



Continuous Integration mit KiCad

Oder auch: Dumme Fehler und nervige Aufgaben vermeiden

Oder auch: Automatisierte Hilfe beim PCB-Review



Über mich

Raphael Lehmann

🎓 *Elektrotechnik, Informationstechnik & technische Informatik @ RWTH Aachen*

🤖 Roberclub Aachen e.V.

⭐ TechAachen e.V.

Kontakt

✉️ raphael@rleh.de

🐦 [@rleh_de](https://twitter.com/rleh_de)

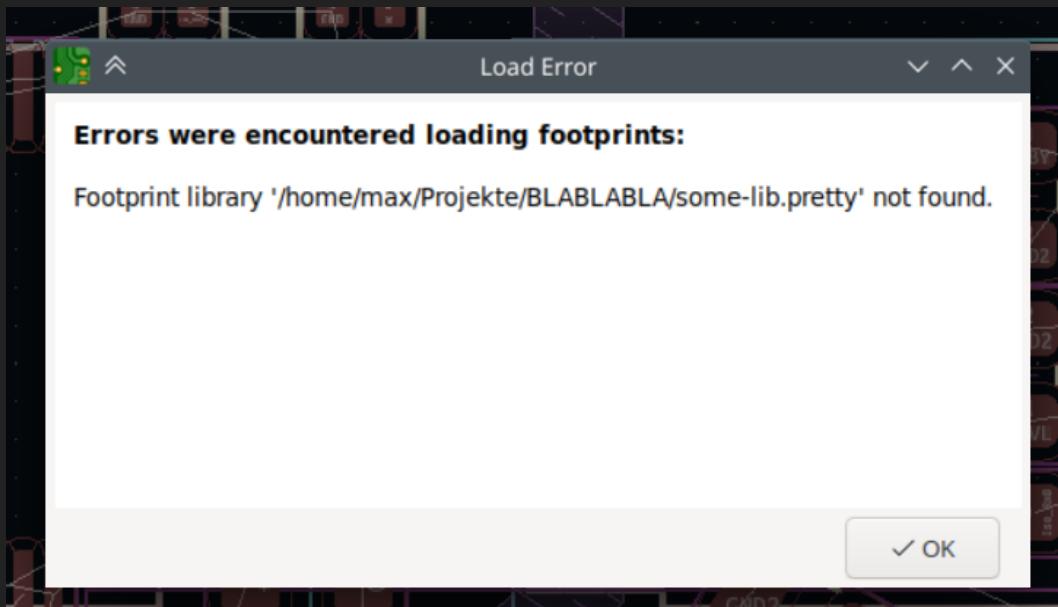
🐱 [@rleh](https://github.com/rleh)

PCB Design Prozess

1. 🖍 Schaltplan
2. 💬 Review
3. 🖌 Layout
4. 💬 Review
5. ✂ Anmerkungen aus Reviews umsetzen
6. 🛠 Fertigungsdaten vorbereiten
7. ⚡ Fehler in Fertigungsdaten oder Bauteile
nicht lieferbar
8. 🔋 Nächste Iteration
9. ↑ TOP ...
10. 📁 Projekt archivieren



Wer kennt es nicht?



- Unterschiedliche Titel, Revision, Datum auf verschiedenen Schaltplanseiten
- Uneindeutige Bezeichnungen
- DRC Errors
- Falsche oder keine KiCad-Version installiert
- Outdated PDFs von Schaltplan und Dokumentation

