KiCad Status 2024

Version 8 and beyond

Ki Roberto Fernandez Bautista
14/06/2024

CERN electronics forum

Version 8

Common Improvements
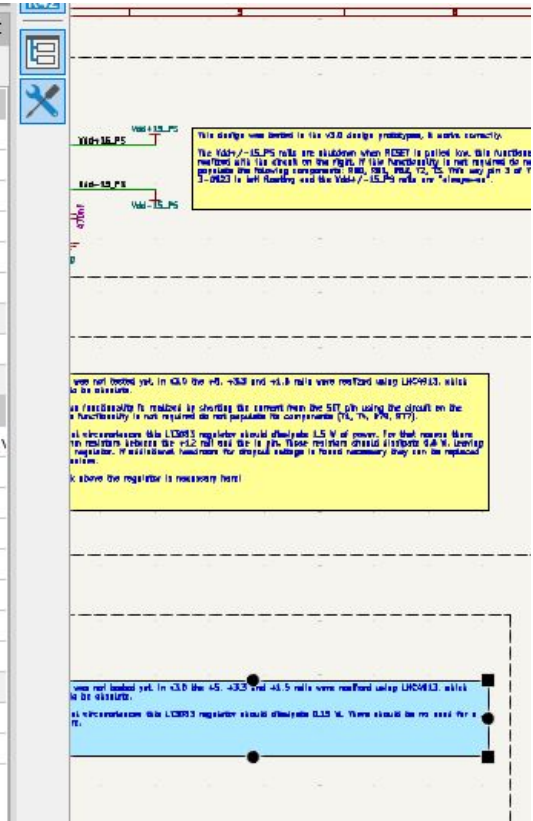
Properties Panel in all editors

Version 8

Common Improvements

Properties Panel in all editors

Import from third party tools

	Project	Schematic	PCB	Symbols	Footprints
Altium Designer		●	●	●	●
Altium Circuit Maker		●	●	●	●
Altium Circuit Studio		●	●	●	●
CADSTAR	●	●	●	●	●
EAGLE	●	●	●	●	●
EasyEDA / JLCEDA Std	●	●	●	●	●
EasyEDA / JLCEDA Pro	●	●	●	●	●
Fabmaster	N/A	N/A	●	N/A	N/A
gEDA					●
LTspice	N/A	●	N/A		N/A
P-CAD			●		
Solidworks PCB		●	●	●	●

Key: ● Supported in KiCad 7.0   ● Supported in KiCad 8.0

+ Import vector graphics (**DXF and SVG**) to Schematic + Symbol editors

Import Vector Graphics File   _ □ ✕

File: [   ] 📁

Placement
● Interactive placement
○ At   X: [0]   mils   Y: [0]   mils

Import Parameters
Import scale: [1.000000]

DXF Parameters
Default line width: [7.87402]   mils
Default units: [Inches   ⌄]

[Cancel] [OK]
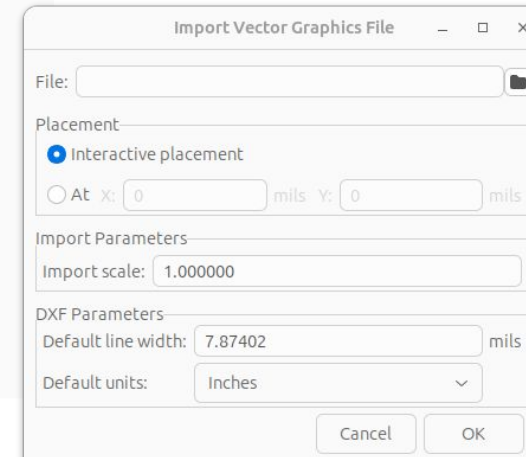
Version 8

```
kicad-cli pcb drc
kicad-cli sch erc
kicad-cli sch bom
```

Common Improvements

Properties Panel in all editors

Import from third party tools

Command line drc, erc and native bom

```
user@pc:$ kicad-cli pcb drc myboard.kicad_pcb
Loading board
Running DRC...
Found 497 violations
Found 200 unconnected items
Saved DRC Report to myboard.rpt
user@pc:$ cat myboard.rpt
** Drc report for myboard.kicad_pcb **
** Created on 2024-06-09T18:22:01+0200 **

** Found 497 DRC violations **
[diff_pair_gap_out_of_range]: Differential pair gap out of range (100 ohm diff pair maximum gap 0.4500 mm; actual 0.6188
mm)
    Rule: 100 ohm diff pair; error
    @(161.8215 mm, 95.9016 mm): Track [/FPGAControler/SPI_SCLK_P] on L4_SIG, length 0.6443 mm
    @(162.3504 mm, 96.4656 mm): Track [/FPGAControler/SPI_SCLK_N] on L4_SIG, length 1.2558 mm
[diff_pair_uncoupled_length_too_long]: Differential uncoupled length too long (100 ohm diff pair maximum uncoupled length
 10.0000 mm; actual 10.8907 mm)
    Rule: 100 ohm diff pair; error
    @(161.3660 mm, 96.3572 mm): Track [/FPGAControler/SPI_SCLK_P] on F.Cu, length 0.7021 mm
    @(161.3624 mm, 97.3536 mm): Track [/FPGAControler/SPI_SCLK_N] on F.Cu, length 0.7071 mm
[diff_pair_gap_out_of_range]: Differential pair gap out of range (100 ohm diff pair maximum gap 0.4500 mm; actual 0.6188
mm)
    Rule: 100 ohm diff pair; error
    @(161.8215 mm, 95.9016 mm): Track [/FPGAControler/SPI_SCLK_P] on L4_SIG, length 0.6443 mm
    @(162.3504 mm, 96.4656 mm): Track [/FPGAControler/SPI_SCLK_N] on L4_SIG, length 1.2558 mm
[diff_pair_gap_out_of_range]: Differential pair gap out of range (100 ohm diff pair maximum gap 0.4500 mm; actual 0.4769
mm)
    Rule: 100 ohm diff pair; error
    @(184.1112 mm, 94.3424 mm): Track [/FPGAControler/SPI_CS_P] on L4_SIG, length 5.0770 mm
    @(183.7836 mm, 94.9048 mm): Track [/FPGAControler/SPI_CS_N] on L4_SIG, length 5.0770 mm
[diff_pair_uncoupled_length_too_long]: Differential uncoupled length too long (100 ohm diff pair maximum uncoupled length
 10.0000 mm; actual 12.6054 mm)
```

