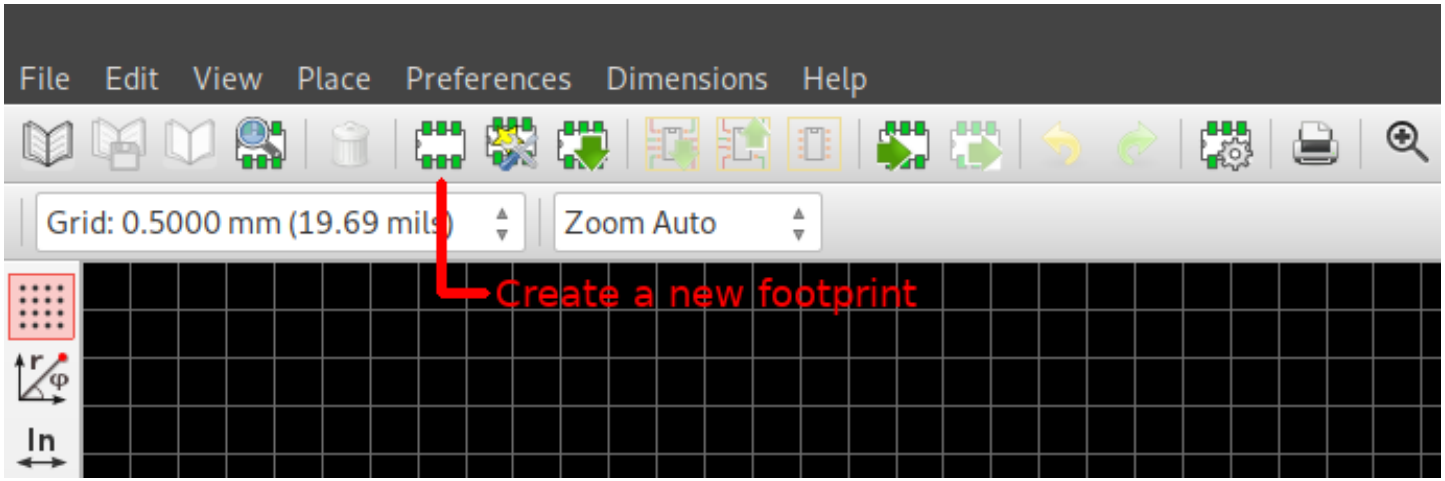
KiCAD
Creating a component footprint

Katharina Fey

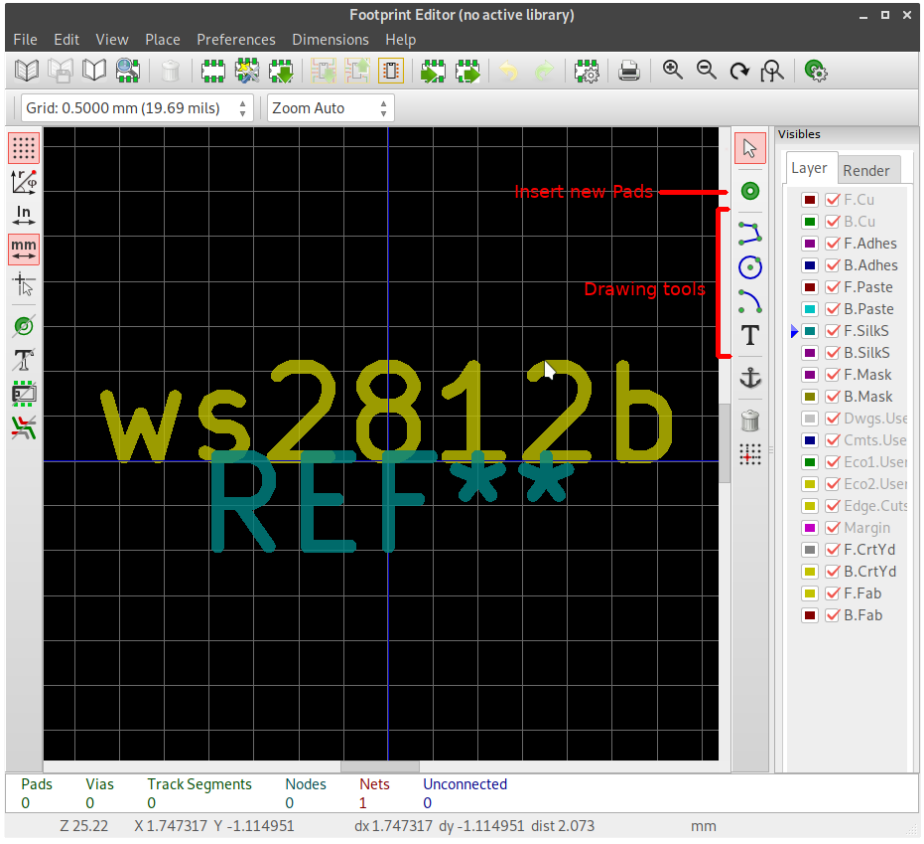
17. March 2018

Create new footprint