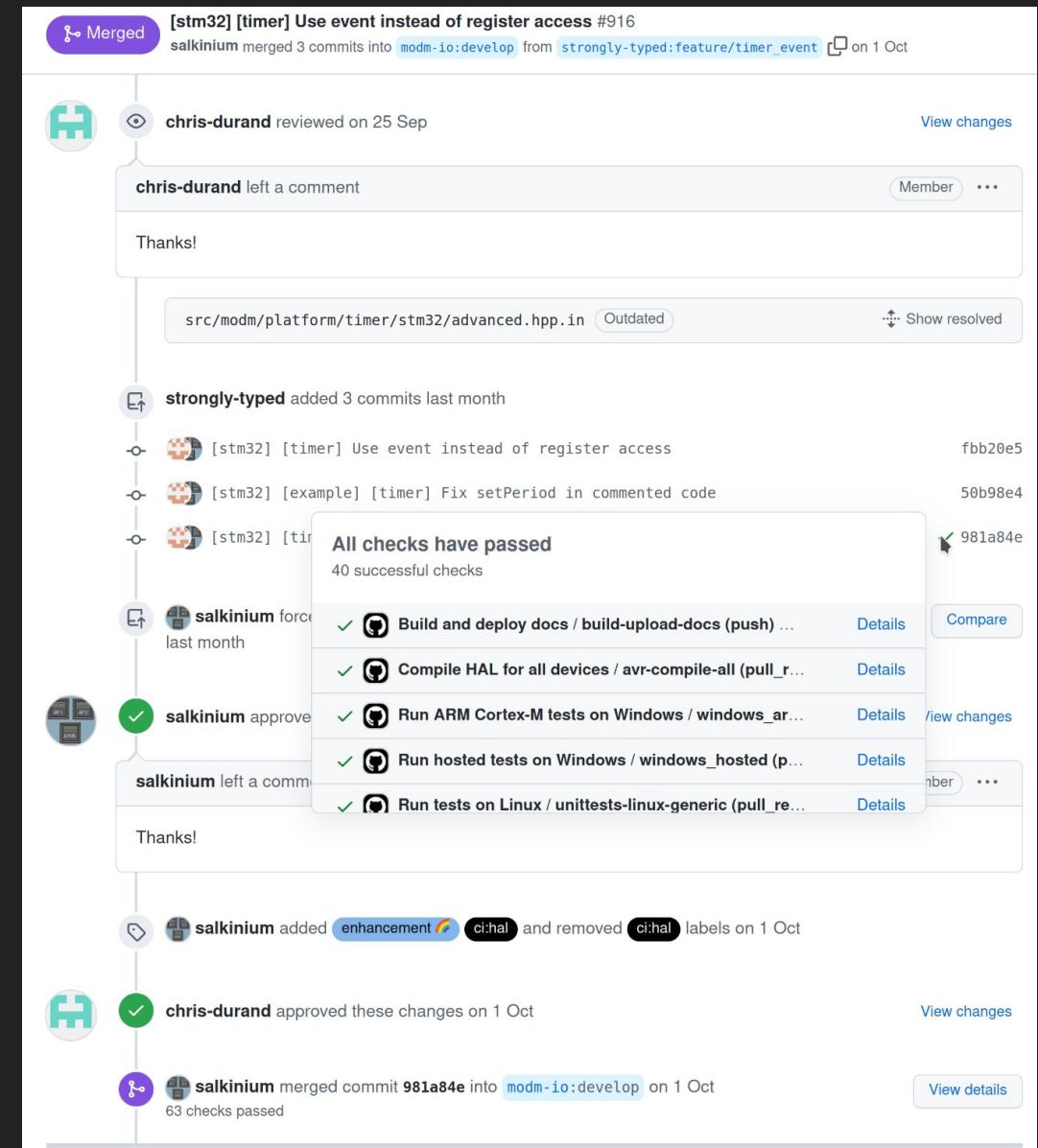
Github Actions

- Skripte bei Events (Push, Pull Request, Merge, ...) ausführen
- Umgebung: Docker

```
docker run
```

```
ghcr.io/rleh/kicad6_and_pandoc:latest
```

- ...



