

# KiCad – Eine Einführung

treff.DARC 05.10.2021

Ein Vortrag von DF1HPK  
om@df1hpk.de

# DF1HPK

- Jahrgang 1987
- QRL: Softwareentwickler (Schwerpunkt Java)
- Funkamateurl  
  - 2009 (Klasse E)
  - 2011 (Klasse A)
  - DOK C28
- <https://df1hpk.de>



# Inhalt

- Was ist KiCad
- KiCad Theorie
  - Vom Schaltplan...
  - ...zum Platinenlayout
  - Eigene Bauteile
- KiCad Praxis
  - LM1085 – Variabler Spannungsregler
  - Ein eigenes Symbol und Footprint erstellen
- Quellen / Weiterführende Informationen

# Was ist KiCad

„A Cross Platform and Open Source Electronics Design Automation Suite“

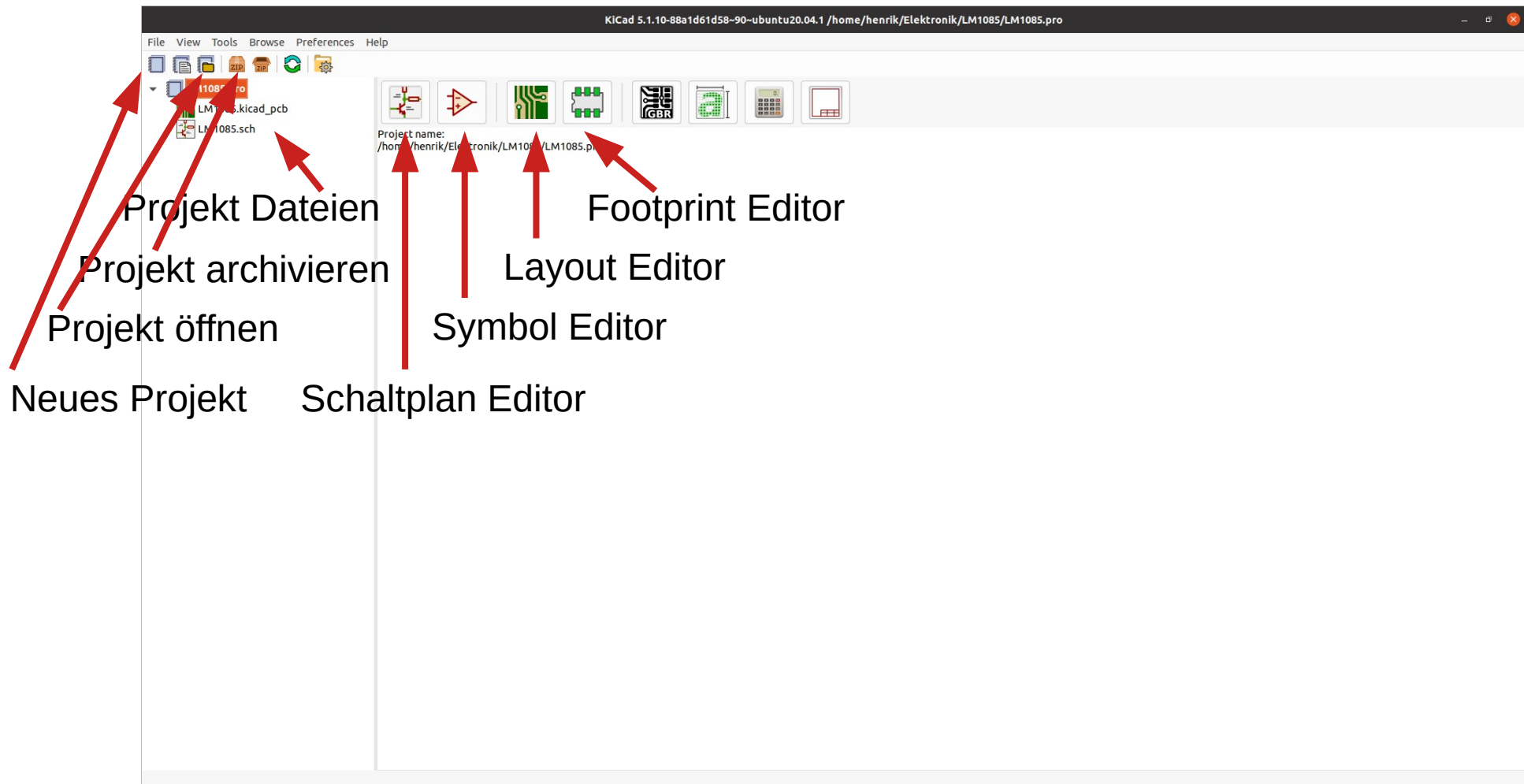
- Cross Platform
  - Linux
    - u.a. Ubuntu, Debian, openSUSE, FreeBSD
  - macOS
  - Windows
- Open Source
  - <https://www.kicad.org/download/source/>
- Electronics Design Automation Suite
  - Schaltplan
  - Platinen Layout (inkl. 3D Viewer)
  - Spice Simulation

