

# KiCAD

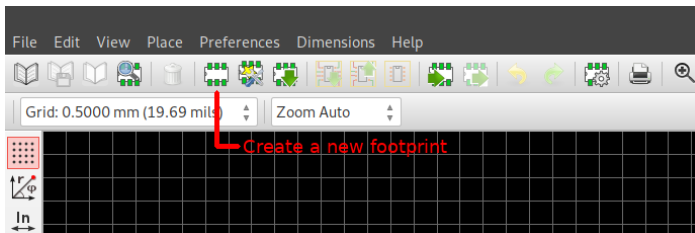
## Creating a component footprint

Katharina Fey

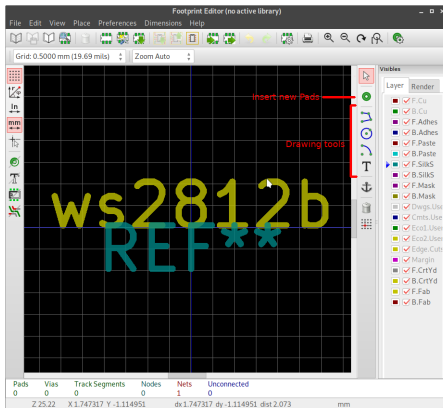
17. March 2018

# Create new footprint

- ▶ Open the footprint editor and create a new footprint
- ▶ Alternatively: load an existing library and add/ clone a footprint there

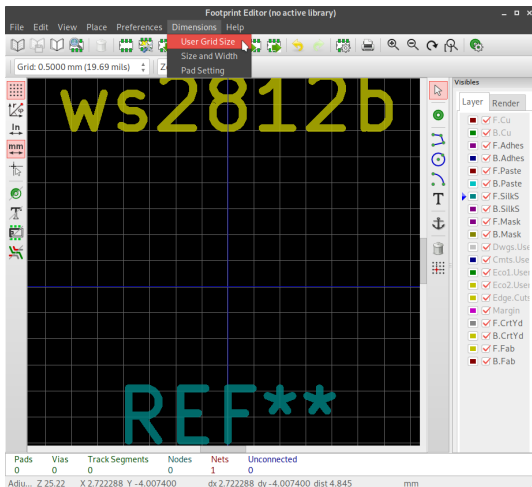


# Create new footprint



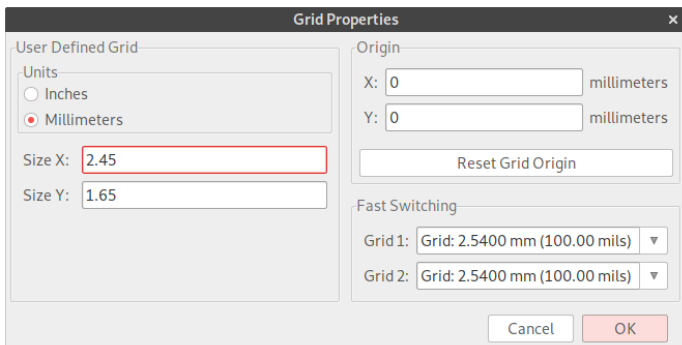
# Set custom grid

- It's easier to align pads & marks on a custom grid size



# Set custom grid

- ▶ Look at useful dimensions from the datasheet
- ▶ Half those dimensions and set them as the grid-sizes



The screenshot shows the 'Grid Properties' dialog box with the following settings:

- User Defined Grid:**
  - Units:** Millimeters (selected with a red dot, while Inches is unselected).
  - Size X:** 2.45
  - Size Y:** 1.65
- Origin:**
  - X:** 0 millimeters
  - Y:** 0 millimeters
  - Reset Grid Origin:** A button to reset the origin values.
- Fast Switching:**
  - Grid 1:** Grid: 2.5400 mm (100.00 mils) [dropdown arrow]
  - Grid 2:** Grid: 2.5400 mm (100.00 mils) [dropdown arrow]

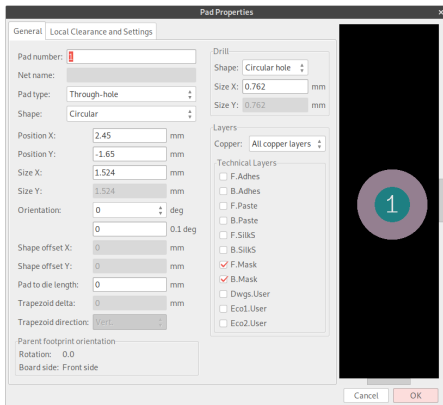
At the bottom right, there are 'Cancel' and 'OK' buttons.

# Create & customise Pads

- ▶ Select the grid, then select the pad-tool and place a pad
- ▶ Hover over the pad and press 'e' to edit it's dimensions

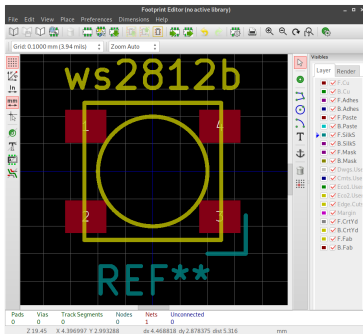
# Create & customise Pads

- ▶ Set the pad type (SMD & Through-Hole most common)
- ▶ Define the pad size & optionally drill sizes



# Design the footprint

- ▶ Place the pads according to the footprint datasheet
- ▶ Use the 'Fab' layers to draw part outlines won't be included on the PCB
- ▶ Use the 'SilkS' layers to add markings for the PCB
  - ▶ Reference marker (included by default)
  - ▶ Orientation markers
  - ▶ Additional outline information





# Save & include new library

- ▶ Save the footprint into a new library
- ▶ Then use the library wizard to include it
- ▶ Optionally: Tweak parameters in the Library Manager