KiBot

<https://github.com/INTI-CMNB/KiBot>

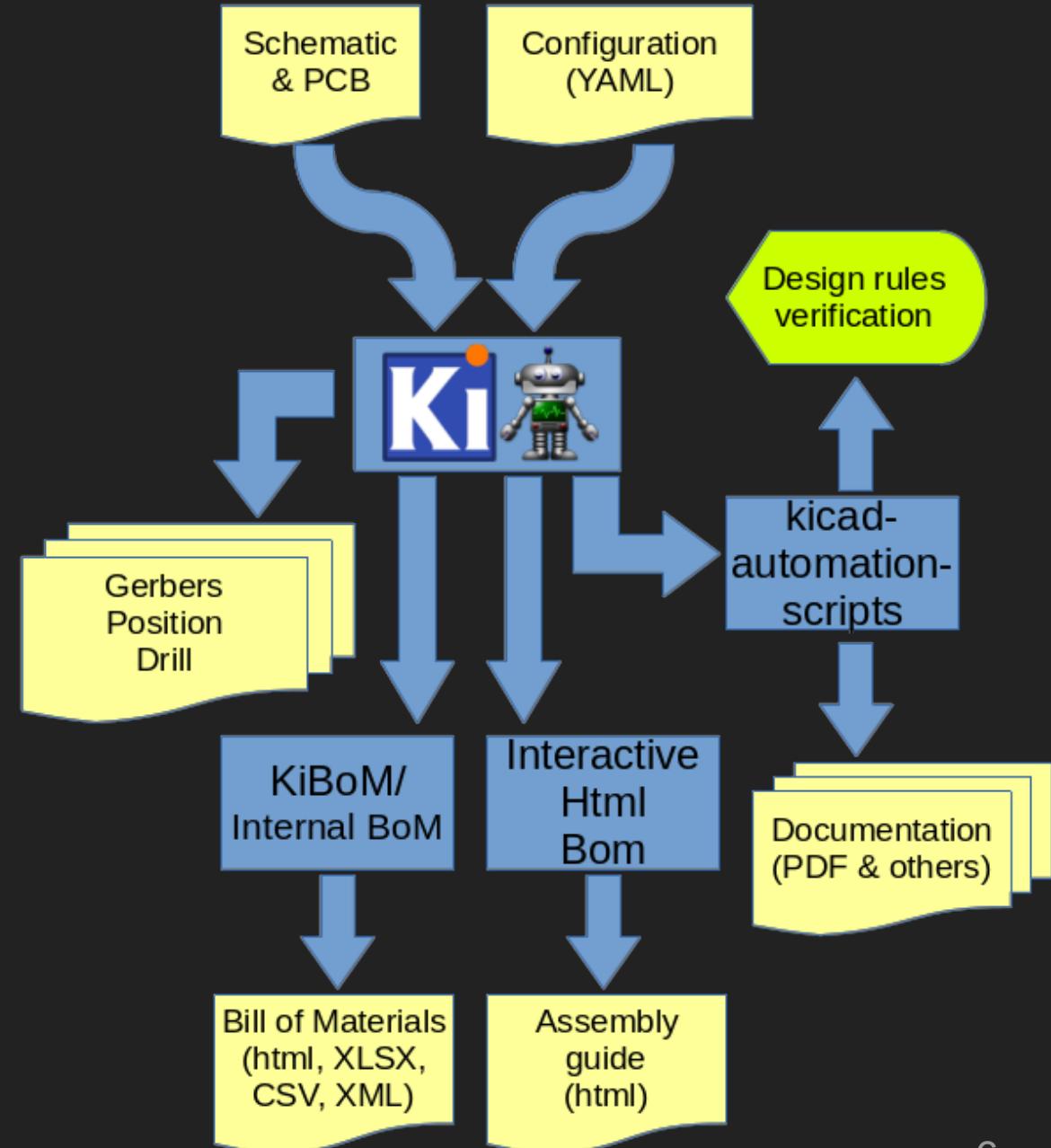
🚧 Under development, funktioniert gut!

☕ Enthalten im Docker image

`ghcr.io/rleh/kicad6_and_pandoc`

🌈 Nutzt coole KiCad plugins

- 🔗 Interactive HTML BOM
- 🔗 KiBoM





.github/workflows/kicad.yml

```
name: "Kicad Checks"

on:
  push:

jobs:
  kicad_checks:
    runs-on: ubuntu-latest
    container: ghcr.io/rleh/kicad6_and_pandoc:latest

# ...
```



