test\netlist_test.sch</source>
    <date>29/08/2010 20:35:21</date>
    <tool>eeschema (2010-08-28 BZR 2458)-unstable</tool>
  </design>
  <components>
    <comp ref="P1">
      <value>CONN_4</value>
      <libsource lib="conn" part="CONN_4"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E2141</tstamp>
    </comp>
    <comp ref="U2">
      <value>74LS74</value>
      <libsource lib="74xx" part="74LS74"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E20BA</tstamp>
    </comp>
    <comp ref="U1">
      <value>74LS04</value>
      <libsource lib="74xx" part="74LS04"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E20A6</tstamp>
    </comp>
    <comp ref="C1">
      <value>CP</value>
      <libsource lib="device" part="CP"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E2094</tstamp>
    </comp>
    <comp ref="R1">
      <value>R</value>
      <libsource lib="device" part="R"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E208A</tstamp>
    </comp>
  </components>
  <libparts>
    <libpart lib="device" part="C">
      <description>Condensateur non polarise</description>
      <footprints>
        <fp>SM*</fp>
        <fp>C?</fp>
        <fp>C1-1</fp>
      </footprints>
      <fields>
        <field name="Reference">C</field>
        <field name="Value">C</field>
      </fields>
      <pins>
        <pin num="1" name="~" type="passive"/>
        <pin num="2" name="~" type="passive"/>
      </pins>
    </libpart>
    <libpart lib="device" part="R">
      <description>Resistance</description>
      <footprints>
        <fp>R?</fp>
        <fp>SM0603</fp>
        <fp>SM0805</fp>
      </footprints>
    </libpart>
  </libparts>
</export>

```

## Conversion dans un nouveau format de netliste

En appliquant un filtre de post-traitement au fichier netliste Intermédiaire, vous pouvez générer des formats inconnus de netliste, ou de BOM. Parce que cette conversion est une transformation de texte en texte, ce filtre de post-traitement pourra être écrit en Python, XSLT, ou tout autre outil capable de prendre du XML en entrée.

XSLT itself is an XML language very suitable for XML transformations. There is a free program called *xsltproc* that you can download and install. The *xsltproc* program can be used to read the Intermediate XML netlist input file, apply a style-sheet to transform the input, and save the results in an output file. Use of *xsltproc* requires a style-sheet file using XSLT conventions. The full conversion process is handled by KiCad, after it is configured once to run *xsltproc* in a specific way.

## L'approche XSLT

Vous trouverez la documentation qui décrit les transformations XSL (XSLT) ici :

<http://www.w3.org/TR/xslt>

## Créer un fichier Netliste Pads-Pcb

Le format pads-pcb contient deux sections.

- La liste des empreintes.
- La Netliste : qui regroupe les références des broches par équipotentiels.

Ci-dessous, une feuille de style qui convertit le fichier netliste intermédiaire au format de netliste pads-pcb

```

<?xml version="1.0" encoding="ISO-8859-1"?>
<!--XSL style sheet to Eeschema Generic Netlist Format to PADS netlist format
Copyright (C) 2010, SoftPLC Corporation.
GPL v2.

How to use:
https://lists.launchpad.net/kicad-developers/msg05157.html
-->

<!DOCTYPE xsl:stylesheet [
  <!ENTITY nl "&#xd;&#xa;"> <!--new line CR, LF -->
]>

<xsl:stylesheet version="1.0" xmlns:xsl="http://www.w3.org/1999/XSL/Transform">
<xsl:output method="text" omit-xml-declaration="yes" indent="no"/>

<xsl:template match="/export">
  <xsl:text>*PADS-PCB*&nl;*PART*&nl;</xsl:text>
  <xsl:apply-templates select="components/comp"/>
  <xsl:text>&nl;*NET*&nl;</xsl:text>
  <xsl:apply-templates select="nets/net"/>
  <xsl:text>*END*&nl;</xsl:text>
</xsl:template>

<!-- for each component -->
<xsl:template match="comp">
  <xsl:text> </xsl:text>
  <xsl:value-of select="@ref"/>
  <xsl:text> </xsl:text>
  <xsl:choose>
    <xsl:when test = "footprint != '' ">
      <xsl:apply-templates select="footprint"/>
    </xsl:when>
    <xsl:otherwise>
      <xsl:text>unknown</xsl:text>
    </xsl:otherwise>
  </xsl:choose>
  <xsl:text>&nl;</xsl:text>
</xsl:template>

<!-- for each net -->
<xsl:template match="net">
  <!-- nets are output only if there is more than one pin in net -->
  <xsl:if test="count(node)>1">
    <xsl:text>*SIGNAL* </xsl:text>
    <xsl:choose>
      <xsl:when test = "@name != '' ">
        <xsl:value-of select="@name"/>
      </xsl:when>
      <xsl:otherwise>
        <xsl:text>N-</xsl:text>
        <xsl:value-of select="@code"/>
      </xsl:otherwise>
    </xsl:choose>
    <xsl:text>&nl;</xsl:text>
    <xsl:apply-templates select="node"/>
  </xsl:if>
</xsl:template>

<!-- for each node -->
<xsl:template match="node">
  <xsl:text> </xsl:text>

```

Voici le fichier de sortie pads-pcb après traitement par xsltproc :

```
*PADS-PCB*
*PART*
P1 unknown
U2 unknown
U1 unknown
C1 unknown
R1 unknown
*NET*
*SIGNAL* GND
U1.7
C1.2
U2.7
P1.4
*SIGNAL* VCC
R1.1
U1.14
U2.4
U2.1
U2.14
P1.1
*SIGNAL* N-4
U1.2
U2.3
*SIGNAL* /SIG_OUT
P1.2
U2.5
U2.2
*SIGNAL* /CLOCK_IN
R1.2
C1.1
U1.1
P1.3

*END*
```

La ligne de commande utilisée pour effectuer cette conversion :

```
kicad\\bin\\xsltproc.exe -o test.net kicad\\bin\\plugins\\netlist_form_pads-pcb.xml
test.tmp
```

## Créer un fichier de netliste Cadstar

Le format Cadstar contient deux sections.

- La liste des empreintes.
- La Netliste : qui regroupe les références des broches par équipotentiels.

Ci-dessous, la feuille de style pour effectuer cette conversion spécifique :

```

<?xml version="1.0" encoding="ISO-8859-1"?>
<!--XSL style sheet to Eeschema Generic Netlist Format to CADSTAR netlist format
      Copyright (C) 2010, Jean-Pierre Charras.
      Copyright (C) 2010, SoftPLC Corporation.
      GPL v2.