Version 8

SPICE simulator improvements
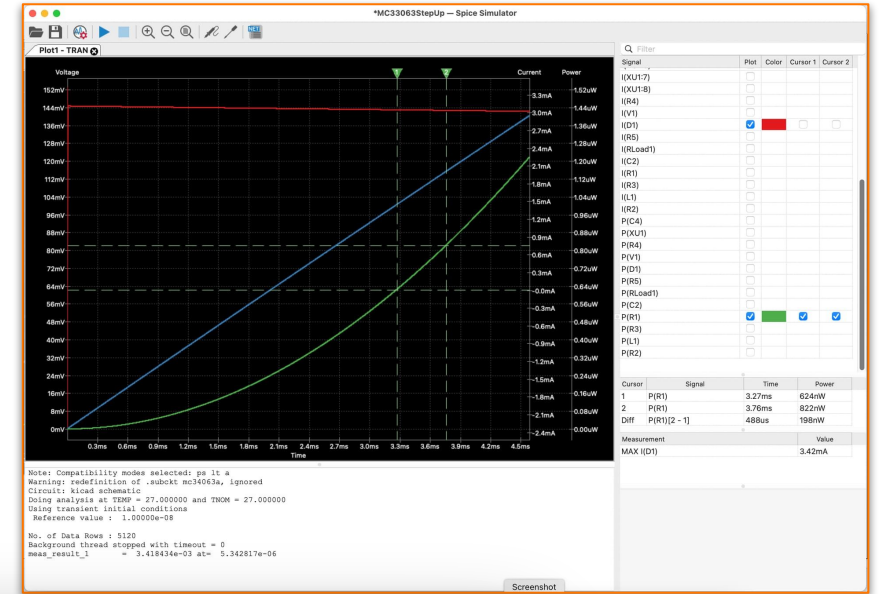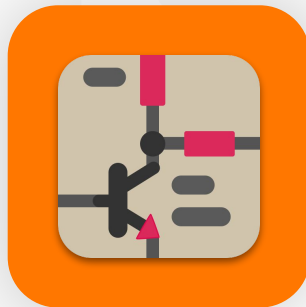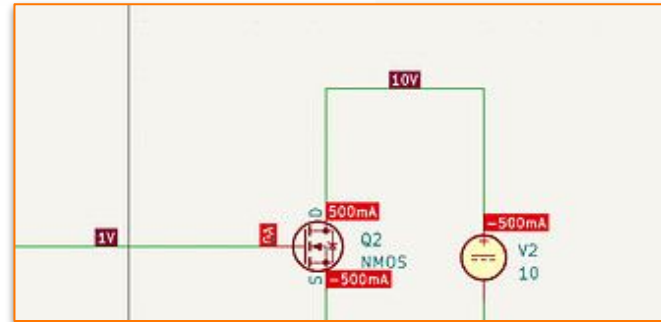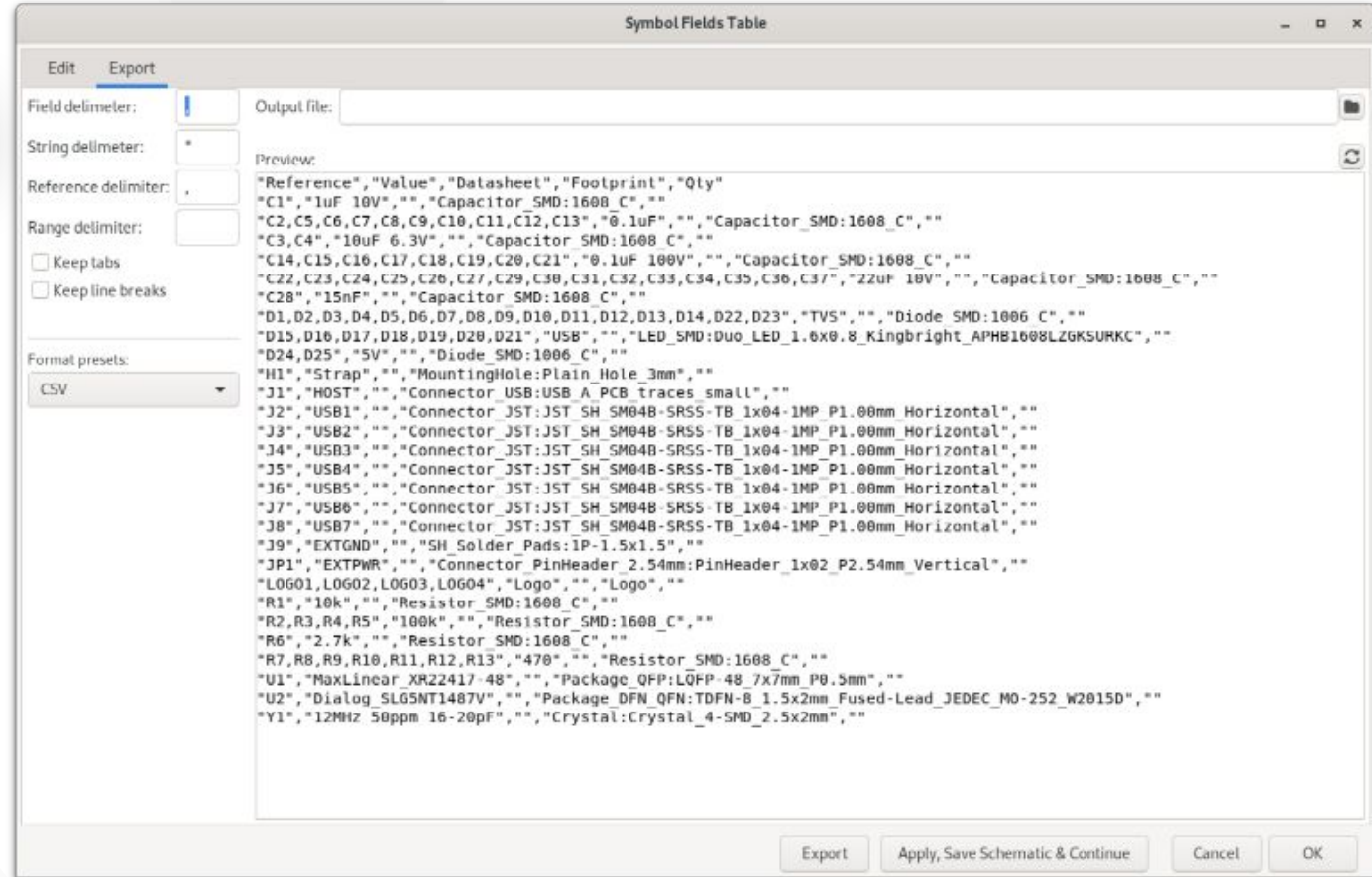