Vorbereitungen

```
- name: Identify changed KiCad projects (compared to main)
  run: |
    echo PROJECTS=$(git diff --diff-filter=ACMRT \
    --name-only origin/main... | grep "pcbs/" | grep -v "pcbs/lib" | \
    cut -d/ -f2 | uniq) >> $GITHUB_ENV

- name: Exclude projects without KiBot file
  shell: bash
  run: |
    FILTERED_PROJECTS=()
    for P in $PROJECTS; do
      if [[ -f "pcbs/$P/config.kibot.yaml" ]]; then
        # This project has a valid KiBot file and will be checked
        FILTERED_PROJECTS+=($P)
      fi
    done
    echo PROJECTS=$(echo ${FILTERED_PROJECTS[@]}) >> $GITHUB_ENV
    echo PROJECT_COUNT=$(echo ${#FILTERED_PROJECTS[@]}) >> $GITHUB_ENV
```



Revisionsvergleich

```
- name: Check revision numbers identical
  if: env.PROJECT_COUNT >= 1
  run: |
    for P in $PROJECTS
    do
      (cd pcbs/$P && ../../github/res/kicad_revision_check.sh)
    done
```



kicad_revision_check.sh

```
#!/bin/bash
return_code=0

echo "All revision strings have to be identical. Checking..."
n=$(grep -hR "^\s*(rev" | cut -c 11- | rev | cut -c 3- | rev | uniq | wc -l)
if [ $n -ne 1 ]; then
    echo "ERROR: Multiple different revision strings detected ($n)"
    ((return_code+=10))
fi

# ...

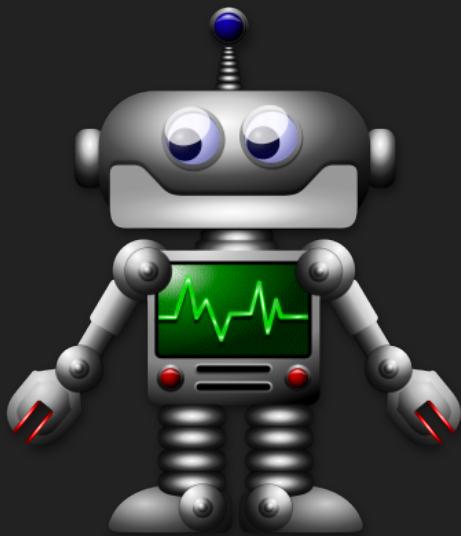
exit $return_code
```

KiBot

```
- name: Print schematic
  if: env.PROJECT_COUNT >= 1
  run: |
    for P in $PROJECTS
    do
      echo $P
      (cd pcbs/$P && kibot -v --skip-pre all print_sch)
    done

- name: Gerber export
  if: env.PROJECT_COUNT >= 1
  run: |
    for P in $PROJECTS
    do
      (cd pcbs/$P && kibot -v --skip-pre all gerbers gerber_drills)
      (cd pcbs/$P/gerber && zip ..../gerber.zip *)
    done
```

From the KiPlot realm comes ...



... a Robot to easier your KiCad journey

KiBot Export Möglichkeiten

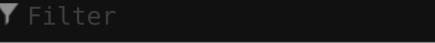
- PDF von Schaltplan und Layout
- Gerber/Drill, Pick & place
- Verschiedene Varianten für BOM
- Interactive HTML zum Bestücken
- 3D Modell (STEP, o.ä.)
- Renderings
- ...

→ [KiBot Dokumentation](#)

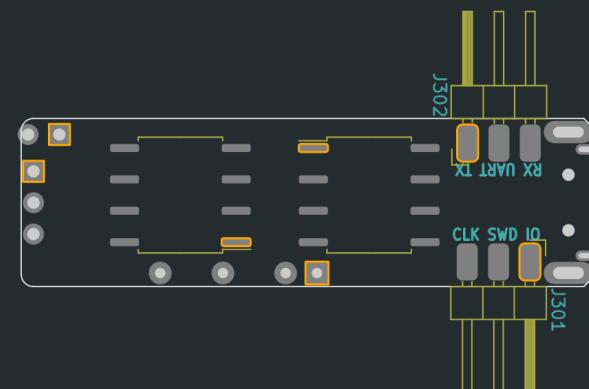
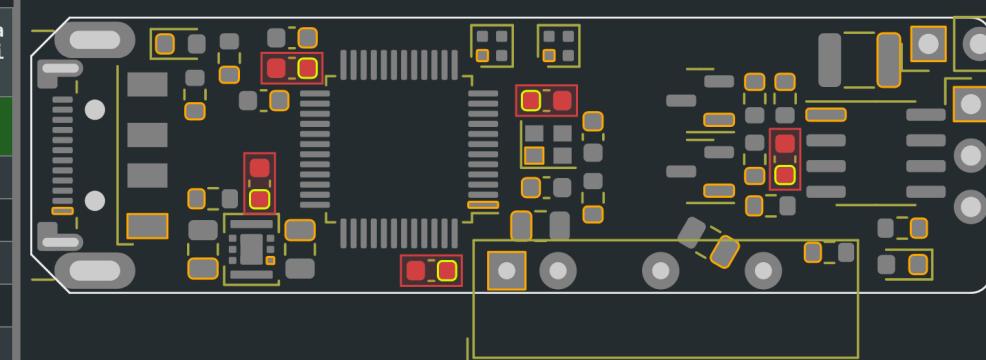
Interactive HTML BOM plugin for KiCad