<!DOCTYPE xsl:stylesheet [
      <!ENTITY nl  "&#xd;&#xa;"> <!--new line CR, LF -->
]>

<xsl:stylesheet version="1.0" xmlns:xsl="http://www.w3.org/1999/XSL/Transform">
<xsl:output method="text" omit-xml-declaration="yes" indent="no"/>

<!-- Netlist header -->
<xsl:template match="/export">
      <xsl:text>.HEA&nl;</xsl:text>
      <xsl:apply-templates select="design/date"/> <!-- Generate line .TIM <time> -->
      <xsl:apply-templates select="design/tool"/> <!-- Generate line .APP <eeschema version>
-->
      <xsl:apply-templates select="components/comp"/> <!-- Generate list of components -->
      <xsl:text>&nl;&nl;</xsl:text>
      <xsl:apply-templates select="nets/net"/> <!-- Generate list of nets and
connections -->
      <xsl:text>&nl;.END&nl;</xsl:text>
</xsl:template>

      <!-- Generate line .TIM 20/08/2010 10:45:33 -->
<xsl:template match="tool">
      <xsl:text>.APP "</xsl:text>
      <xsl:apply-templates/>
      <xsl:text>"&nl;</xsl:text>
</xsl:template>

      <!-- Generate line .APP "eeschema (2010-08-17 BZR 2450)-unstable" -->
<xsl:template match="date">
      <xsl:text>.TIM </xsl:text>
      <xsl:apply-templates/>
      <xsl:text>&nl;</xsl:text>
</xsl:template>

<!-- for each component -->
<xsl:template match="comp">
      <xsl:text>.ADD_COM </xsl:text>
      <xsl:value-of select="@ref"/>
      <xsl:text> </xsl:text>
      <xsl:choose>
        <xsl:when test = "value != '' ">
          <xsl:text>"</xsl:text> <xsl:apply-templates select="value"/> <xsl:text>"
</xsl:text>
        </xsl:when>
        <xsl:otherwise>
          <xsl:text>"</xsl:text>
        </xsl:otherwise>
      </xsl:choose>
      <xsl:text>&nl;</xsl:text>
</xsl:template>

<!-- for each net -->
<xsl:template match="net">
      <!-- nets are output only if there is more than one pin in net -->
      <xsl:if test="count(node)>1">
        <xsl:variable name="netname">

```



Le fichier de sortie au format Cadstar :

```
.HEA
.TIM 21/08/2010 08:12:08
.APP "eeschema (2010-08-09 BZR 2439)-unstable"
.ADD_COM P1 "CONN_4"
.ADD_COM U2 "74LS74"
.ADD_COM U1 "74LS04"
.ADD_COM C1 "CP"
.ADD_COM R1 "R"

.ADD_TER U1.7 "GND"
.TER      C1.2
          U2.7
          P1.4
.ADD_TER R1.1 "VCC"
.TER      U1.14
          U2.4
          U2.1
          U2.14
          P1.1
.ADD_TER U1.2 "N-4"
.TER      U2.3
.ADD_TER P1.2 "/SIG_OUT"
.TER      U2.5
          U2.2
.ADD_TER R1.2 "/CLOCK_IN"
.TER      C1.1
          U1.1
          P1.3

.END
```

## Create an OrcadPCB2 netlist file

Ce format a une seule section, qui est la liste des empreintes. Chaque empreinte inclue sa liste de broches avec leurs références d'équipotentiels.

Ci-dessous, la feuille de style pour cette conversion spécifique :

```

<?xml version="1.0" encoding="ISO-8859-1"?>
<!--XSL style sheet to Eeschema Generic Netlist Format to CADSTAR netlist format
      Copyright (C) 2010, SoftPLC Corporation.
      GPL v2.

      How to use:
      https://lists.launchpad.net/kicad-developers/msg05157.html
-->

<!DOCTYPE xsl:stylesheet [
  <!ENTITY nl "&#xd;&#xa;"> <!--new line CR, LF -->
]>

<xsl:stylesheet version="1.0" xmlns:xsl="http://www.w3.org/1999/XSL/Transform">
<xsl:output method="text" omit-xml-declaration="yes" indent="no"/>

<!--
      Netlist header
      Creates the entire netlist
      (can be seen as equivalent to main function in C
-->
<xsl:template match="/export">
  <xsl:text>( { Eeschema Netlist Version 1.1  </xsl:text>
  <!-- Generate line .TIM <time> -->
<xsl:apply-templates select="design/date"/>
<!-- Generate line eeschema version ... -->
<xsl:apply-templates select="design/tool"/>
<xsl:text>}&nl;</xsl:text>

<!-- Generate the list of components -->
<xsl:apply-templates select="components/comp"/> <!-- Generate list of components -->

<!-- end of file -->
<xsl:text>)&nl;*&nl;</xsl:text>
</xsl:template>

<!--
      Generate id in header like "eeschema (2010-08-17 BZR 2450)-unstable"
-->
<xsl:template match="tool">
  <xsl:apply-templates/>
</xsl:template>

<!--
      Generate date in header like "20/08/2010 10:45:33"
-->
<xsl:template match="date">
  <xsl:apply-templates/>
  <xsl:text>&nl;</xsl:text>
</xsl:template>