# KiCad Theorie – Vom Schaltplan...

- Screenshots
  - KiCad 5.1.10
  - Ubuntu 20.04

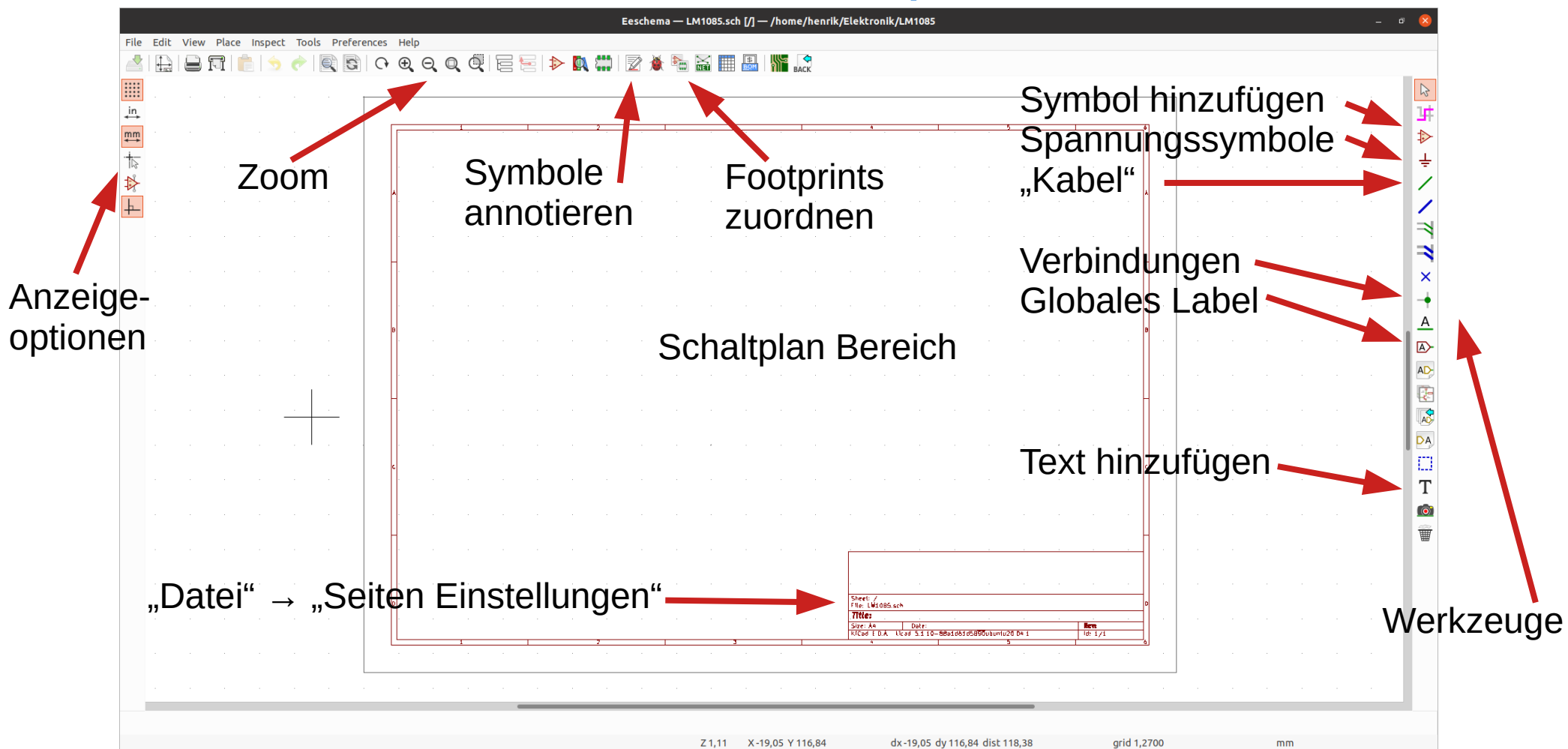
# KiCad Theorie – Vom Schaltplan...

## Startfenster



# KiCad Theorie – Vom Schaltplan...

## Eeschema - Schaltplanel editor



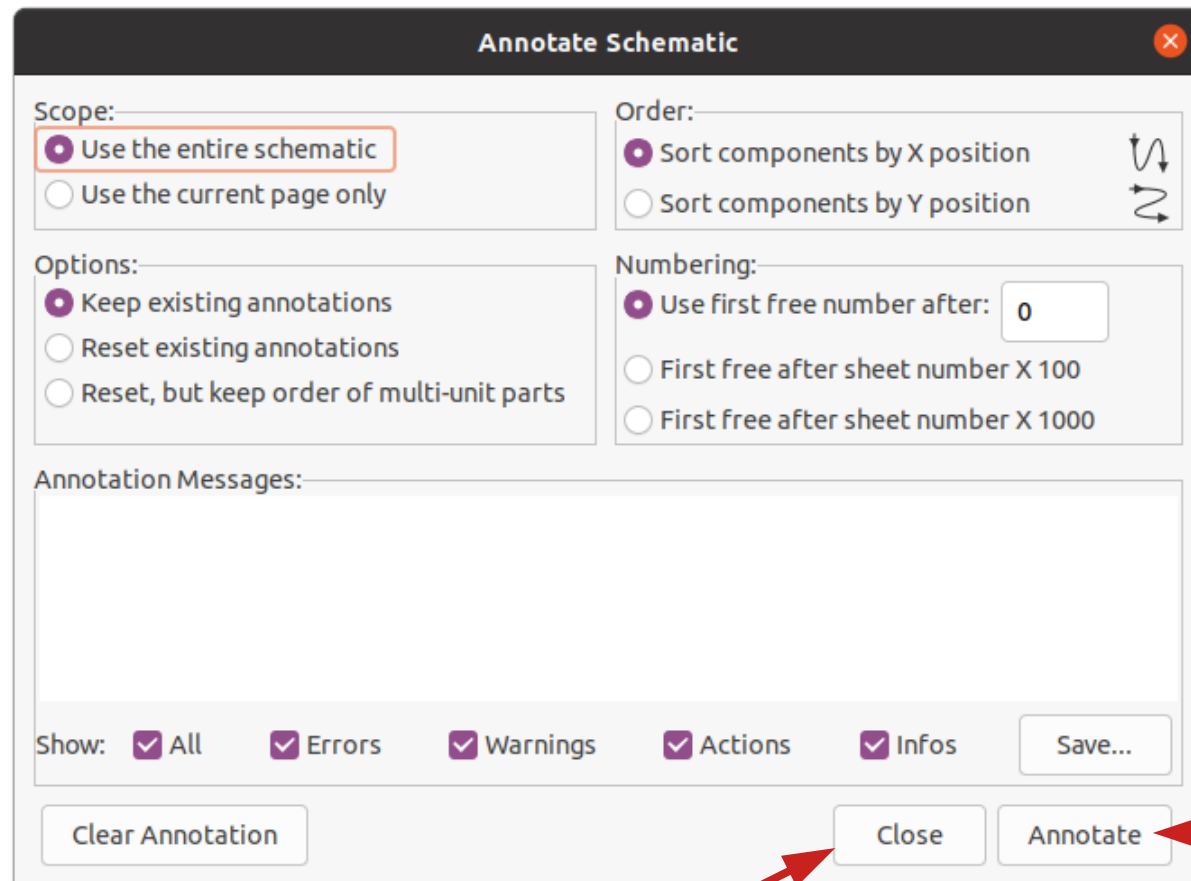
# KiCad Theorie – Vom Schaltplan...