Operating point (.op) Display
Differential Cursors
Power Plotting (e.g. Vin - Vout)
New simulations: pole-zero, noise, S-parameter, and FFT

Version 8

Schematic Editor Improvements
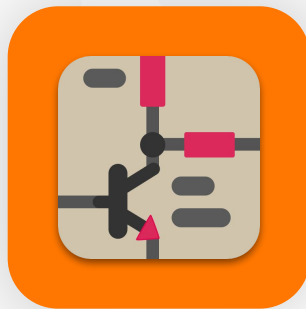
SPICE simulator improvements

Native BOM export tool

Version 8

Schematic Editor Improvements

SPICE simulator improvements

Native BOM export tool

New UI panels

**Properties Panel
Net Navigator Panel
Search Panel**

Version 8

PCB Editor Improvements
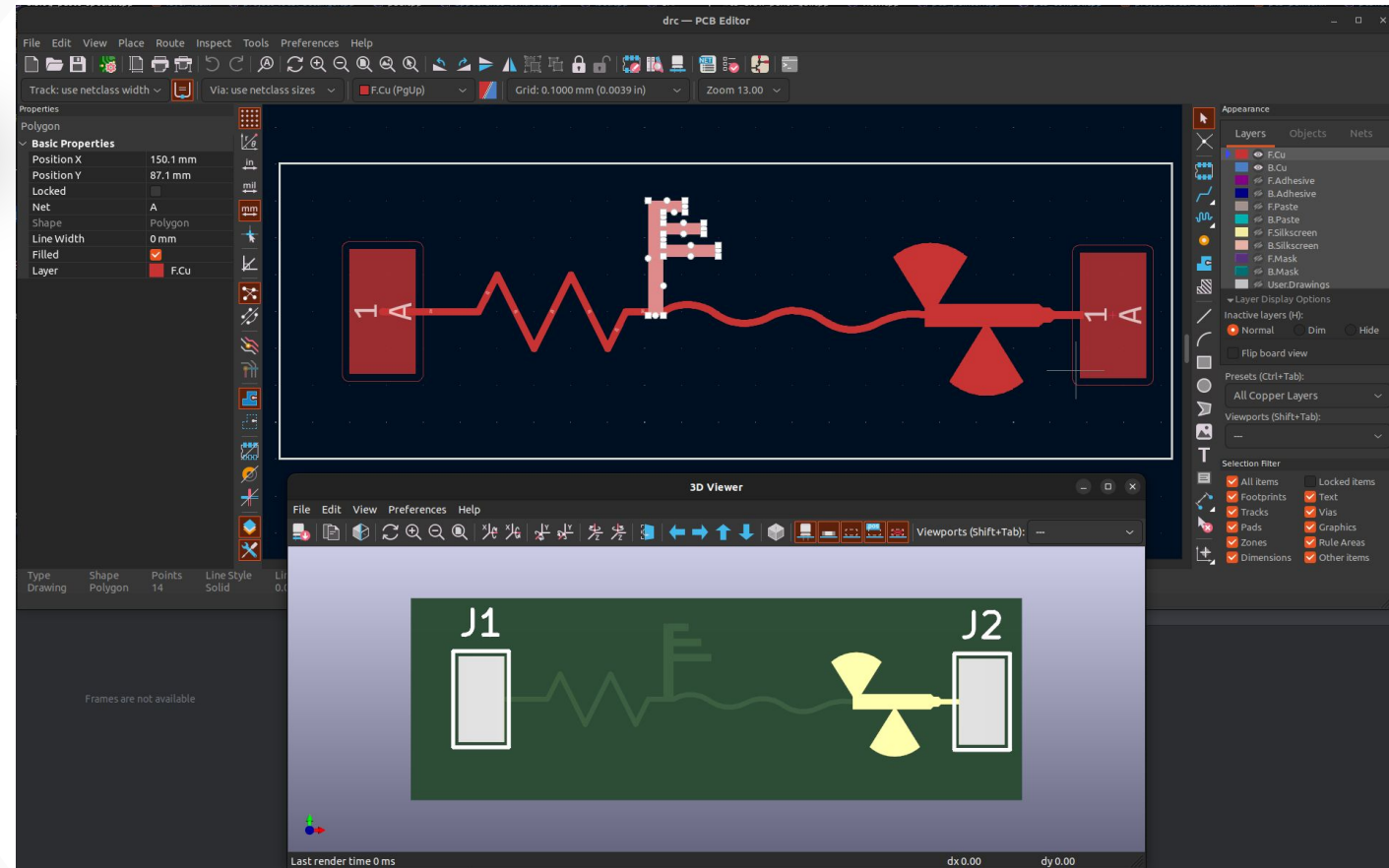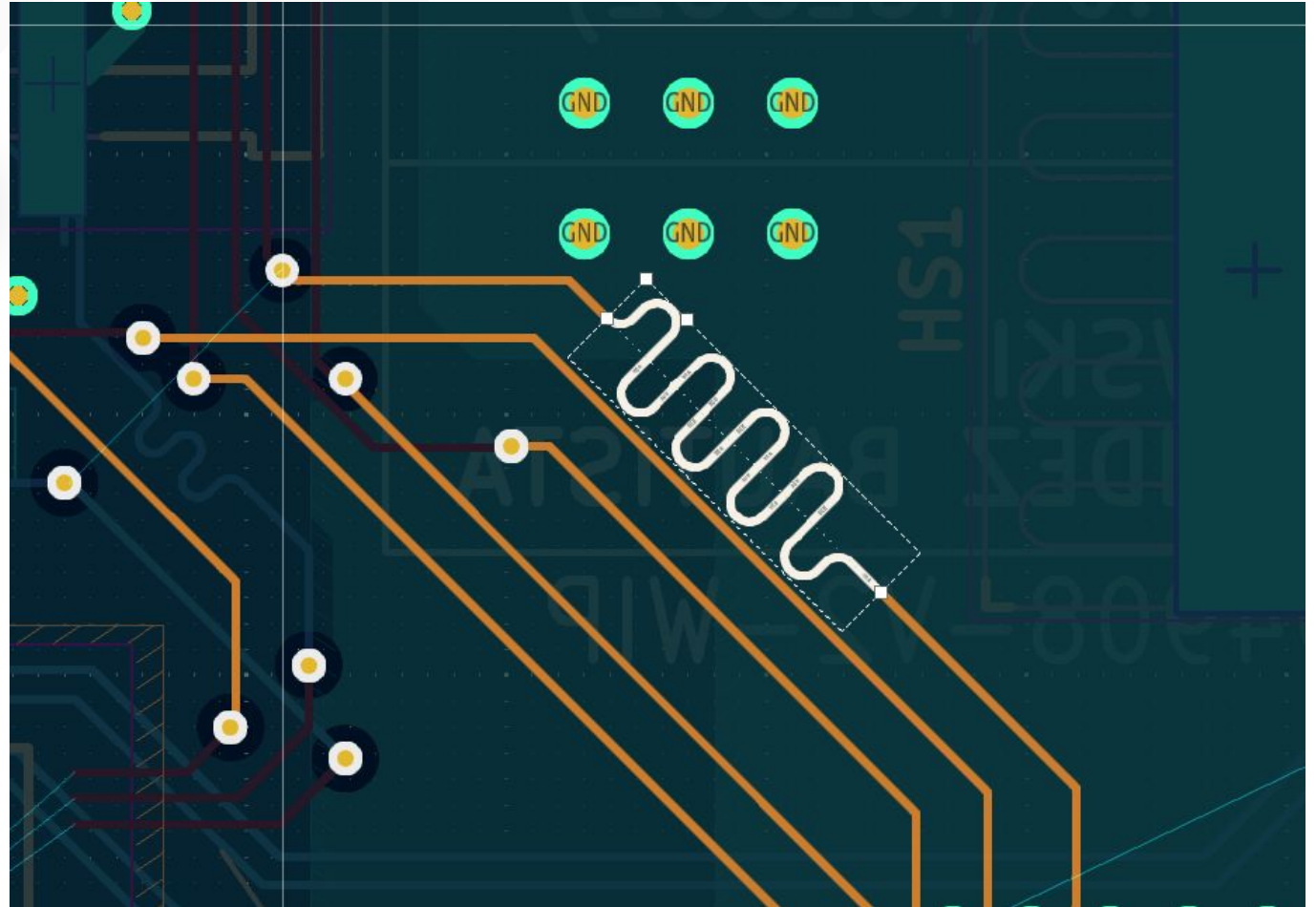
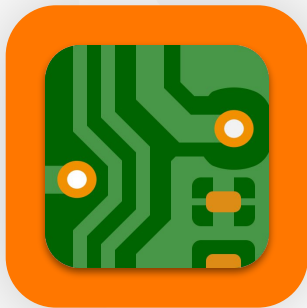Assign nets to Graphic Shapes