<!--
      This template read each component
      (path = /export/components/comp)
      creates lines:
      ( 3EBF7DBD $noname U1 74LS125
        ... pin list ...
      )
      and calls "create_pin_list" template to build the pin list
-->
<xsl:template match="comp">
  <xsl:text> ( </xsl:text>

```

Le fichier de sortie au format OrcadPCB2 :

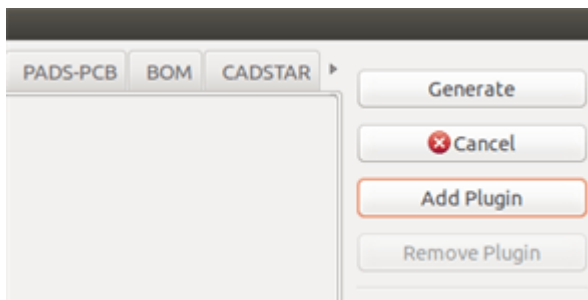
```
( { Eeschema Netlist Version 1.1 29/08/2010 21:07:51
eeschema (2010-08-28 BZR 2458)-unstable}
( 4C6E2141 $noname P1 CONN_4
( 1 VCC )
( 2 /SIG_OUT )
( 3 /CLOCK_IN )
( 4 GND )
)
( 4C6E20BA $noname U2 74LS74
( 1 VCC )
( 2 /SIG_OUT )
( 3 N-04 )
( 4 VCC )
( 5 /SIG_OUT )
( 6 ? )
( 7 GND )
( 14 VCC )
)
( 4C6E20A6 $noname U1 74LS04
( 1 /CLOCK_IN )
( 2 N-04 )
( 7 GND )
( 14 VCC )
)
( 4C6E2094 $noname C1 CP
( 1 /CLOCK_IN )
( 2 GND )
)
( 4C6E208A $noname R1 R
( 1 VCC )
( 2 /CLOCK_IN )
)
)
*
```

## Netlist plugins interface

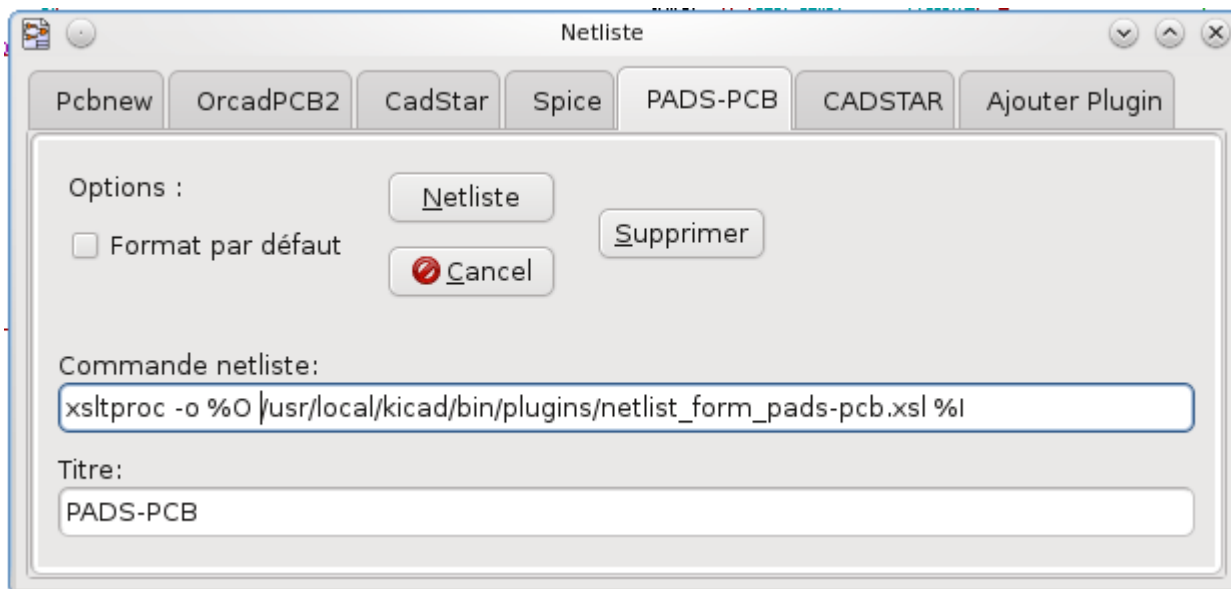
Intermediate Netlist converters can be automatically launched within the Schematic Editor.

## Ouvrez la fenêtre de configuration

Vous pouvez ajouter un nouveau plugin par le bouton "Ajouter Plugin".



Voici l'onglet de configuration du plugin pour Pads-Pcb :



## Configuration des paramètres du plugin

The netlist plug-in configuration dialog requires the following information:

- Un titre : pour l'onglet, comme le nom du format de Netliste.
- La ligne de commande pour lancer la conversion.

Quand vous cliquez sur le bouton netliste :

1. KiCad creates an intermediate netlist file \*.xml, for instance test.xml.
2. KiCad runs the plug-in by reading test.xml and creates test.net.

## Génération de fichiers netlistes en ligne de commande

Partant du fait que nous utilisons le programme *xsltproc.exe* pour appliquer la feuille de style au fichier intermédiaire, *xsltproc.exe* sera exécuté avec la commande suivante :

*xsltproc.exe -o <fichier de sortie> <fichier feuille de style> <fichier XML d'entrée à convertir>*

Sous Windows, la ligne de commande sera la suivante :

*f:/kicad/bin/xsltproc.exe -o "%O" f:/kicad/bin/plugins/netlist\_form\_pads-pcb.xml "%I"*

Sous Linux, la ligne de commande sera la suivante :

*xsltproc -o "%O" /usr/local/kicad/bin/plugins/netlist\_form\_pads-pcb.xml "%I"*

Where *netlist\_form\_pads-pcb.xml* is the style-sheet that you are applying. Do not forget the double quotes around the file names, this allows them to have spaces after the substitution by KiCad.

Le format de la ligne de commande accepte des paramètres de substitution pour les noms de fichiers :

Les paramètres autorisés sont.

- %B ⇒ nom et chemin du fichier de sortie, sans le point et l'extension.
- %I ⇒ nom et chemin complets du fichier d'entrée (le fichier intermédiaire de netliste).

%O = nom et chemin complets du fichier de sortie.

%I sera remplacé par le nom de fichier intermédiaire de netliste.

%O sera remplacé par le nom de fichier de sortie.

## Format de ligne de commande : exemple pour xsltproc

Le format de ligne de commande de xsltproc est le suivant :

<chemin vers xsltproc> xsltproc <paramètres de xsltproc >

Sous Windows

**f:/kicad/bin/xsltproc.exe -o "%O" f:/kicad/bin/plugins/netlist\_form\_pads-pcb.xml "%I"**

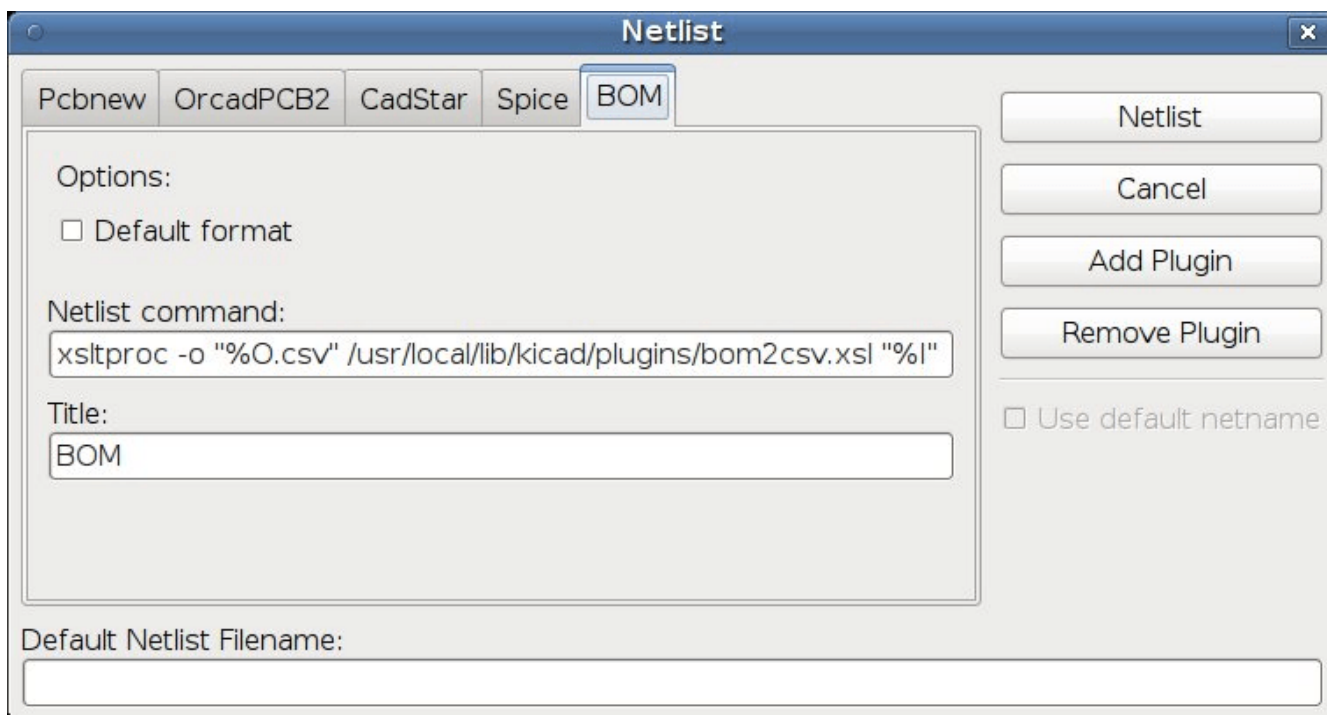
Sous Linux

**xsltproc -o "%O" /usr/local/kicad/bin/plugins/netlist\_form\_pads-pcb.xml "%I"**

Les exemples ci-dessus supposent que xsltproc est installé sur votre PC sous Windows et que tous les fichiers sont situés dans F:\kicad\bin.

## Génération de listes de composants (BOM)

Puisque le fichier netliste intermédiaire contient toutes les informations sur les composants utilisés, une liste de composants peut en être extraite. Voici la fenêtre de configuration du plugin (sous Linux) permettant de créer un fichier de BOM (Bill Of Materials) personnalisé :



Le chemin vers la feuille de style bom2csv.xml dépend de votre système. Actuellement, la meilleure feuille de style XSLT pour la génération du BOM est nommée *bom2csv.xml*. Vous êtes libre de la modifier en fonction de vos besoins, et si vous développez un autre modèle utile à tous, vous pouvez demander qu'il fasse partie du projet KiCad.

## Exemples de lignes de commandes pour les scripts Python

Le format d'une ligne de commande pour python ressemble à ceci :

python <fichier script> <fichier d'entrée> <fichier de sortie>

Sous Windows