- **Wichtige Tasten**
  - [ESC] = Beende Aktion (z.B. Symbolplatzierung)
  - [Entf] = Symbol löschen
  - [r] = Symbol im Uhrzeigersinn rotieren
  - [x] = Spiegelung an x-Achse
  - [y] = Spiegelung an y-Achse
  - [v] = Wert editieren
  - [c] = Symbol kopieren
  - [m] = Symbol bewegen



# KiCad Theorie – Vom Schaltplan...

## Eeschema – Schaltplaneditor – Symbole annotieren

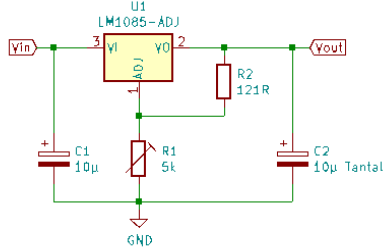
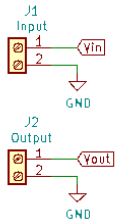


1. annotieren

2. schließen

# KiCad Theorie – Vom Schaltplan...

## Eeschema – Schaltplaneditor – Footprints zuordnen



Footprint Bibliotheken

Assign Footprints

File Preferences Help

Footprint Libraries

- Arduino
- Battery
- Button\_Switch\_Keyboard
- Button\_Switch\_SMD
- Button\_Switch\_THT
- Buzzer\_Beeper
- Calibration\_Scale
- Capacitor\_SMD
- Capacitor\_Tantalum\_SMD
- Capacitor\_THT
- Connector
- Connector\_AMASS
- Connector\_Ampheno1
- Connector\_Audio
- Connector\_BarrelJack
- Connector\_Card

Symbol : Footprint Assignments

ID	Component	Value
1	C1	10µ
2	C2	10µ Tantal
3	J1	Input
4	J2	Output
5	R1	5k
6	R2	121R
7	U1	LM1085-ADJ

Filtered Footprints

- 1 Capacitor\_SMD:CP\_Elec\_3x5.3
- 2 Capacitor\_SMD:CP\_Elec\_3x5.4
- 3 Capacitor\_SMD:CP\_Elec\_4x3
- 4 Capacitor\_SMD:CP\_Elec\_4x3.9
- 5 Capacitor\_SMD:CP\_Elec\_4x4.5
- 6 Capacitor\_SMD:CP\_Elec\_4x5.3
- 7 Capacitor\_SMD:CP\_Elec\_4x5.4
- 8 Capacitor\_SMD:CP\_Elec\_4x5.7
- 9 Capacitor\_SMD:CP\_Elec\_4x5.8
- 10 Capacitor\_SMD:CP\_Elec\_5x3
- 11 Capacitor\_SMD:CP\_Elec\_5x3.9
- 12 Capacitor\_SMD:CP\_Elec\_5x4.4
- 13 Capacitor\_SMD:CP\_Elec\_5x4.5
- 14 Capacitor\_SMD:CP\_Elec\_5x5.3
- 15 Capacitor\_SMD:CP\_Elec\_5x5.4
- 16 Capacitor\_SMD:CP\_Elec\_5x5.7

Filtered by key words (CP\_\*), pin count (2): 219  
Description: CP, Axial series, Axial, Horizontal, pin pitch= 5mm, , length\*diameter=10\*4.5mm^2, Electrolytic Capacitor, , http://www.vishay.com/docs/8325/021asm.pdf; Key w  
Library location: /usr/share/kicad/modules/Capacitor\_THT.pretty

Apply, Save Schematic & Continue    Abbrechen    OK

Footprint Zuweisungen

1. Footprints zuordnen
2. Speichern
3. Verlassen

„Passende“ Footprints

# KiCad Theorie - ...zum Platinenlayout

Footprint Update aus Schaltplan  
Design Rule Check

## Pcbnew - Layouteditor

The screenshot shows the KiCad PCBnew layout editor interface. The main workspace is a dark grid with a red rectangular footprint. Red arrows point from text labels to various parts of the interface:

- Leiterbahnbreite** (Track Width): Points to the track width settings in the top toolbar.
- Via Maße** (Via Dimensions): Points to the via dimensions settings in the top toolbar.
- Rastermaß** (Grid Size): Points to the grid size settings in the top toolbar.
- Footprint**: Points to the footprint selection icon in the top toolbar.
- Leiterbahn** (Track): Points to the track drawing icon in the top toolbar.
- Fläche** (Area): Points to the area drawing icon in the top toolbar.
- Grafische Linie** (Graphic Line): Points to the graphic line drawing icon in the top toolbar.
- Text**: Points to the text drawing icon in the top toolbar.
- Anzeigeoptionen** (Display Options): Points to the vertical toolbar on the left side.
- Werkzeuge** (Tools): Points to the vertical toolbar on the right side.
- Layer Auswahl** (Layer Selection): Points to the Layers Manager panel on the right side.