Version 8

PCB Editor Improvements

Assign nets to Graphic Shapes
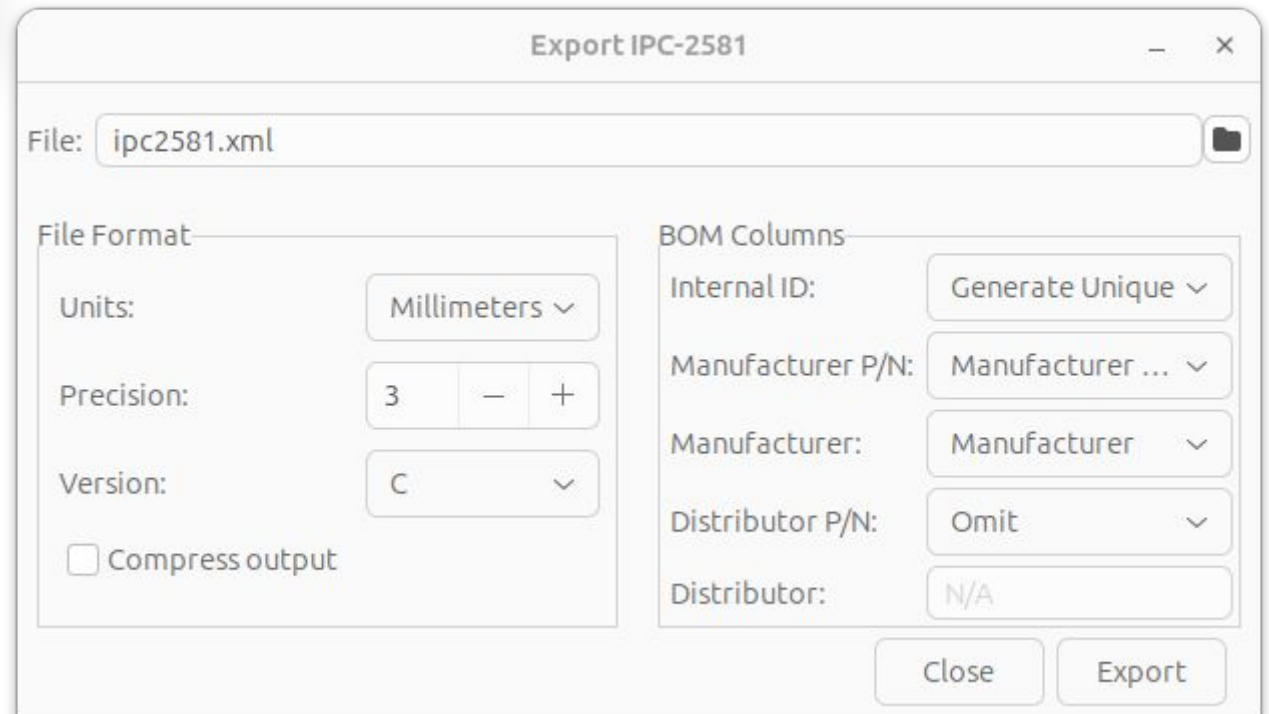
Interactive editable length tuner