```
python *.exe f:/kicad/python/my_python_script.py "%I" "%O"
```

Sous Linux

```
python /usr/local/kicad/python/my_python_script.py "%I" "%O"
```

Partant du fait que Python est effectivement installé sur votre PC..

## Structure du fichier de netliste intermédiaire

L'exemple qui suit donne une idée du format du fichier de netliste intermédiaire.

```

<?xml version="1.0" encoding="utf-8"?>
<export version="D">
  <design>
    <source>F:\kicad_aux\netlist_test\netlist_test.sch</source>
    <date>29/08/2010 21:07:51</date>
    <tool>eeschema (2010-08-28 BZR 2458)-unstable</tool>
  </design>
  <components>
    <comp ref="P1">
      <value>CONN_4</value>
      <libsource lib="conn" part="CONN_4"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E2141</tstamp>
    </comp>
    <comp ref="U2">
      <value>74LS74</value>
      <libsource lib="74xx" part="74LS74"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E20BA</tstamp>
    </comp>
    <comp ref="U1">
      <value>74LS04</value>
      <libsource lib="74xx" part="74LS04"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E20A6</tstamp>
    </comp>
    <comp ref="C1">
      <value>CP</value>
      <libsource lib="device" part="CP"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E2094</tstamp>
    <comp ref="R1">
      <value>R</value>
      <libsource lib="device" part="R"/>
      <sheetpath names="/" tstamps="/" />
      <tstamp>4C6E208A</tstamp>
    </comp>
  </components>
  <libparts/>
  <libraries/>
  <nets>
    <net code="1" name="GND">
      <node ref="U1" pin="7"/>
      <node ref="C1" pin="2"/>
      <node ref="U2" pin="7"/>
      <node ref="P1" pin="4"/>
    </net>
    <net code="2" name="VCC">
      <node ref="R1" pin="1"/>
      <node ref="U1" pin="14"/>
      <node ref="U2" pin="4"/>
      <node ref="U2" pin="1"/>
      <node ref="U2" pin="14"/>
      <node ref="P1" pin="1"/>
    </net>
    <net code="3" name="">
      <node ref="U2" pin="6"/>
    </net>
    <net code="4" name="">
      <node ref="U1" pin="2"/>
      <node ref="U2" pin="3"/>
    </net>
  </nets>

```

## Structure générale

Le fichier de netliste intermédiaire contient cinq sections.

- La section Entête.
- The components section.
- La section Composants en librairie.
- La section Librairies.
- La section Équipotentiellles

Le contenu du fichier a pour balises de délimitations <export>

```
<export version="D">
...
</export>
```

## Section Entête (Header)

L'entête a pour balises de délimitations <design>

```
<design>
<source>F:\kicad_aux\netlist_test\netlist_test.sch</source>
<date>21/08/2010 08:12:08</date>
<tool>eeschema (2010-08-09 BZR 2439)-unstable</tool>
</design>
```

Cette section peut être considérée comme une section de commentaires.

## Section Composants

La section composants a pour balises de délimitations <components>

```
<components>
<comp ref="P1">
<value>CONN_4</value>
<libsource lib="conn" part="CONN_4"/>
<sheetpath names="/" tstamps="/">
<tstamp>4C6E2141</tstamp>
</comp>
</components>
```

Cette section contient la liste des composants de votre schéma. Chaque composant est décrit comme ceci :



```

<comp ref="P1">
<value>CONN_4</value>
<libsource lib="conn" part="CONN_4"/>
<sheetpath names="/" tstamps="/" />
<tstamp>4C6E2141</tstamp>
</comp>

```

libsource	name of the lib where this component was found.
part	component name inside this library.
sheetpath	path of the sheet inside the hierarchy: identify the sheet within the full schematic hierarchy.
tstamps (time stamps)	time stamp of the schematic file.
tstamp (time stamp)	time stamp of the component.

## Note à propos de l'horodatage des composants

Pour identifier un composant dans une netliste, et par voie de conséquence sur le circuit, l'horodatage est utilisé comme référence unique pour chaque composant. Toutefois, Kicad fournit un autre moyen pour identifier un composant, qui est son empreinte correspondante sur le circuit. Ceci permet la ré-annotation de composants dans un projet de schéma sans perdre le lien entre le composant et son empreinte.

Un horodatage (timestamp) est un identifiant unique pour chaque composant, ou chaque feuille d'un projet schématique. Cependant, dans des hiérarchies complexes, la même feuille étant utilisée plus d'une fois, cette feuille contiendra des composants avec le même horodatage.

Une feuille donnée à l'intérieur d'une hiérarchie complexe dispose d'un identifiant unique : son chemin de feuille (sheetpath). Un composant donné (à l'intérieur d'une hiérarchie complexe) a donc un identifiant unique : le sheetpath + son timestamp.

## Section Composants en librairie (libparts)

La section libparts a pour délimiteur <libparts>, et le contenu de cette section est celui défini dans les librairies schématiques. La section libparts contient :

- The allowed footprints names (names use wildcards) delimiter <fp>.
- Les champs définis en librairie, avec pour délimiteur <fields>.
- La liste des pins, avec pour délimiteur <pins>.