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

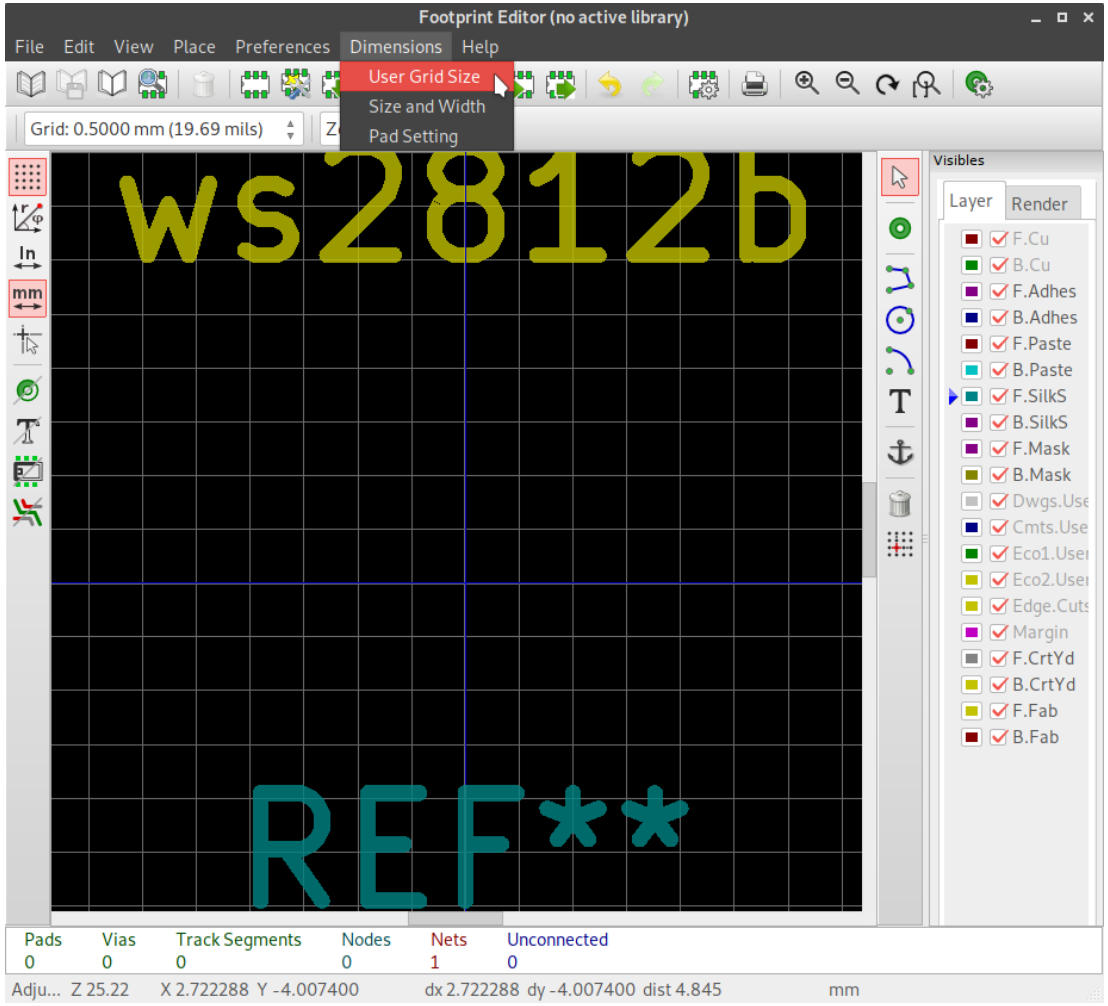


Create new footprint



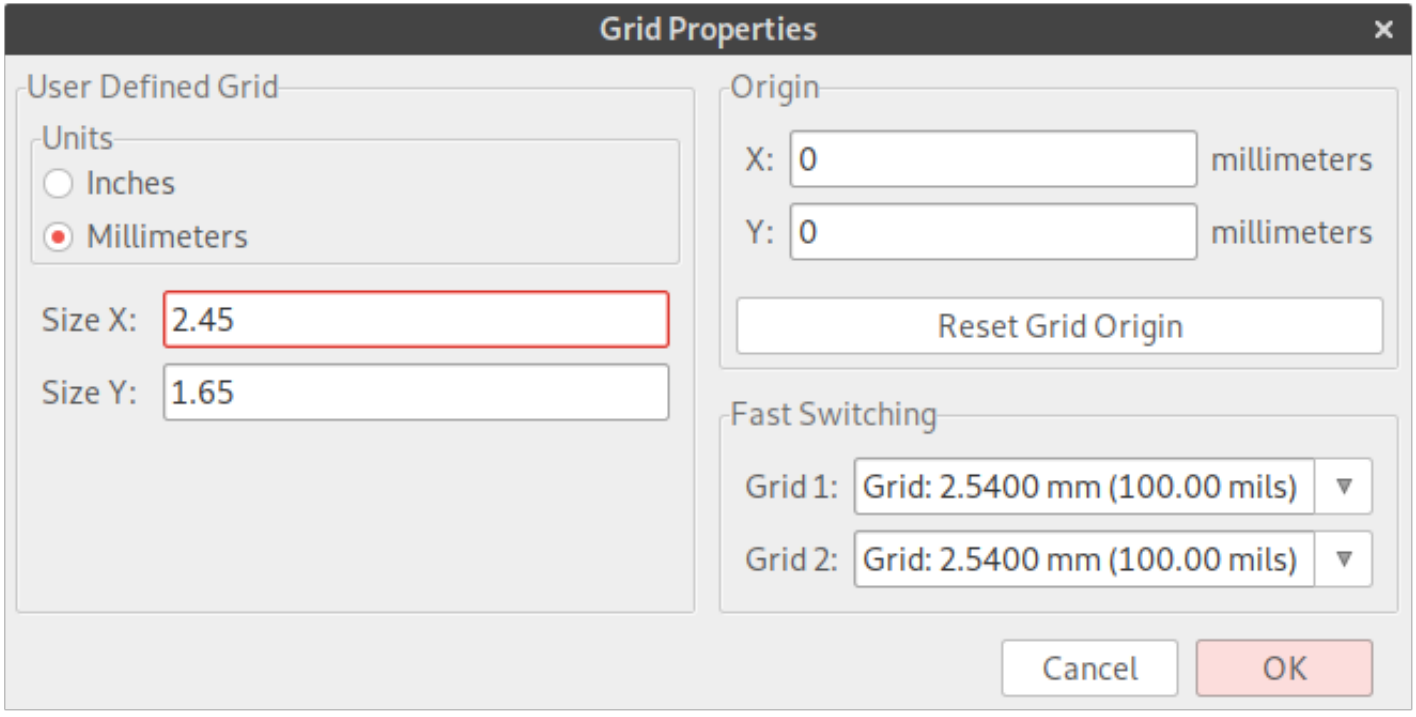