Version 8

PCB Editor Improvements

Assign nets to Graphic Shapes

Interactive editable length tuner

IPC-2581 output

Export IPC-2581   — ✕

File: ipc2581.xml 📁

File Format

Units: Millimeters ⌄

Precision: 3 — +

Version: C ⌄

☐ Compress output

BOM Columns

Internal ID: Generate Unique ⌄

Manufacturer P/N: Manufacturer ... ⌄

Manufacturer: Manufacturer ⌄

Distributor P/N: Omit ⌄

Distributor: N/A

Close   Export

Version 8

PCB Editor Improvements

Assign nets to Graphic Shapes
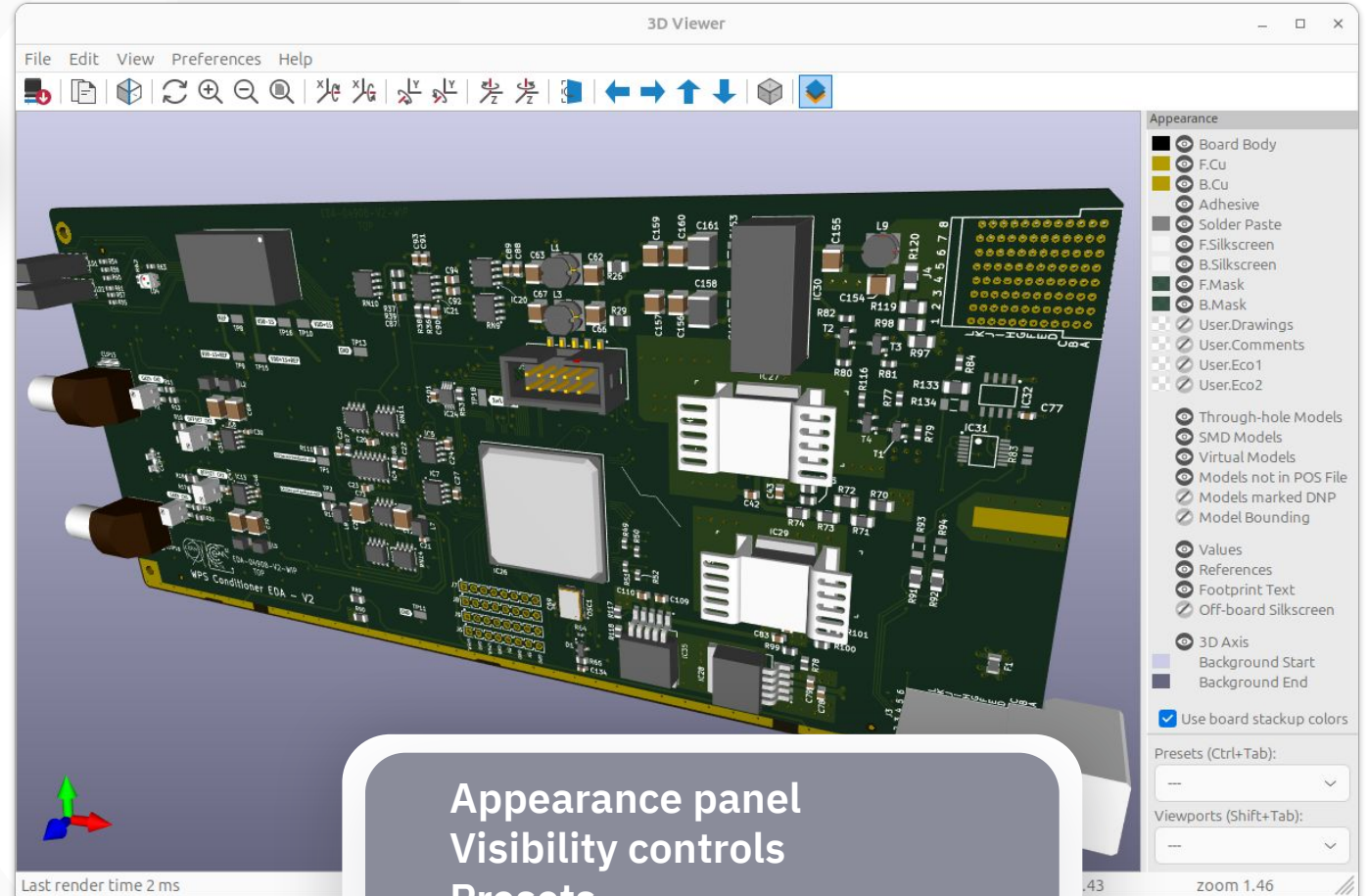
Interactive editable length tuner

IPC-2581 output

3D viewer

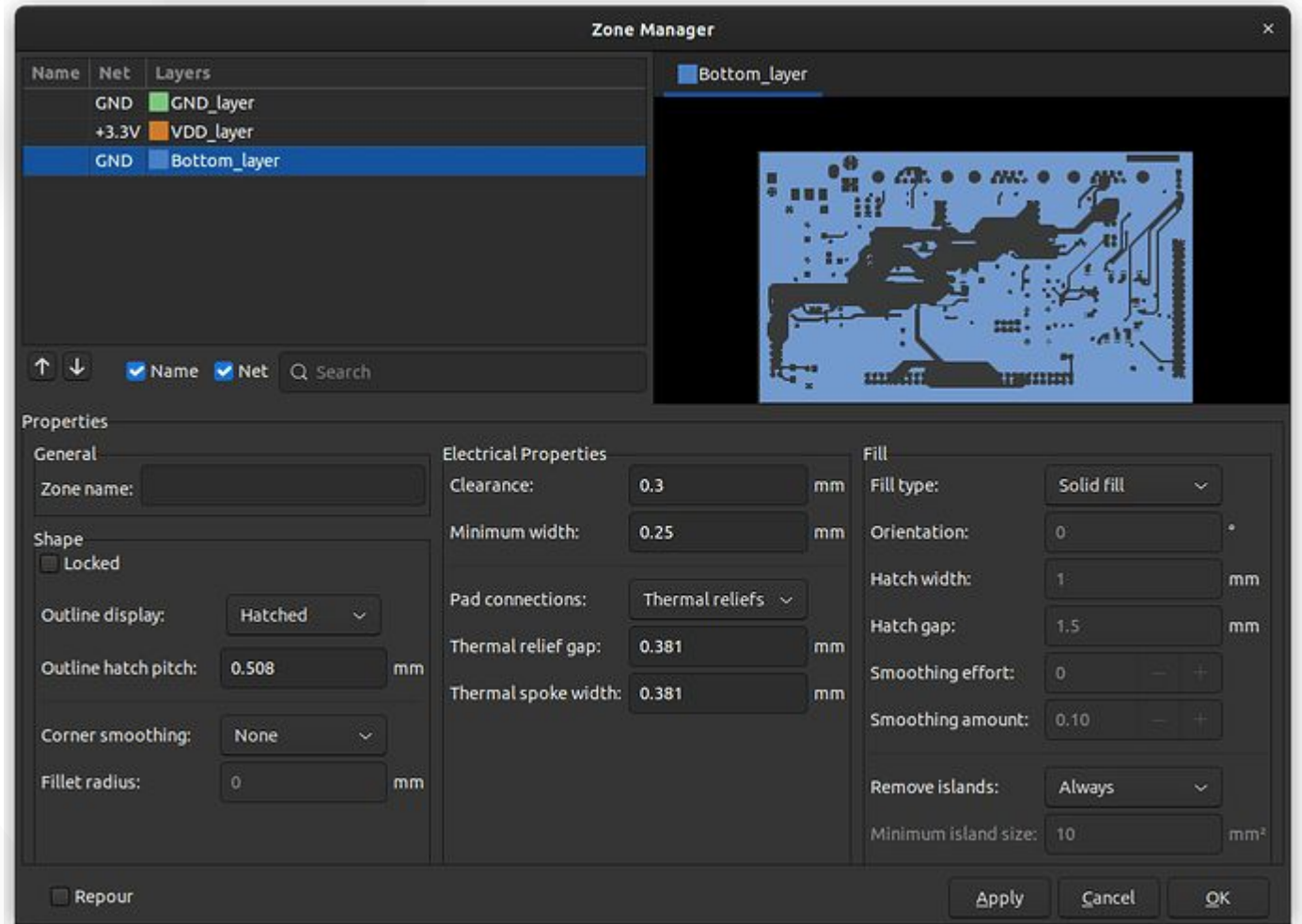**Appearance panel**
Visibility controls
Presets
Viewports