```

<libparts>
<libpart lib="device" part="CP">
  <description>Condensateur polarise</description>
  <footprints>
    <fp>CP*</fp>
    <fp>SM*</fp>
  </footprints>
  <fields>
    <field name="Reference">C</field>
    <field name="Valeur">CP</field>
  </fields>
  <pins>
    <pin num="1" name="1" type="passive"/>
    <pin num="2" name="2" type="passive"/>
  </pins>
</libpart>
</libparts>

```

Les lignes `<pin num="1" type="passive"/>` donnent aussi le type électrique de la pin. Les types électriques possibles sont :

Input	Entrée
Output	Sortie
Bidirectional	Entrée ou Sortie
Tri-state	Trois-états
Passive	Extrémités de composants passifs
Unspecified	Non-Spécifié
Power input	Entrée d'alimentation d'un composant
Power output	Sortie d'alimentation, comme celle des régulateurs
Open collector	Collecteur ouvert
Open emitter	Émetteur ouvert
Not connected	Non-connecté, sera laissé en l'air dans le schéma

## Section Librairies

La section librairies a pour délimiteur `<libraries>`. Cette section contient la liste des librairies utilisées dans le projet.

```

<libraries>
  <library logical="device">
    <uri>F:\kicad\share\library\device.lib</uri>
  </library>
  <library logical="conn">
    <uri>F:\kicad\share\library\conn.lib</uri>
  </library>
</libraries>

```

## Section Équipotentielles (nets)

La section nets a pour délimiteur <nets>. Cette section contient la liste des équipotentiels, la "connectivité" du schéma.

```

<nets>
  <net code="1" name="GND">
    <node ref="U1" pin="7"/>
    <node ref="C1" pin="2"/>
    <node ref="U2" pin="7"/>
    <node ref="P1" pin="4"/>
  </net>
  <net code="2" name="VCC">
    <node ref="R1" pin="1"/>
    <node ref="U1" pin="14"/>
    <node ref="U2" pin="4"/>
    <node ref="U2" pin="1"/>
    <node ref="U2" pin="14"/>
    <node ref="P1" pin="1"/>
  </net>
</nets>

```

Cette section recense toutes les équipotentiels du schéma.

Une entrée net peut contenir :

```

<net code="1" name="GND">
  <node ref="U1" pin="7"/>
  <node ref="C1" pin="2"/>
  <node ref="U2" pin="7"/>
  <node ref="P1" pin="4"/>
</net>

```

net code	Identifiant interne pour ce net
name	Nom de ce net
node	Référence une pin de composant connectée à ce net

## Complément sur xsltproc

Réfère à la page : <http://xmlsoft.org/XSLT/xsltproc.html>

## Introduction

xsltproc est un outil en ligne de commande pour appliquer des feuilles de styles XSLT à des documents XML. Bien qu'il ait été développé au sein du projet GNOME, il peut opérer indépendamment du bureau GNOME.

xsltproc est invoqué à partir de la ligne de commande, avec le nom de la feuille de style à utiliser, suivi du nom du ou des fichiers auxquels la feuille de style doit être appliquée. Il utilisera l'entrée standard si le nom de fichier d'entrée fournit est -.

Si une feuille de style est incluse dans un document XML, au moyen d'une instruction de traitement de feuille de style, il n'est pas nécessaire de spécifier la feuille de style sur la ligne de commande. xsltproc détectera automatiquement la feuille de style incluse et l'utilisera. Par défaut, la sortie est la sortie standard. Vous pouvez préciser un fichier de sortie en utilisant l'option -o.

## Synoptique