usb2fdcan

Rev: 2022-10-01 20:54:46

	Sou rced	Pla ced	References	Value	Footprint	Qua ntity
1	<input type="checkbox"/>	<input type="checkbox"/>	C303, C307, C308, C309, C401	100n	C_0603_1608Metric	5
2	<input type="checkbox"/>	<input type="checkbox"/>	C301, C302	9p	C_0603_1608Metric	2
3	<input type="checkbox"/>	<input type="checkbox"/>	C305, C306	1u	C_0603_1608Metric	2
4	<input type="checkbox"/>	<input type="checkbox"/>	C601, C602	4.7u	C_0805_2012Metric	2
5	<input type="checkbox"/>	<input type="checkbox"/>	C603, C604	10u	C_0805_2012Metric	2
6	<input type="checkbox"/>	<input type="checkbox"/>	R101, R201, R202, R203, R205	1k	R_0603_1608Metric	5
7	<input type="checkbox"/>	<input type="checkbox"/>	R204, R501, R601	5.1k	R_0603_1608Metric	3
8	<input type="checkbox"/>	<input type="checkbox"/>	R401	0	R_0603_1608Metric	1
9	<input type="checkbox"/>	<input type="checkbox"/>	R402	120	R_1210_3225Metric	1
10	<input type="checkbox"/>	<input type="checkbox"/>	R602	82	R_0603_1608Metric	1
11	<input type="checkbox"/>	<input type="checkbox"/>	D101, D102	Inolux_IN-PI15TAT5R5G5B	LED_Inolux_IN-PI15TAT5R5G5B_1.5x1.5mm_P0 .65mm	2
12	<input type="checkbox"/>	<input type="checkbox"/>	D103, D601	blue	LED_0603_1608Metric	2
13	<input type="checkbox"/>	<input type="checkbox"/>	U201, U202	6N137S-TA1-L	SOP-8_6.62x9.15mm_P2.54mm	2
14	<input type="checkbox"/>	<input type="checkbox"/>	U301	STM32G431CB T6	LQFP-48_7x7mm_P0.5mm	1
15	<input type="checkbox"/>	<input type="checkbox"/>	U401	TLE9251VSJ	SOIC-8_3.9x4.9mm_P1.27mm	1



Bauteil-Matching

- Automatisierte Zuordnung von Bauteilen in Bauteildatenbank
 - PartsBox oder ? PartKeepr
- Heuristik für Standard-Bauteile (Widerstände, Kondensatoren, LEDs, ...)
- Stringmatching: MPN im *Value*-Feld
- Report als PDF/HTML/... generieren
- Merge: Projekt/BOM in Bauteildatenbank automatisiert anlegen

Bauteil-Matching

🔥 KiCad Python-Scripting

```
#!/usr/bin/env python3
import pcbnew

filename = "test.kicad_pcb"
board: pcbnew.BOARD = pcbnew.LoadBoard(filename)

components = board.GetFootprints()
rev = str(board.GetTitleBlock().GetRevision())

# ...
```

Bauteildatenbank

The screenshot shows a web-based application for managing electronic components. At the top, there's a navigation bar with links for PartsBox, Parts, Storage, Projects, Purchasing, Reports, News (marked as New), and user account information. Below the navigation is a toolbar with icons for Parts, Stock, Create, Scan, Selected..., and a search field for LED.