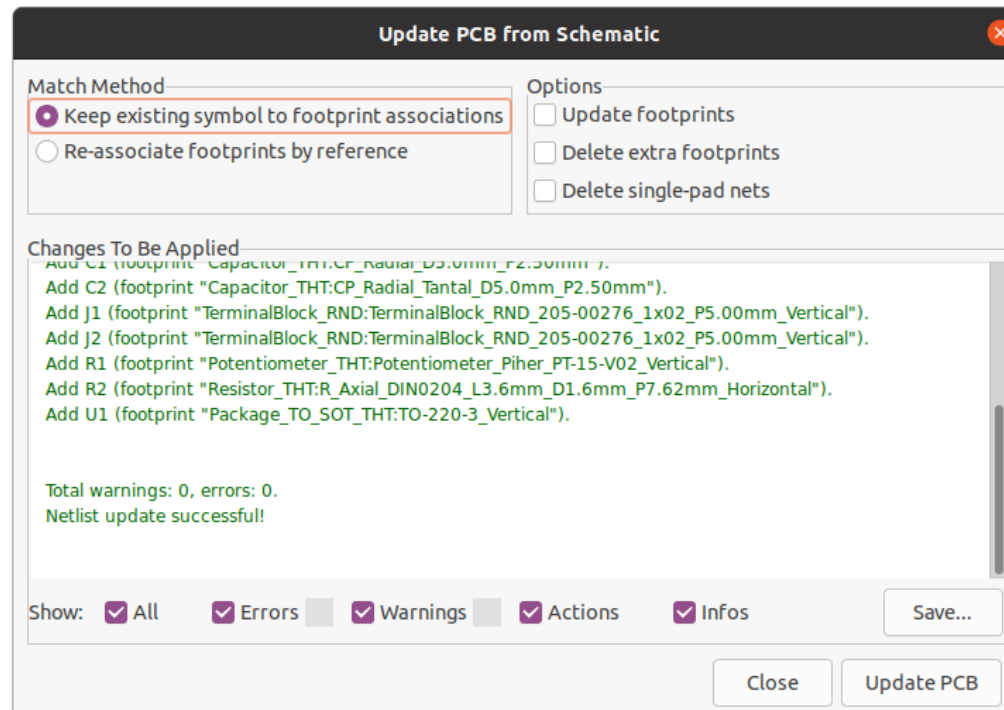
The Layers Manager panel on the right shows a list of layers with checkboxes and color swatches:

Layer	Item
F.Cu	F.Cu
B.Cu	B.Cu
F.Adhes	F.Adhes
B.Adhes	B.Adhes
F.Paste	F.Paste
B.Paste	B.Paste
F.Silks	F.Silks
B.Silks	B.Silks
F.Mask	F.Mask
B.Mask	B.Mask
Dwgs.User	Dwgs.User
Cmts.User	Cmts.User
Eco1.User	Eco1.User
Eco2.User	Eco2.User
Edge.Cuts	Edge.Cuts
Margin	Margin
F.CrtYd	F.CrtYd
B.CrtYd	B.CrtYd
F.Fab	F.Fab
B.Fab	B.Fab

The status bar at the bottom shows: Z 1,05 X 138,430000 Y 100,330000 dx 138,430000 dy 100,330000 dist 170,965 grid X 1,270000 Y 1,270000 mm