```
xsltproc [[-V] | [-v] | [-o *file* ] | [--timing] | [--repeat] |
[--debug] | [--novalid] | [--noout] | [--maxdepth *val* ] | [--html] |
[--param *name* *value* ] | [--stringparam *name* *value* ] | [--nonet] |
[--path *paths* ] | [--load-trace] | [--catalogs] | [--xinclude] |
[--profile] | [--dumpextensions] | [--nowrite] | [--nomkdir] |
[--writesubtree] | [--noddatttr]] [ *stylesheet* ] [ *file1* ] [ *file2* ]
[ *....* ]
```

## Options de la ligne de commande

-V ou --version

Affiche les versions de libxml et libxslt qui sont utilisées.

-v ou --verbose

Affiche chaque étape de xsltproc lors du traitement du la feuille de style et du document.

-o ou --output *fichier*

Redirige la sortie vers le fichier nommé *fichier*. Pour des sorties multiples, que l'on appelle également ``chunking'', -o répertoire/ redirige les fichiers de sortie vers un répertoire donné. Le répertoire doit déjà exister.

--timing

Affiche le temps qu'il a fallu pour traiter la feuille de style, traiter le document, appliquer la feuille de style et enregistrer le résultat. Il est affiché en millisecondes.

--repeat

Lance la transformation 20 fois de suite. Utile pour des tests de vitesse.

--debug

Affiche un arbre XML du document transformé afin de déboguer.

--novalid

Évite le chargement de la DTD du document.

*--noout*

N'affiche pas le résultat.

*--maxdepth valeur*

Ajuste la profondeur maximale de la pile avant que libxslt ne conclue qu'il y ait une boucle infinie. La valeur par défaut est 500.

*--html*

Le document en entrée est un fichier HTML.

*--param nom valeur*

Passe un paramètre de nom *nom* et de valeur *valeur* à la feuille de style. Vous pouvez passer plusieurs paires nom/valeur, jusqu'à 32 valeurs. Si la valeur qui est spécifiée est une chaîne de caractères au lieu du nom d'identification d'un noeud, vous devez utiliser *--stringparam* à la place.

*--stringparam nom valeur*

Passe un paramètre de nom *nom* et de valeur *valeur* où *valeur* est une chaîne de caractères plutôt qu'un identifiant de noeud. (Note : La chaîne doit être en utf-8.)

*--nonet*

Ne pas utiliser Internet pour récupérer les DTD ou les entités.

*--path chemins*

Use the list (separated by space or column) of filesystem paths specified by *paths* to load DTDs, entities or documents.

*--load-trace*

Affiche sur la sortie d'erreurs standard (stderr) tous les documents chargés pendant le traitement.

*--catalogs*

Utilise les catalogues SGML pour résoudre l'emplacement des entités externes. Par défaut xsltproc utilise les catalogues XML installés dans /etc/xml/catalog.

*--xinclude*

Traite le document en entrée en utilisant les spécifications Xinclude. Vous pouvez obtenir plus de détails dans les spécifications de Xinclude : <http://www.w3.org/TR/xinclude/>.

*--profile --norman*

Donne des informations détaillant le temps passé pour chaque partie de la feuille de style. C'est utile pour optimiser les performances de la feuille de style.

*--dumpextensions*

Affiche la liste de toutes les extensions enregistrées sur la sortie standard (stdout).

*--nowrite*

N'écrit sur aucun fichier ni ressource.

*--nomkdir*

Ne crée aucun répertoire.

*--writ subtree chemin*

Autorise l'écriture de fichiers seulement sur le sous-répertoire *chemin*.

*--nodtdattr*

N'applique pas les attributs par défaut de la DTD du document.

## Valeurs de retour de xsltproc

xsltproc renvoie un code fournissant des informations qui peuvent être très utiles lorsqu'on l'utilise dans des scripts.

0 : normal

1 : pas d'argument

2 : trop de paramètres

3 : option inconnue

4 : le traitement de la feuille de style a échoué

5 : erreur dans la feuille de style

6 : erreur dans un des documents

7 : méthode de sortie xsl (xsl:output) non-supportée

8 : la chaîne de paramètres contient à la fois des guillemets simples et doubles

9 : erreur interne de traitement

10 : le traitement a été stoppé par un signal d'achèvement

11: Impossible d'écrire le résultat dans le fichier de sortie

## Plus d'infos sur xsltproc

Page web de la libxml : <http://www.xmlsoft.org/>

Page XSLT sur le W3C : <http://www.w3.org/TR/xslt>

# Simulator

KiCad provides an embedded electrical circuit simulator using [ngspice](#) as the simulation engine.

When working with the simulator, you may find the official *pspice* library useful. It contains common symbols used for simulation like voltage/current sources or transistors with pins numbered to match the ngspice node order specification.

There are also a few demo projects to illustrate the simulator capabilities. You will find them in *demos/simulation* directory.

## Assigning models

Before a simulation is launched, components need to have Spice model assigned.

Each component can have only one model assigned, even if component consists of multiple units. In such case, the first unit should have the model specified.

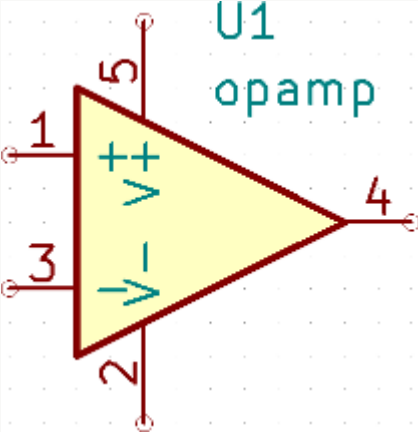
Passive components with reference matching a device type in Spice notation (*R\** for resistors, *C\** for capacitors, *L\** for inductors) will have models assigned implicitly and use the value field to determine their properties.

### NOTE

Keep in mind that in Spice notation 'M' stands for milli and 'Meg' corresponds to mega. If you prefer to use 'M' to indicate mega prefix, you may request doing so in the [simulation settings dialog](#).

Spice model information is stored as text in symbol fields, therefore you may either define it in symbol editor or schematics editor. Open symbol properties dialog and click on *Edit Spice Model* button to open Spice Model Editor dialog.

Spice Model Editor dialog has three tabs corresponding to different model types. There are two options common to all model types:

Disable symbol for simulation	When checked the component is excluded from simulation.
Alternate node sequence	<p>Allows one to override symbol pin to model node mapping. To define a different mapping, specify pin numbers in order expected by the model.</p> <p>'Example:'</p> <p>“ * connections:</p> <ul style="list-style-type: none"> <li>* 1: non-inverting input</li> <li>* 2: inverting input</li> <li>* 3: positive power supply</li> <li>* 4: negative power supply</li> <li>* 5: output</li> </ul> <p>.subckt tl071 1 2 3 4 5</p>  <p>To match the symbol pins to the Spice model nodes shown above, one needs to use an alternate node sequence option with value: "1 3 5 2 4". It is a list of pin numbers corresponding to the Spice model nodes order.</p>

## Passive

*Passive* tab allows the user to assign a passive device model (resistor, capacitor or inductor) to a component. It is a rarely used option, as normally passive components have models assigned [implicitly](#), unless component reference does not match the actual device type.

### NOTE

Explicitly defined passive device models have priority over the ones assigned implicitly. It means that once a passive device model is assigned, the reference and value fields are not taken into account during simulation. It may lead to a confusing situation when assigned model value does not match the one displayed on a schematic sheet.



Spice Model Editor

Passive

Model

Source

Type: Resistor

Passive type

Value: 1k

Spice value in simulation

In Spice values,the decimal separator is the point.  
Values can use Spice unit symbols.

Spice unit symbols in values (case insensitive):

f	femto	1e-15
p	pico	1e-12
n	nano	1e-9
u	micro	1e-6
m	milli	1e-3
k	kilo	1e3
meg	mega	1e6
g	giga	1e9
t	tera	1e12

☐ Disable symbol for simulation

☐ Alternate node sequence:

Cancel

OK

Type	Selects the device type (resistor, capacitor or inductor).
Value	Defines the device property (resistance, capacitance or inductance). The value may use common Spice unit prefixes (as listed below the text input field) and should use point as the decimal separator. Note that Spice does not correctly interpret prefixes intertwined in the value (e.g. 1k5).

## Model

*Model* tab is used to assign a semiconductor or a complex model defined in an external library file. Spice model libraries are often offered by device manufacturers.

The main text widget displays the selected library file contents. It is a common practice to put models description inside library files, including the node order.

The image shows a 'Spice Model Editor' window with three tabs: 'Passive', 'Model', and 'Source'. The 'Model' tab is active. It contains the following fields and controls:

- Library:** A text field containing 'ad8051.lib' and a 'Select file...' button.
- Model:** A dropdown menu showing 'AD8051'.
- Type:** A dropdown menu showing 'Subcircuit'.
- Model Definition:** A large text area containing the following SPICE code:

```
.SUBCKT AD8051 1 2 99 50 45
*
* INPUT STAGE
*
Q1 4 3 5 QPI
Q2 6 2 7 QPI
RC1 50 4 20.5k
RC2 50 6 20.5k
RE1 5 8 5k
RE2 7 8 5k
EOS 3 1 POLY(1) 53 98 1.7E-3 1
IOS 1 2 0.1u
FNOI1 1 0 VMEAS2 1E-4
FNOI2 2 0 VMEAS2 1E-4