The main area is a table listing components:

Type	Part	Description	Footprint	Stock	Last accessed	Where is it?
>	LED blue 20mA 0402		0402	82	16.01.2022 20:11	- Diodes (82)
>	LED blue 20mA 0603		0603	0	-	-
>	LED green 20mA 0402		0402	97	22.08.2021 04:41	- Diodes (97)
>	LED green 20mA 0603		0603	0	-	-
>	LED green 20mA 0603 FC-DA1608UGK-520D10	LCSC C84267	0603	47 26	17.08.2021 23:31	- Reels C (4726)
>	LED red 20mA 0402		0402	87	16.01.2022 20:11	- Diodes (87)
>	LED red 20mA 0603		0603	0	-	-
>	LED yellow 20mA 0402		0402	10	27.10.2019 21:06	- Diodes (10)
>	LED yellow 20mA 0603		0603	0	-	-
>	LP5036RJVR	36-Channel I2C constant current RGB LED driver 46-VQFN-40 to 125		2	30.10.2019 20:33	- ICs G-L (2)

Wichtig: Einheitliche Konvention für Bauteilbezeichnungen
Z.B.: LED {color} {current rating} {package}

Bauteil-Matching 🔥🌈

```
for c in components:
    reference = str(c.GetReference())
    value = str(c.GetValue())
    footprint = str(c.GetFPID().GetLibItemName())
    package = re.search(r"(?P<package>0402|0603|0805|1206)", footprint)
    if package is not None:
        package = package.group("package")

    #
    # ...

    if "LED" in c.footprint:
        color = re.search(r"(?P<color>red|green|blue|yellow|orange)", value.lower())
        if package and color:
            color = color.group("color")
            return partsbox_search_by_name(f"LED {color} 20mA {package}")
    if re.search(r'C\d+', reference):
        #
        # ...
    #
else:
    # fallthrough case: MPN specified in value field
    return partsbox_search_by_name(value)

# ...
```

Vielen Dank für eure Aufmerksamkeit!

Fragen?

Backup

KiCad Libraries

- 🥇 Qualitätsstandard übertrifft kommerzielle Bibliotheken um Größenordnungen
- 📈 Aktuell ≥ 900 offene Merge-Requests für Symbole und Footprints 📚
- 👎 Wechsel vom Github zu Gitlab am 01.10.2020
→ ≥ 500 Pull-Requests faktisch beerdigt

Review-Prozess

- 👥 Zu wenig Reviewer
- ⚙️ Viel manueller Aufwand
- Mehr Automatisierung

Wie können *wir* das langfristig verbessern? 🤔



This repository has been archived by the owner. It is now read-only.

KiCad / [kicad-symbols](#)

[Public archive](#)[Notifications](#)[Fork 793](#)[Star 654](#)[Code](#)[Issues 229](#)[Pull requests 215](#)[Actions](#)[Projects](#)[Security](#)[Insights](#) is:pr is:open[Labels 28](#)[Milestones 1](#)[215 Open](#) ✓ 2,395 Closed[Author ▾](#)[Label ▾](#)[Projects ▾](#)[Milestones ▾](#)[Reviews ▾](#)[Assignee ▾](#)[Sort ▾](#)

Added symbols for TE PE0xxFxx bistable latching relays. Replaces PR #2649. ✘ Abandoned



3

[Addition](#) [Pending footprint](#) [Re-push](#)

#2946 opened on 29 Aug 2020 by amil109

Added DE119 from DE152 ✘ [Addition](#) [Pending footprint](#)



2

#2936 opened on 26 Aug 2020 by valerionew

Added DE152 from DE122 ✘ [Addition](#) [Pending footprint](#)



13

#2932 opened on 25 Aug 2020 by valerionew

Added some TM* led and lcd driver symbols ✓ [Abandoned](#) [Addition](#) [Pending footprint](#)



11

#2924 opened on 22 Aug 2020 by purdeaandrei

Create IGBT Module library ✘ [Addition](#) [Pending footprint](#)



8



This repository has been archived by the owner. It is now read-only.