# KiCad Theorie - ...zum Platinenlayout

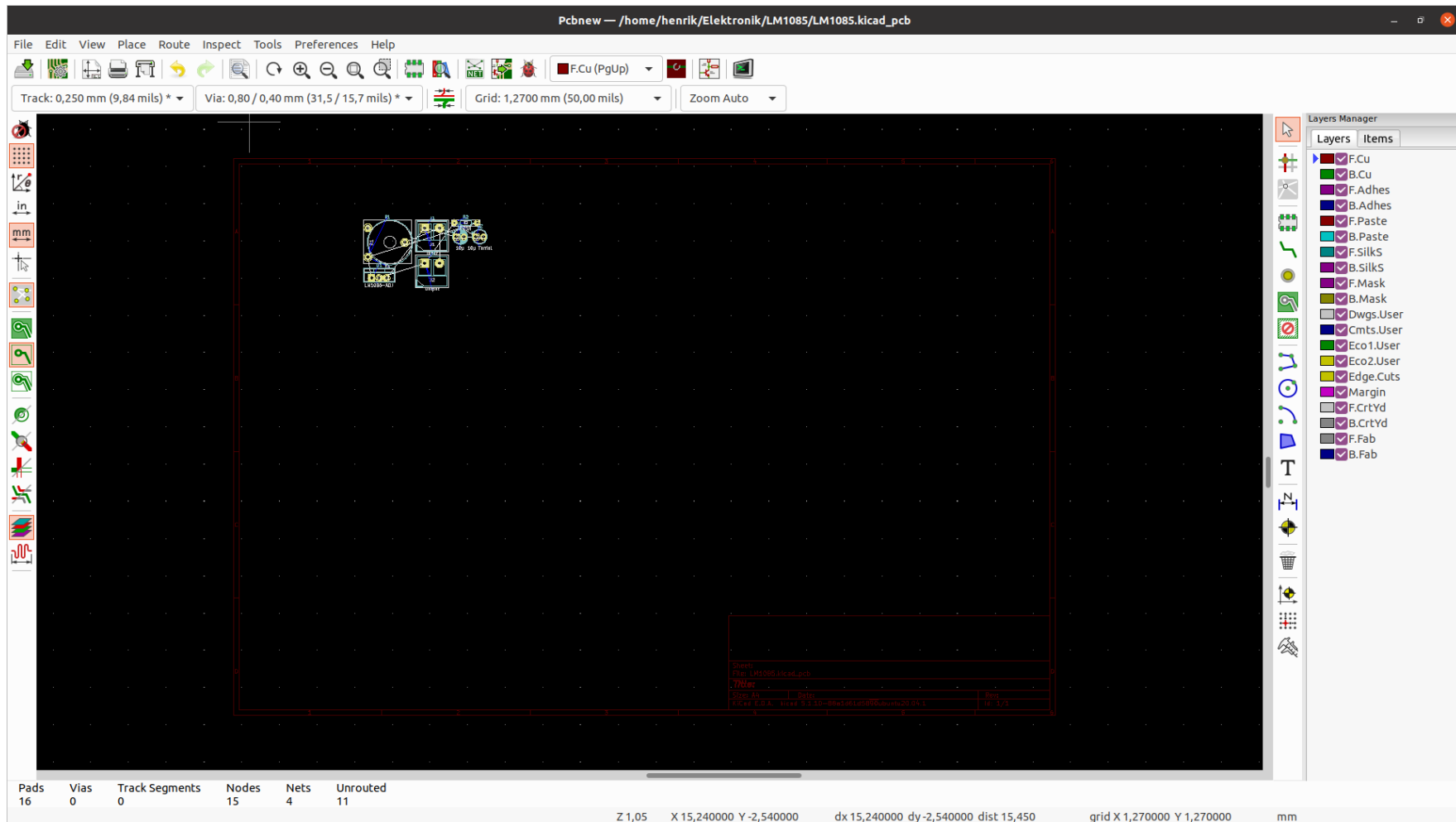
## Pcbnew - Layouteditor



1. Update PCB – Fügt die Footprints hinzu
2. Close – Rückkehr zum Editor

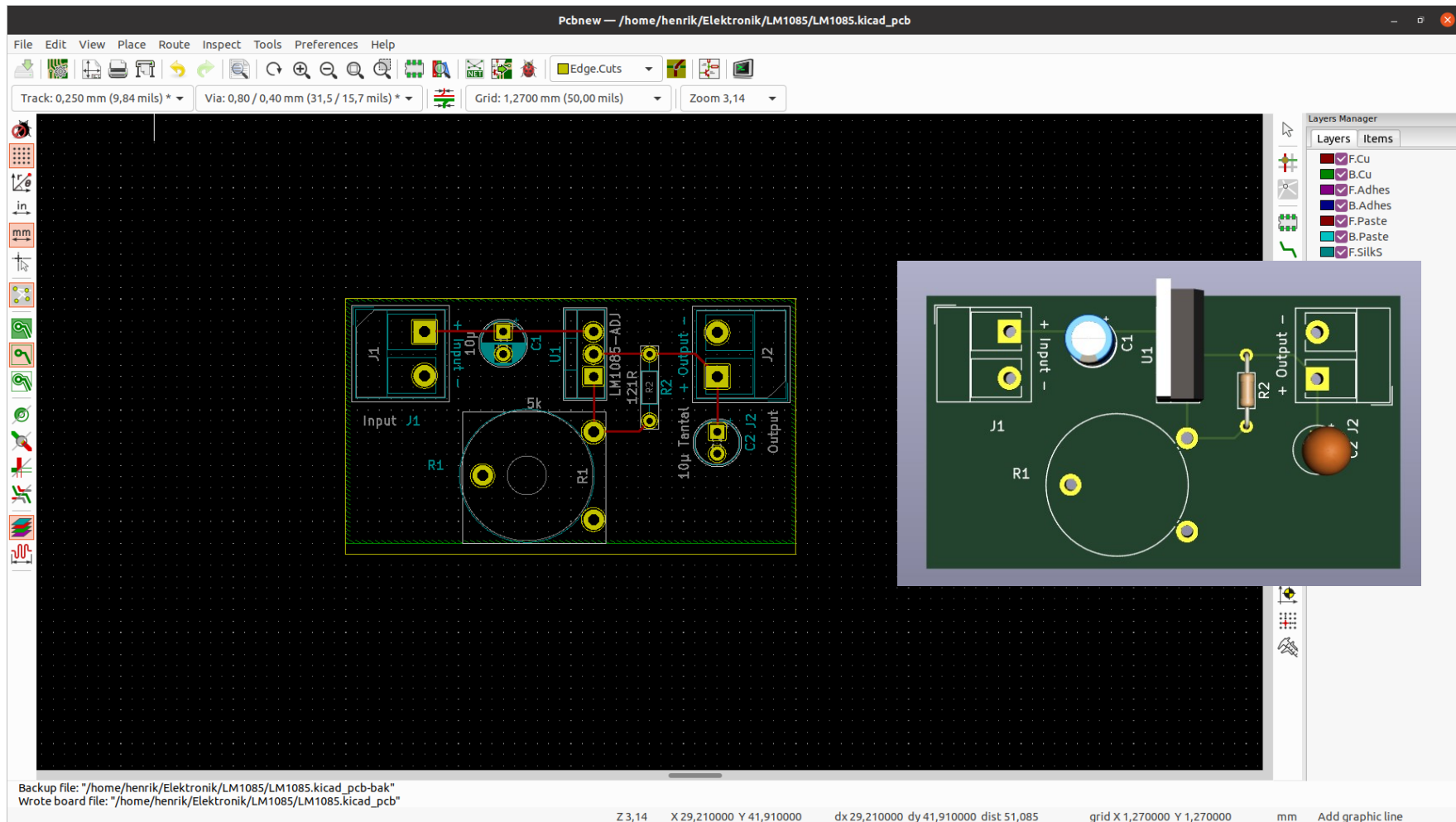