CPAR1 3 50 1.7p
CPAR2 2 50 1.7p
VCMH1 99 9 1
VCMH2 99 10 1
D1 5 9 DX
D2 7 10 DX
IBIAS 99 8 73u
*
* INTERNAL VOLTAGE REFERENCE
*
EREF1 98 0 POLY(2) 99 0 50 0 0 0.5 0.5
```
- Options:** Two checkboxes at the bottom: 'Disable symbol for simulation' and 'Alternate node sequence:'. The 'Alternate node sequence' checkbox is followed by an empty text field.
- Buttons:** 'Cancel' and 'OK' buttons at the bottom right.

File	Path to a Spice library file. This file is going to be used by the simulator, as it is added using <i>.include</i> directive.
Model	Selected device model. When a file is selected, the list is filled with available models to choose from.
Type	Selects model type (subcircuit, BJT, MOSFET or diode). Normally it is set automatically when a model is selected.

## Source

*Source* tab is used to assign a power or signal source model. There are two sections: *DC/AC analysis* and *Transient analysis*. Each defines source parameters for the corresponding simulation type.

*Source type* option applies to all simulation types.

The image shows the 'Spice Model Editor' dialog box with three tabs: 'Passive', 'Model', and 'Source'. The 'Source' tab is selected. It contains two main sections: 'DC/AC analysis' and 'Transient analysis'.

**DC/AC analysis:**

- DC: [ ] Volts/Amps
- AC magnitude: 1 [ ] Volts/Amps
- AC phase: [ ] radians

**Transient analysis:**

Four sub-tabs are available: 'Pulse' (selected), 'Sinusoidal', 'Exponential', and 'Piece-wise Linear'.

**Pulse settings:**

- Initial value: [ ] Volts/Amps
- Pulsed value: [ ] Volts/Amps
- Delay time: [ ] seconds
- Rise time: [ ] seconds
- Fall time: [ ] seconds
- Pulse width: [ ] seconds
- Period: [ ] seconds

**Source type:**

- ☒ Voltage ☐ Current

**Other options:**

- ☐ Disable symbol for simulation
- ☐ Alternate node sequence: [ ]

At the bottom right are 'Cancel' and 'OK' buttons.

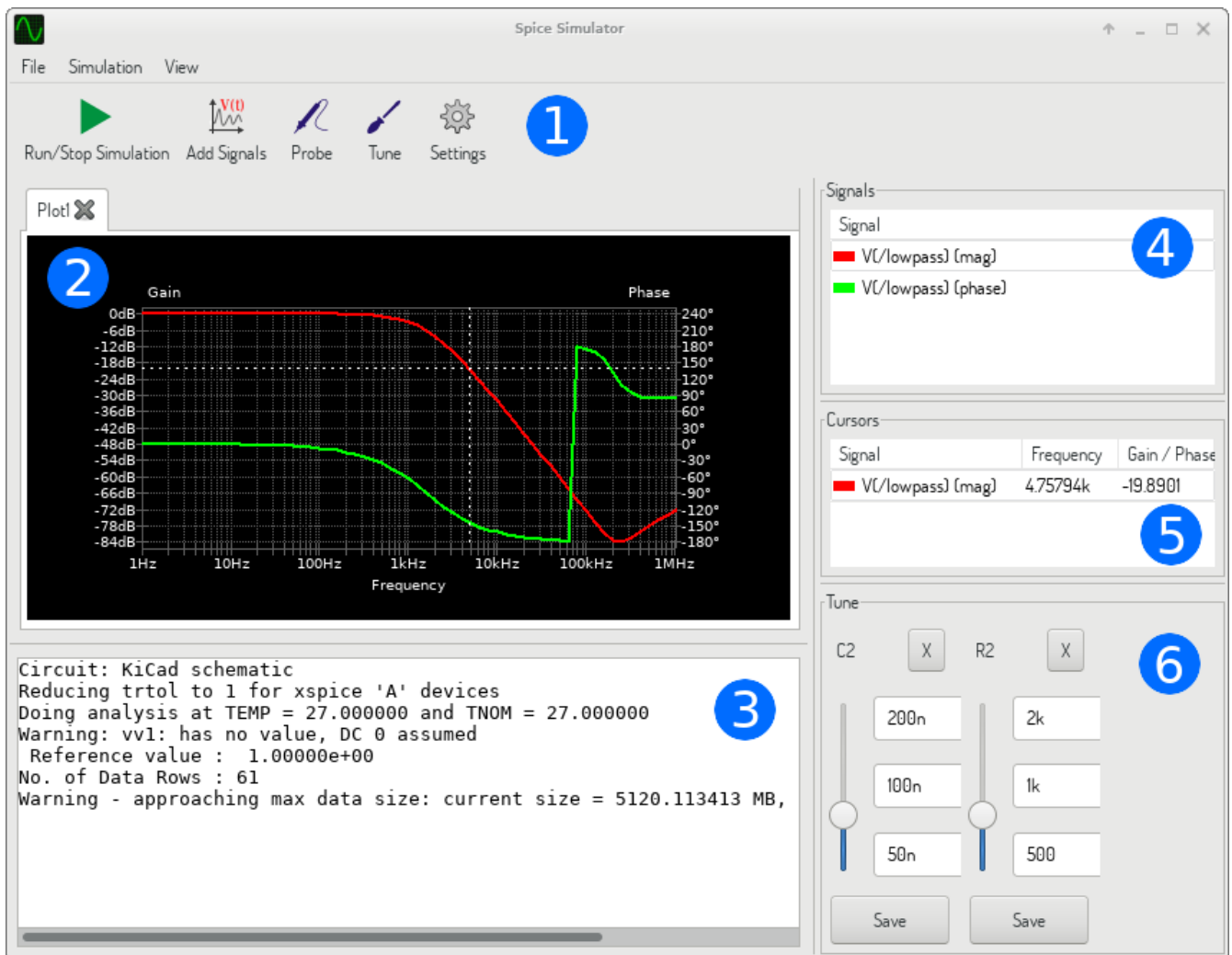
Refer to the [ngspice documentation](#), chapter 4 (Voltage and Current Sources) for more details about sources.

## Spice directives

It is possible to add Spice directives by placing them in text fields on a schematic sheet. This approach is convenient for defining the default simulation type. This functionality is limited to Spice directives starting with a dot (e.g. ".tran 10n 1m"), it is not possible to place additional components using text fields.

## Simulation

To launch a simulation, open *Spice Simulator* dialog by selecting menu *Tools* → *Simulator* in the schematics editor window.



The dialog is divided into several sections:

- [Toolbar](#)
- [Plot panel](#)
- [Output console](#)
- [Signals list](#)
- [Cursors list](#)
- [Tune panel](#)

## Menu

## File

New Plot	Create a new tab in the plot panel.
Open Workbook	Open a list of plotted signals.
Save Workbook	Save a list of plotted signals.
Save as image	Export the active plot to a .png file.
Save as .csv file	Export the active plot raw data points to a .csv file.
Exit Simulation	Close the dialog.

## Simulation

Run Simulation	Perform a simulation using the current settings.
Add signals...	Open a dialog to select signals to be plotted.
Probe from schematics	Start the schematics <a href="#">Probe</a> tool.
Tune component value	Start the <a href="#">Tuner</a> tool.
Show SPICE Netlist...	Open a dialog showing the generated netlist for the simulated circuit.
Settings...	Open the <a href="#">simulation settings dialog</a> .

## View

Zoom In	Zoom in the active plot.
Zoom Out	Zoom out the active plot.
Fit on Screen	Adjust the zoom setting to display all plots.
Show grid	Toggle grid visibility.
Show legend	Toggle plot legend visibility.

## Toolbar



The top toolbar provides access to the most frequently performed actions.

Run/Stop Simulation	Start or stop the simulation.
Add Signals	Open a dialog to select signals to be plotted.
Probe	Start the schematics <a href="#">Probe</a> tool.
Tune	Start the <a href="#">Tuner</a> tool.
Settings	Open the <a href="#">simulation settings dialog</a> .

## Plot panel

Visualizes the simulation results as plots. One can have multiple plots opened in separate tabs, but only the active one is updated when a simulation is executed. This way it is possible to compare simulation results for different runs.