KiCad / kicad-footprints Public archive

NotificationsFork 745Star 571[Code](#)[Issues 133](#)[Pull requests 323](#)[Actions](#)[Projects](#)[Wiki](#)[Security](#)[Insights](#) is:pr is:open[Labels 23](#)[Milestones 1](#)[323 Open ✓ 1,830 Closed](#)[Author ▾](#)[Label ▾](#)[Projects ▾](#)[Milestones ▾](#)[Reviews ▾](#)[Assignee ▾](#)[Sort ▾](#)

[Add 5x5mm VQFN-32 with a nominal 3.15x3.15mm EP ✓](#) Addition

#2508 opened on 1 Oct 2020 by justyn 5 tasks done

[3](#)

[WDFN-8-1EP_8x6mm_P1.27mm_EP6x4.8mm with and without Thermal Pads added. ✓](#) Addition

[3](#)Pending reviewer

#2506 opened on 30 Sep 2020 by serkanxselcuk 1 of 5 tasks



[Add IDC 2x05 edge mount ✗](#) Addition Pending reviewer

[3](#)

#2497 opened on 27 Sep 2020 by dbozec 5 tasks done

[Relay_DPDT_Panasonic_AGQ2x0Axx ✓](#) Addition Pending reviewer

[3](#)

#2492 opened on 25 Sep 2020 by rleh 5 tasks done



[SoC PolarFire: Add footprints for 484 & 1152 RGA](#) Addition

[15](#)

[About GitLab](#)[Pricing](#)[Talk to an expert](#) Search GitLab

/

[Sign up now](#)[Login](#)[KiCad](#) > [KiCad Libraries](#) > [KiCad Symbols](#) > **Merge requests**[Open 476](#)[Merged 2,196](#)[Closed 471](#)[All 3,143](#) [Subscribe to RSS feed](#)[Recent searches](#) Search or filter results...[Created date](#)**Add NC hidden pins to ATECC608A** 1 of 7 checklist items completed

!3631 · created 7 hours ago by Frank Severinsen

8✓ 1 left 0

updated 7 hours ago

Add MagnTek MT6816 6 of 6 checklist items completed

!3630 · created 1 day ago by Max Mustermann

8✓ 1 left 0

updated 1 day ago

Add MT6701 symbol 6 of 6 checklist items completed

!3629 · created 1 day ago by Max Mustermann

8✓ 1 left 0

updated 1 day ago

Add ADG658YCPZ: Analog Devices +3 V/+5 V/±5 V CMOS 8-Channel Analog MUX

!3628 · created 1 day ago by Armin Schoisswohl

8✓ 1 left 0

updated 19 hours ago

Fix duplicate pin number 6 for 4508

!3627 · created 2 days ago by Jan Alexander

8✗ 1 left 0

updated 2 days ago



[About GitLab](#)[Pricing](#)[Talk to an expert](#) Search GitLab

/

[Sign up now](#)[Login](#)[KiCad](#) > [KiCad Libraries](#) > [KiCad Footprints](#) > **Merge requests****Open** 488

Merged 1,683

Closed 398

All 2,569

[Subscribe to RSS feed](#)

Recent searches ▾

Search or filter results...

Created date ▾

**Add all missing packages for STM32 microcontrollers**!2924 · created 3 days ago by Nein Seg Addition✓ 8v1 left 0

updated 3 days ago

Added IDC SMD connectors, script generated

!2923 · created 4 days ago by Carlos Nieves

✓ 8v1 left 0

updated 4 days ago

Added TDK VLS6045EX and VLS6045AF

!2920 · created 1 week ago by Carlos Nieves

✓ 8v1 left 0

updated 1 week ago

Added HDGC 0.5K-A-nPBF FPC connectors generated by script

!2919 · created 1 week ago by TiZed Tz

✓ 8v1 left 0

updated 1 week ago

Added LED PLCC-2 3x2mm

!2918 · created 1 week ago by Carlos Nieves

✓ 8v1 left 0

updated 1 week ago

»

Idee: KiCad Library Hackathon

-  Tooling verbessern
-  KiCad Nutzer als Library-Maintainer gewinnen
-  Backlog an offenen PRs mit Symbolen und Footprints abarbeiten
-  An mehreren Orten?

Ende 