Set custom grid

- 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

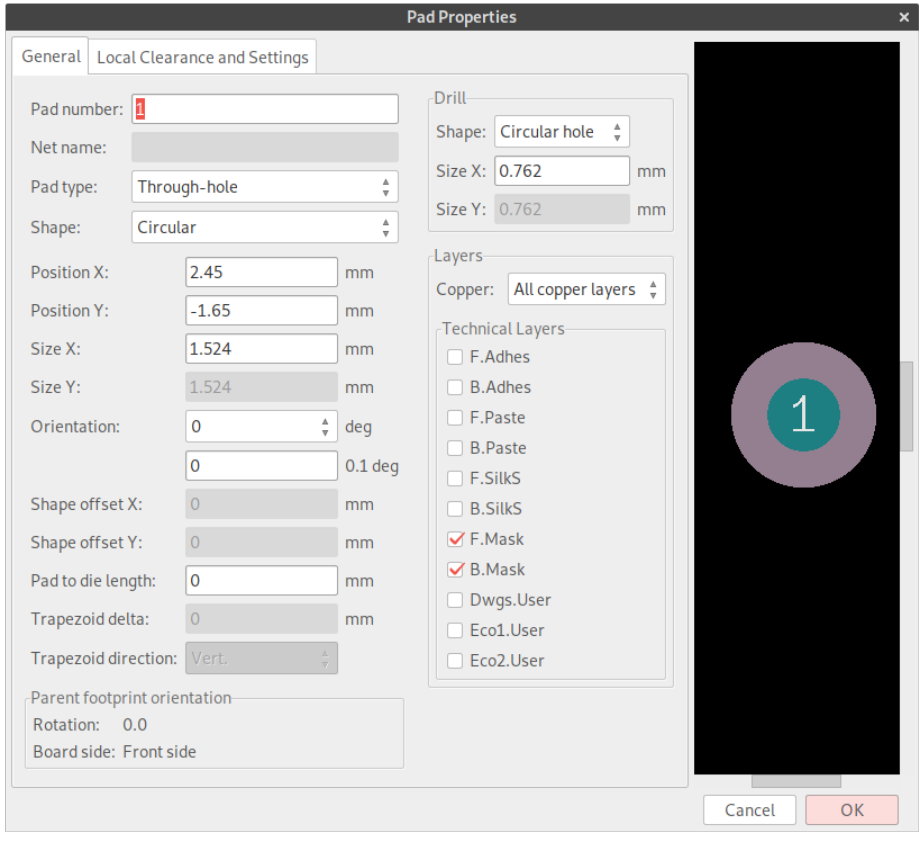


Create & customise Pads