# KiCad Theorie - ...zum Platinenlayout

## Pcbnew - Layouteditor



# KiCad Theorie - ...zum Platinenlayout

## Pcbnew - Layouteditor

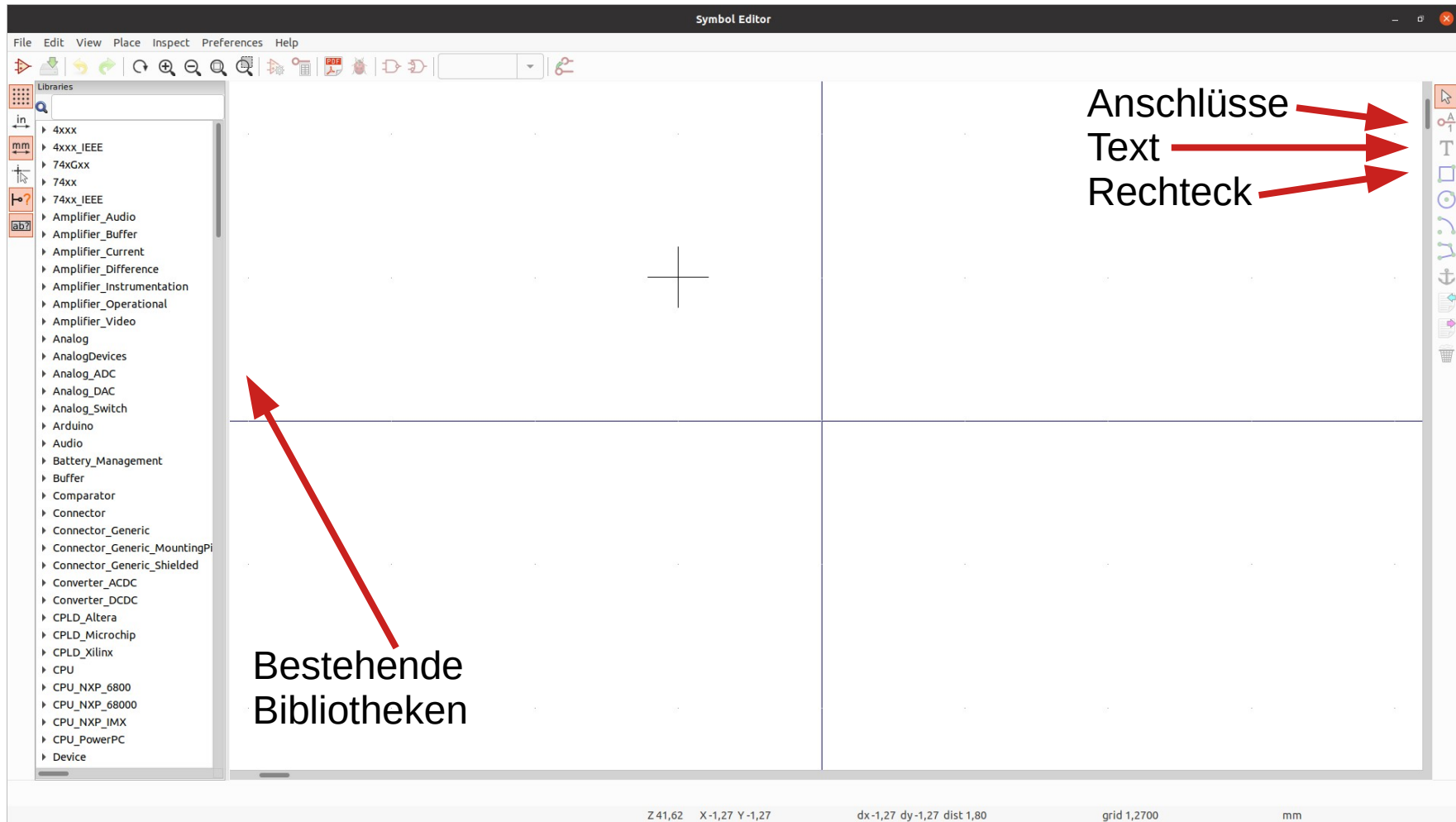


# KiCad Theorie – Eigene Bauteile

Anzeigeoptionen