Upcoming Version 9 (Jan 2025)

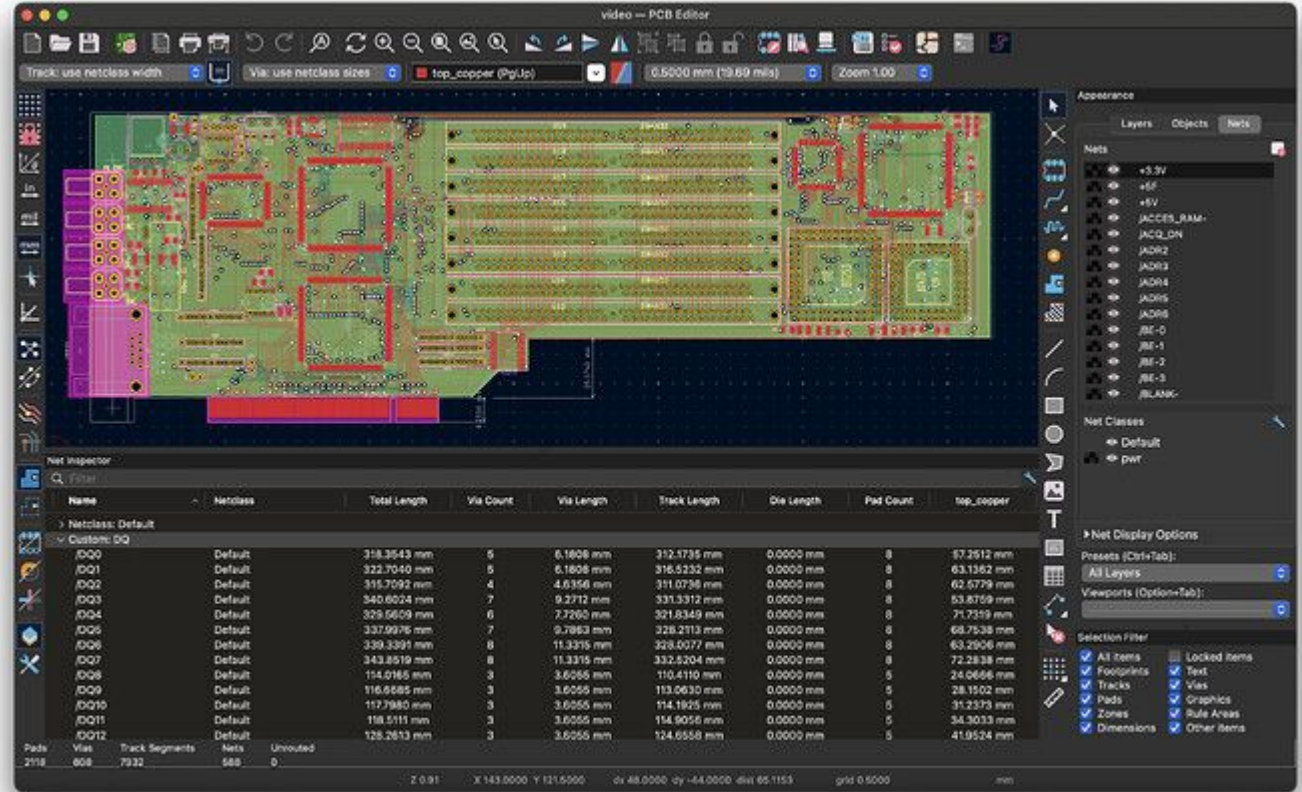**Completed Improvements**

Zone Manager

Upcoming Version 9 (Jan 2025)