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

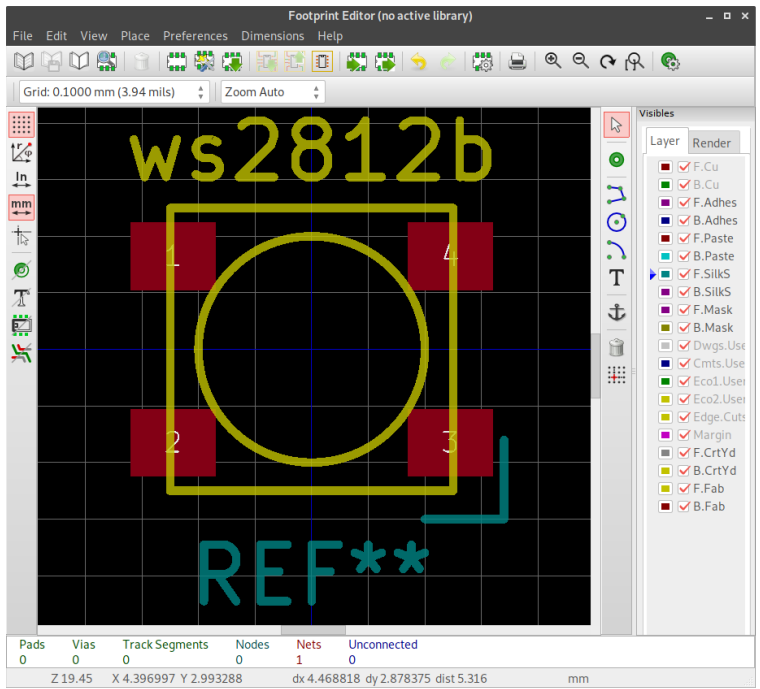
Create & customise Pads

- 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