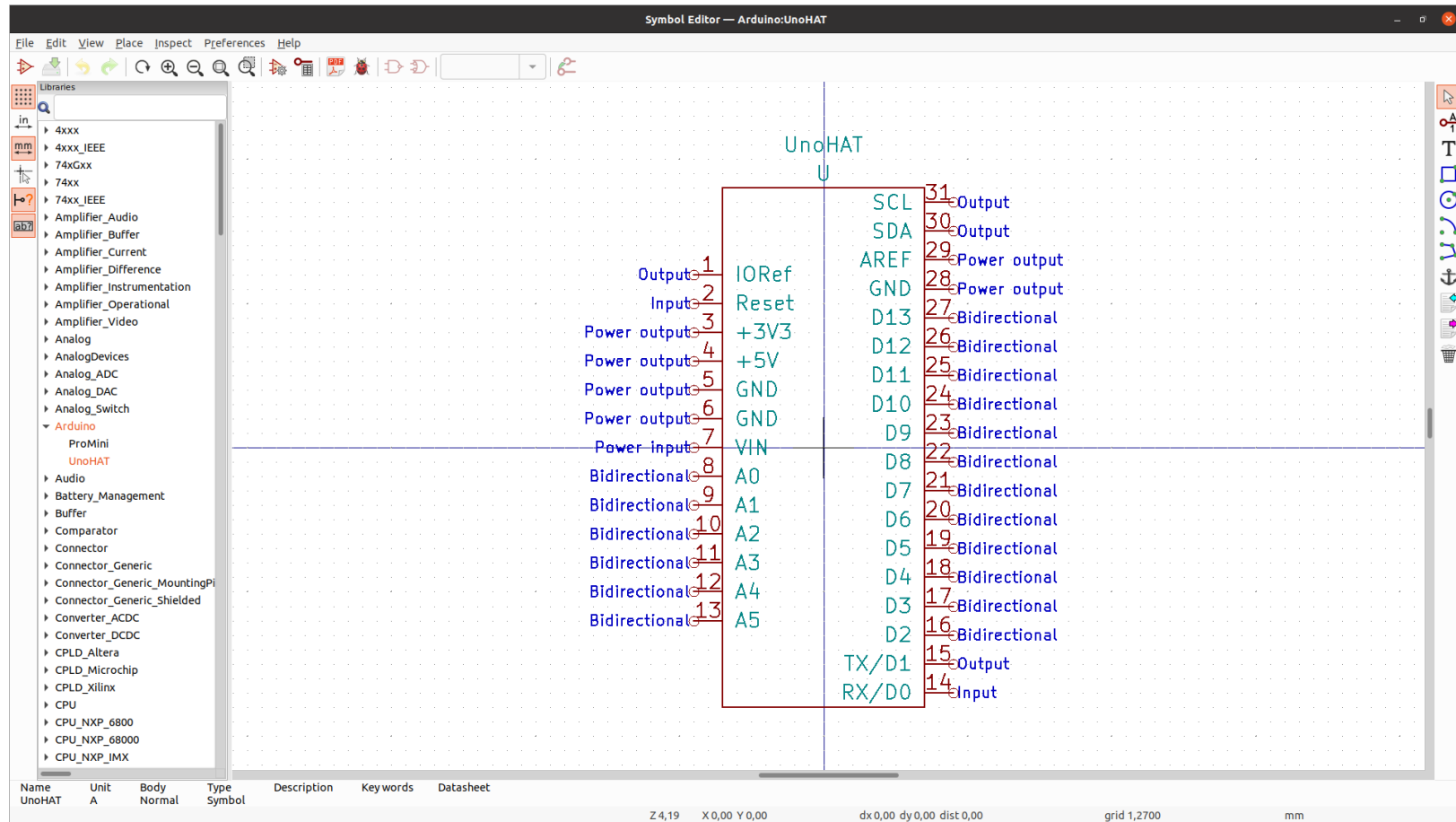
## Symbol Editor

Werkzeuge



# KiCad Theorie – Eigene Bauteile

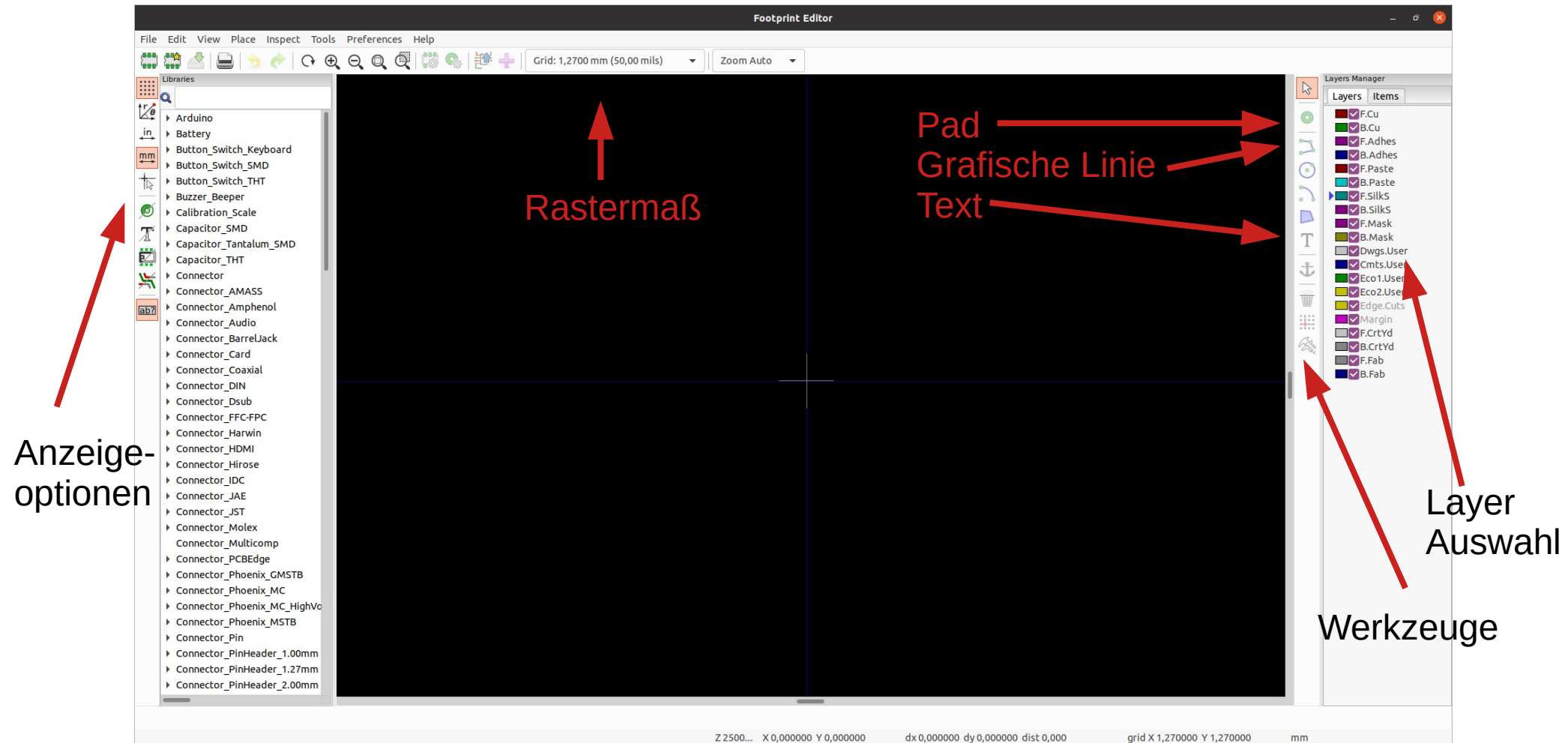
## Symbol Editor – Beispiel (Arduino UNO HAT)