Completed Improvements
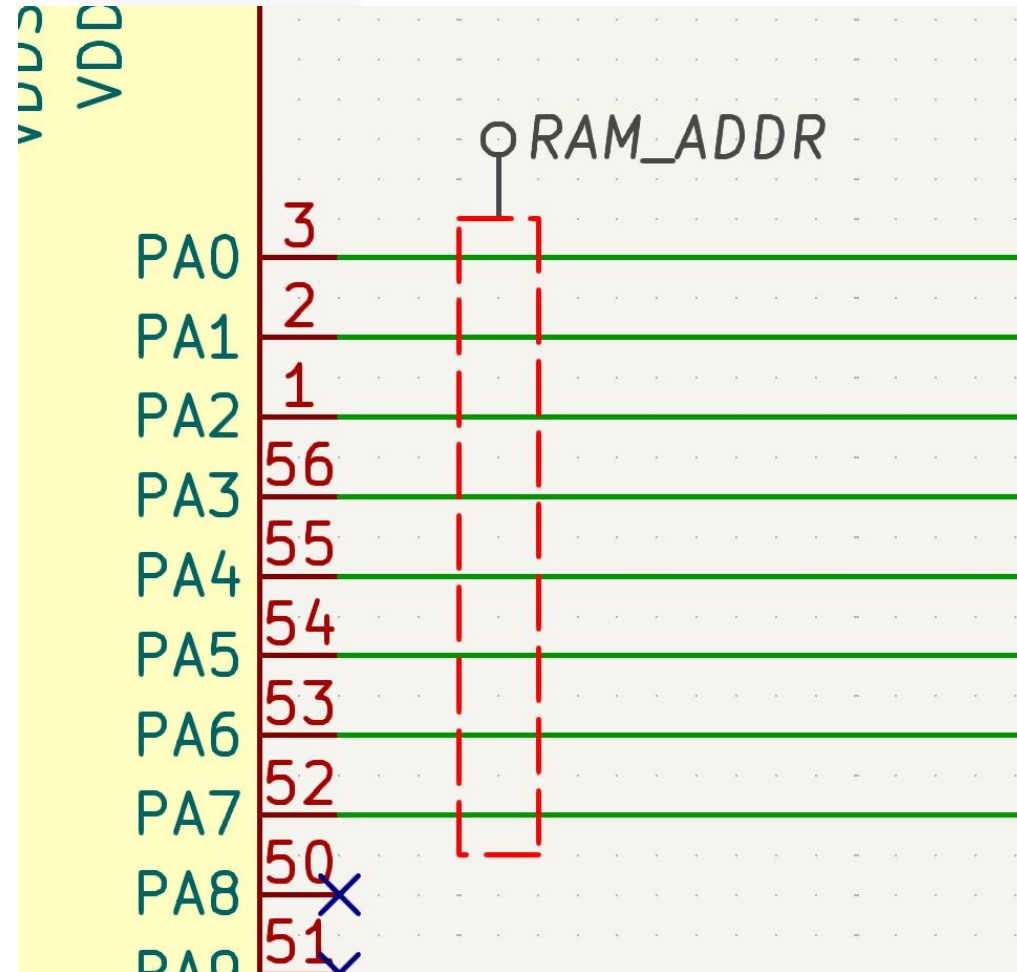
Zone Manager

Net Inspector docked panel

Upcoming Version 9 (Jan 2025)

Completed Improvements

Zone Manager

Net Inspector docked panel

Schematic Rule Areas (Blankets)
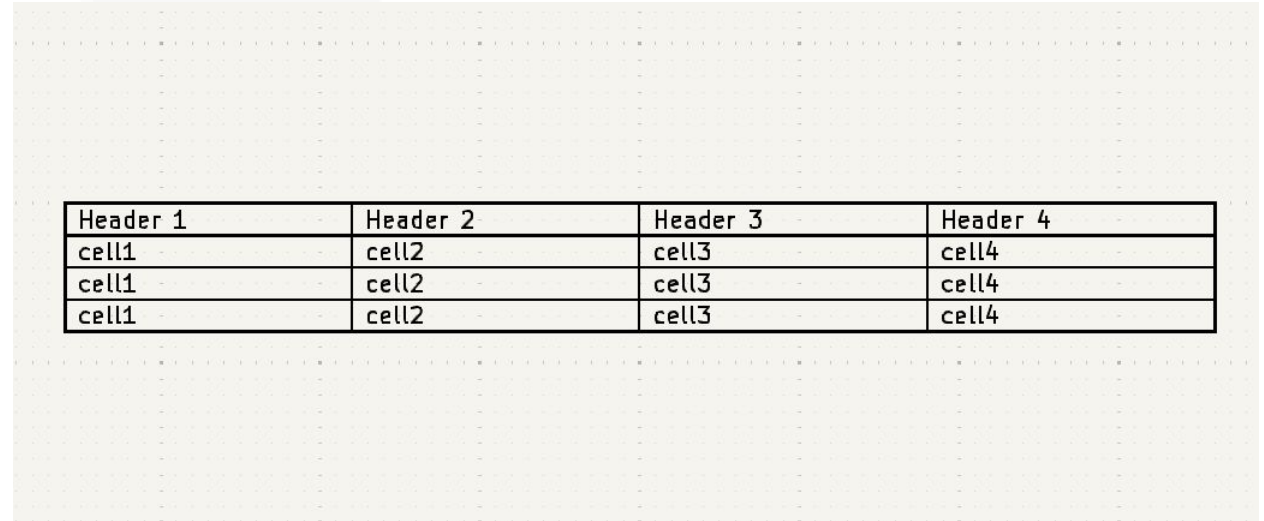
Upcoming Version 9 (Jan 2025)
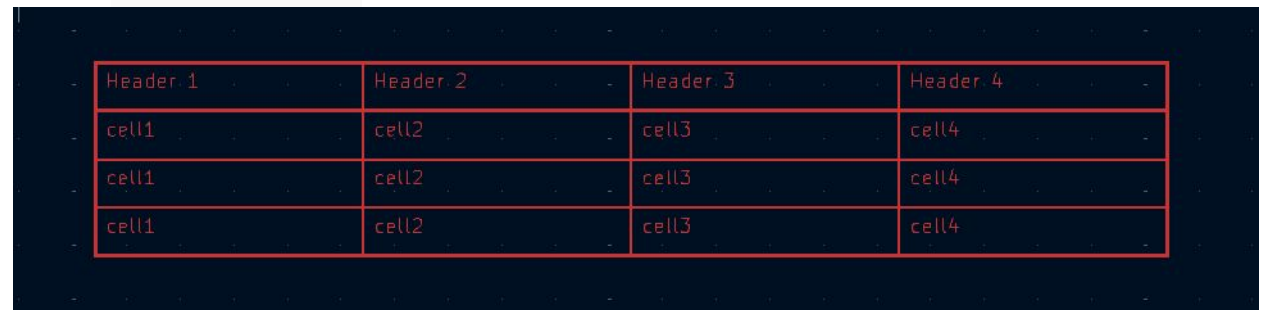
Completed Improvements

Zone Manager

Net Inspector docked panel

Schematic Rule Areas (Blankets)

Native table support

Header 1	Header 2	Header 3	Header 4
cell1	cell2	cell3	cell4
cell1	cell2	cell3	cell4
cell1	cell2	cell3	cell4

Header 1	Header 2	Header 3	Header 4
cell1	cell2	cell3	cell4
cell1	cell2	cell3	cell4
cell1	cell2	cell3	cell4

Upcoming Version 9 (Jan 2025)

Completed Improvements

Zone Manager

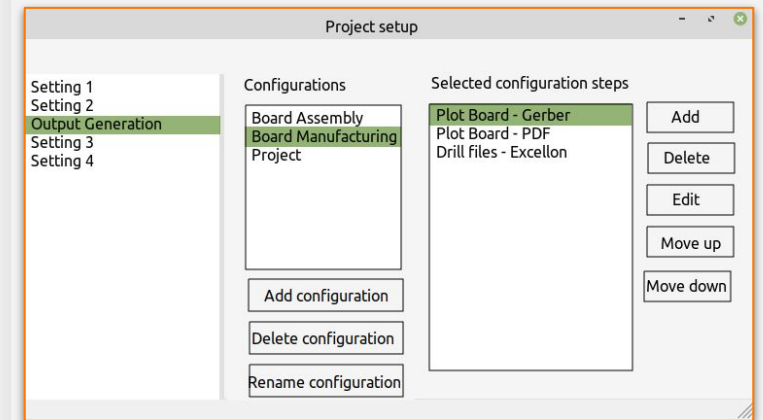Net Inspector docked panel

Schematic Rule Areas (Blankets)