Plots might be customized by toggling grid and legend visibility using [View](#) menu. When a legend is visible, it can be dragged to change its position.

Plot panel interaction:

- scroll mouse wheel to zoom in/out
- right click to open a context menu to adjust the view
- draw a selection rectangle to zoom in the selected area
- drag a cursor to change its coordinates

## Output console

Output console displays messages from the simulator. It is advised to check the console output to verify there are no errors or warnings.

## Signals list

Shows the list of signals displayed in the active plot.

Signals list interaction:

- right click to open a context menu to hide signal or toggle cursor
- double click to hide signal

## Cursors list

Shows the list of cursors and their coordinates. Each signal may have one cursor displayed. Cursors visibility is set using the [Signals](#) list.

## Tune panel

Displays components picked with the [Tuner](#) tool. Tune panel allows the user to quickly modify component values and observe their influence on the simulation results - every time a component value is changed, the simulation is rerun and plots are updated.

For each component there a few controls associated:

The top text field sets the maximum component value.

- The middle text field sets the actual component value.
- The bottom text field sets the minimum component value.
- Slider allows the user to modify the component value in a smooth way.
- *Save* button modifies component value on the schematics to the one selected with the slider.
- *X* button removes component from the Tune panel and restores its original value.

The three text fields recognize Spice unit prefixes.

## Tuner tool

Tuner tool lets the user pick components for tuning.

To select a component for tuning, click on one in the schematics editor when the tool is active. Selected components will appear in the [Tune](#) panel. Only passive components might be tuned.

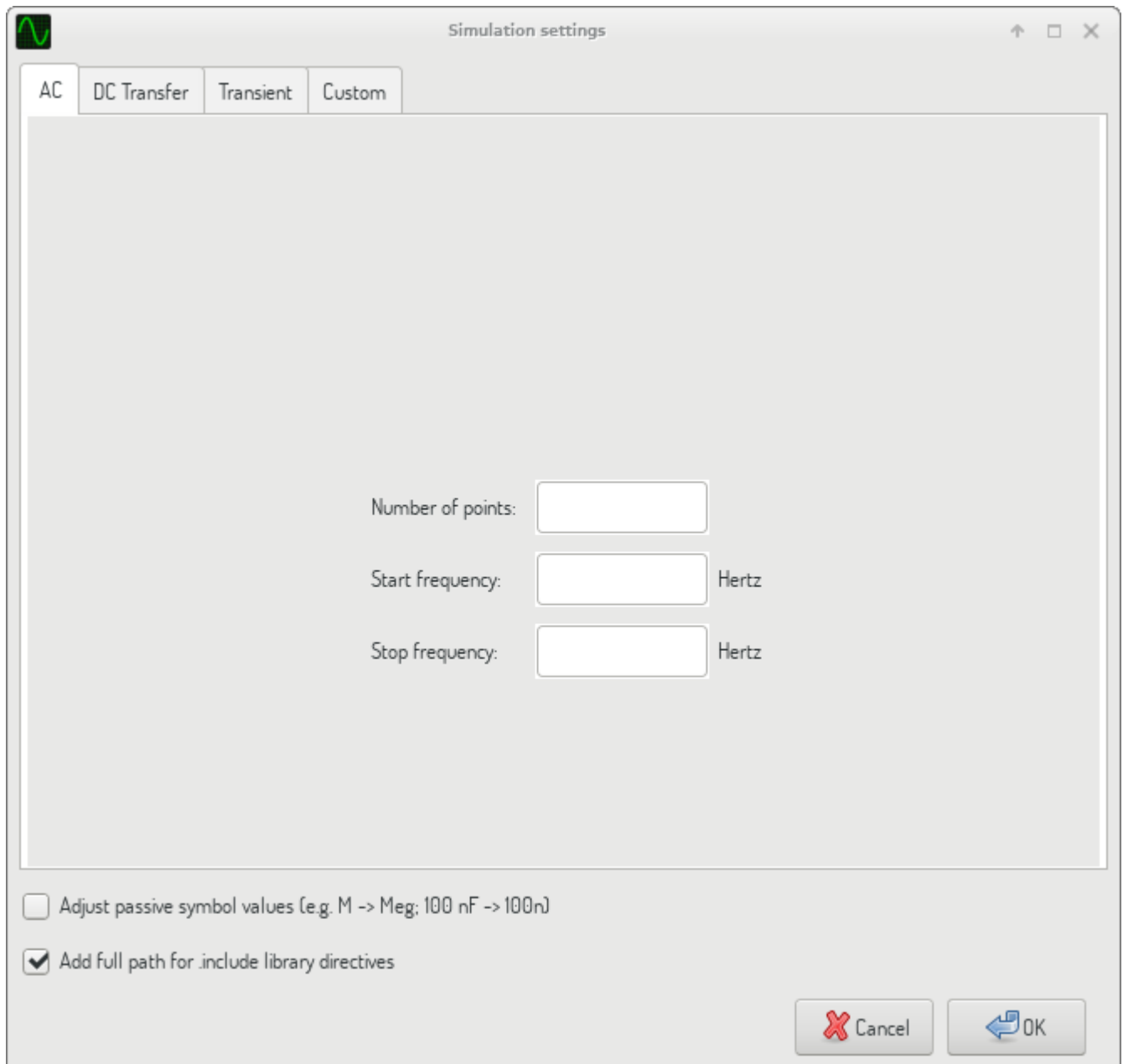
## Probe tool

Probe tool provides an user-friendly way of selecting signals for plotting.

To add a signal to plot, click on a corresponding wire in the schematics editor when the tool is active.



## Simulation settings



The image shows a 'Simulation settings' dialog box with a title bar containing a green waveform icon, the text 'Simulation settings', and standard window controls. Below the title bar are four tabs: 'AC', 'DC Transfer', 'Transient', and 'Custom'. The 'AC' tab is selected. The main area of the dialog is a large, light gray rectangle. Inside this area, there are three rows of input fields: 'Number of points:' followed by a text box; 'Start frequency:' followed by a text box and the unit 'Hertz'; and 'Stop frequency:' followed by a text box and the unit 'Hertz'. Below the main area, there are two checkboxes: 'Adjust passive symbol values (e.g. M -> Meg; 100 nF -> 100n)' which is unchecked, and 'Add full path for .include library directives' which is checked. At the bottom right, there are two buttons: 'Cancel' with a red 'X' icon and 'OK' with a blue arrow icon.

Simulation settings dialog lets the user set the simulation type and parameters. There are four tabs:

- AC
- DC Transfer
- Transient
- Custom

The first three tabs provide forms where simulation parameters might be specified. The last tab allows the user to type in custom Spice directives to set up a simulation. You can find more information about simulation types and parameters in the [ngspice documentation](#), chapter 1.2.

An alternative way to configure a simulation is to type [Spice directives](#) into text fields on schematics. Any text field directives related to simulation type are overridden by the settings selected in the dialog. It means

that once you start using the simulation dialog, the dialog overrides the schematics directives until the simulator is reopened.

There are two options common to all simulation types:

Adjust passive symbol values	Replace passive symbol values to convert common component values notation to Spice notation.
Add full path for .include library directives	Prepend Spice model library file names with full path. Normally full path is required by ngspice to access a library file.