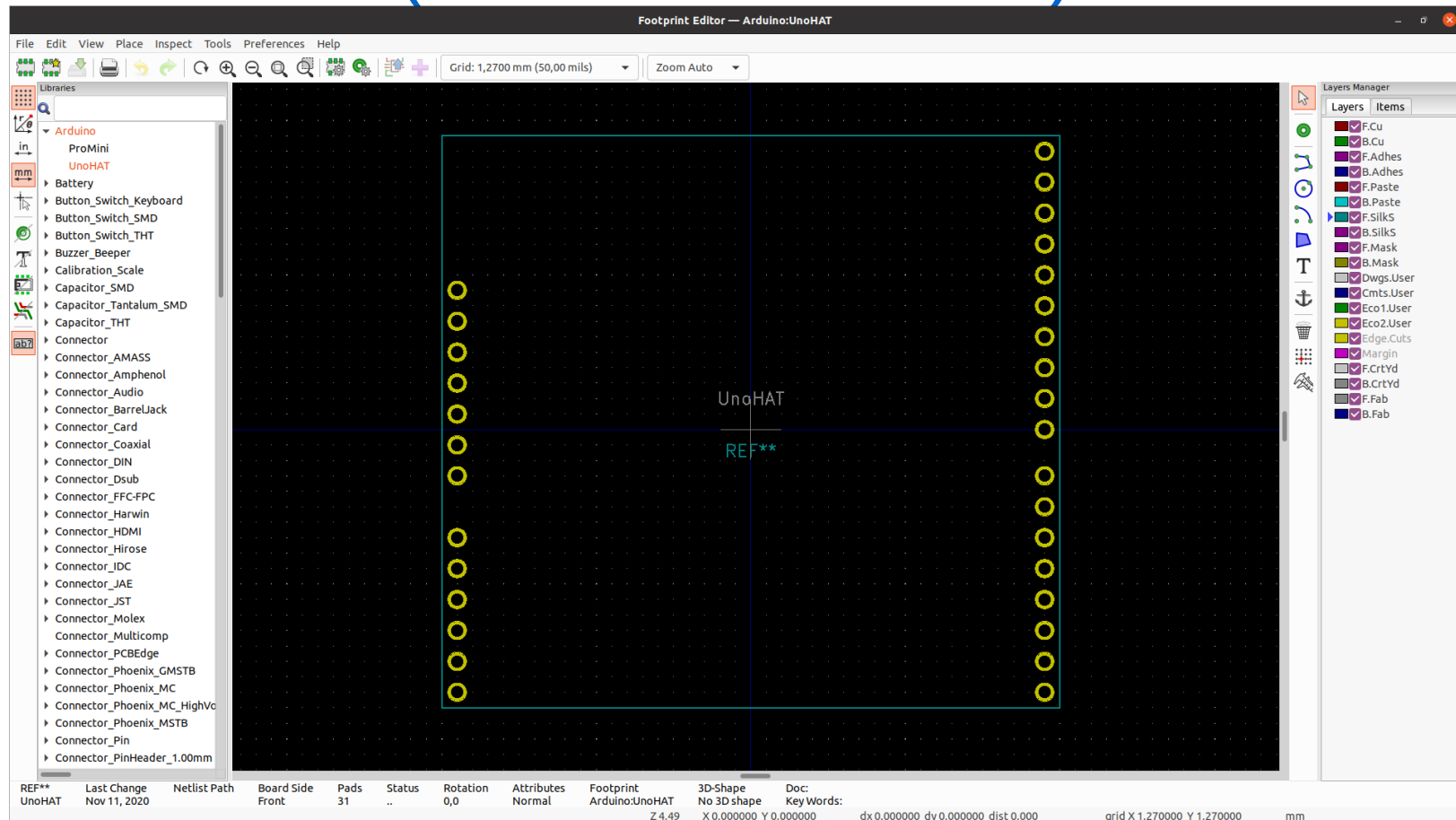
# KiCad Theorie – Eigene Bauteile

## Footprint Editor



# KiCad Theorie – Eigene Bauteile

## Footprint Editor – Beispiel (Arduino UNO HAT)



# KiCad Praxis

- Variabler Spannungsregler LM1085
  - Schaltung (In < 29V, Out 1.2 – 15V, Max 3A)
  - Layout
  - Platine online bestellen
- Ein eigenes Symbol und Footprint erstellen

# Quellen / Weiterführende Informationen

- <https://www.kicad.org>
- <https://www.ti.com/lit/ds/symlink/lm1085.pdf>
- <https://www.elektor.de/kicad-wie-ein-profi>
- <https://aisler.net/>
- <https://jlcpcb.com/>