Native table support

Embedded files (e.g. 3D models)

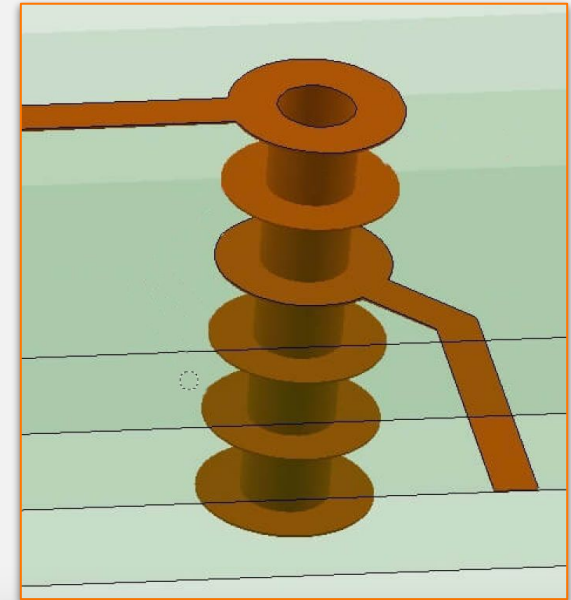# Upcoming Version 9 (Jan 2025)

Planned features

GUI Output Jobs

Upcoming Version 9 (Jan 2025)

Planned features

Padstacks

GUI Output Jobs

Tented/Untented Vias ✔️
Import from other tools

Upcoming Version 9 (Jan 2025)

Planned features

Bare board

Assembly Variants

Padstacks

GUI Output Jobs

Assembly Variant 1

Assembly Variant 2

Upcoming Version 9 (Jan 2025)

Planned features

ODB++

Assembly Variants

Padstacks

GUI Output Jobs
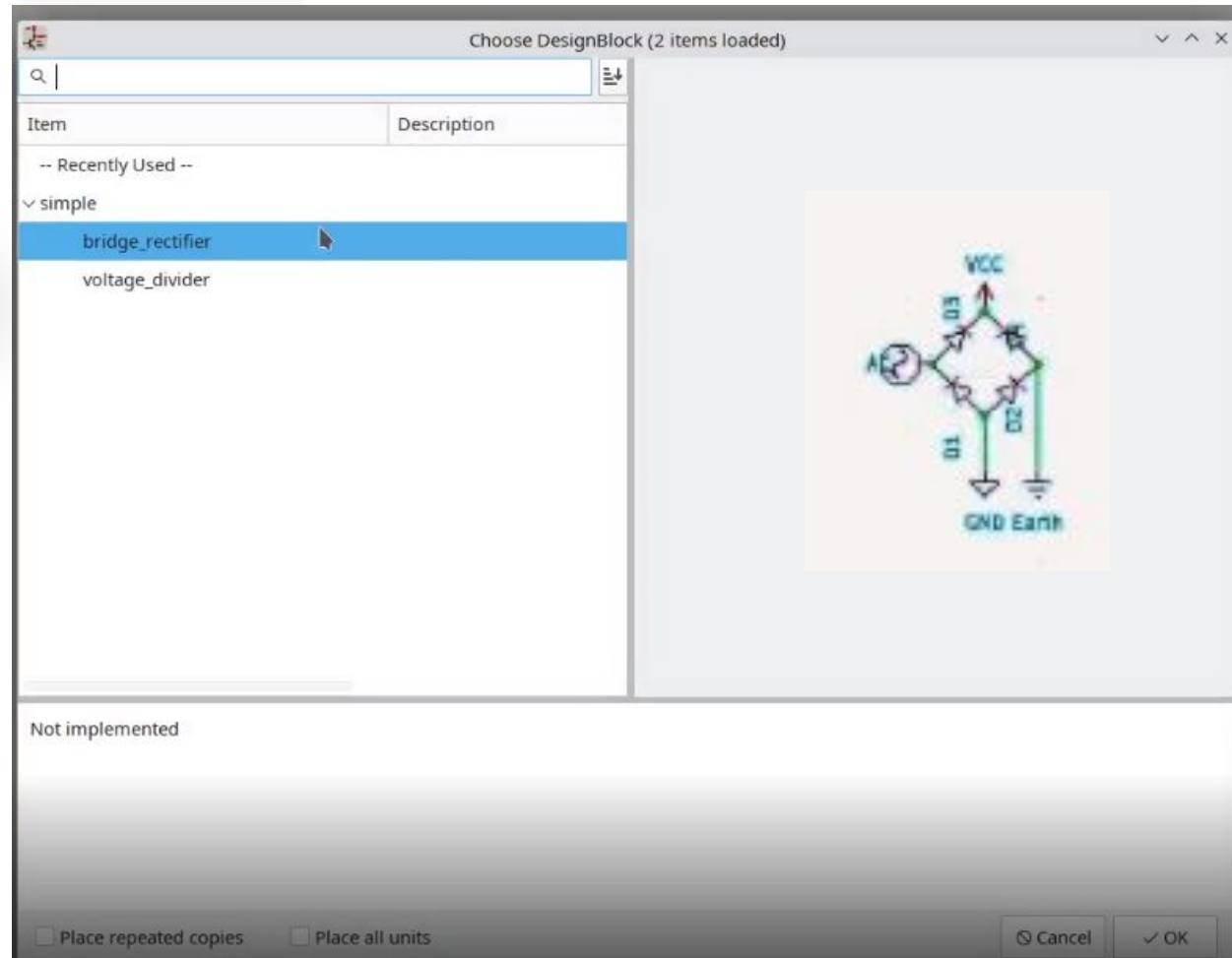
Upcoming Version 9 (Jan 2025)

Planned features

Reusable Blocks

ODB++

Assembly Variants

Padstacks

GUI Output Jobs

Want to know more?

KiCad v8 release blog post:

https://www.kicad.org/blog/2024/02/Version-8.0.0-Released/

Upcoming KiCad v9 features forum post:

https://forum.kicad.info/t/post-v8-new-features-and-development-news/48614

Questions?