How to create footprints using KiCAD.

It is important to know that components often come in standard packages, this means that one specific type of packaging might be the same for different components. For example, we are using the packaging 1206 for resistors, but the same packaging applies to 1206 capacitors or 1206 LED's, meaning that you can use the same footprint for either of those. With that in mind, the first thing to do is to determine the type of packaging (using the datasheet), and find out if that footprint already exists in KiCAD libraries (or maybe it is available on the internet).

In case the footprint doesn't exist, you can create it. For that, you will need the datasheet of the component (or use a caliper to measure all the dimensions of the component).

For example, if you have a component with the packaging SOT223 (might be a mosfet transistor, a voltage regulator, etc.), look for the appropriate datasheet and read the footprint information.
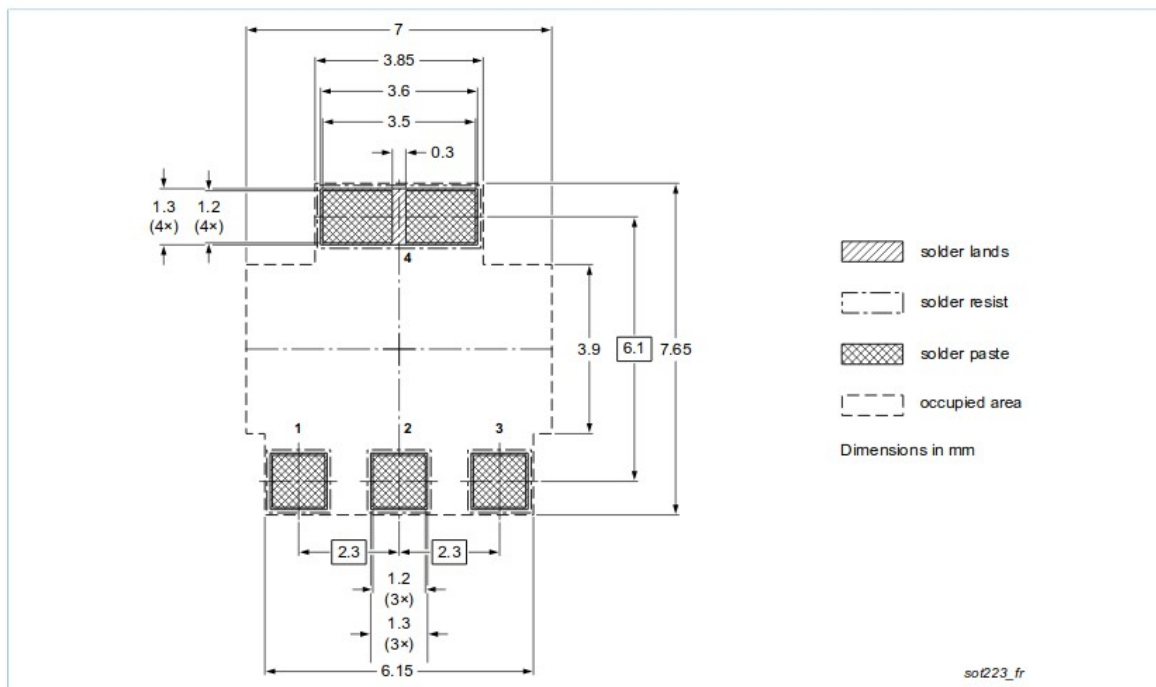
https://www.nxp.com/docs/en/package-information/SOT223.pdf

NXP Semiconductors **SOT223**

plastic surface-mounted package with increased heatsink; 4 leads
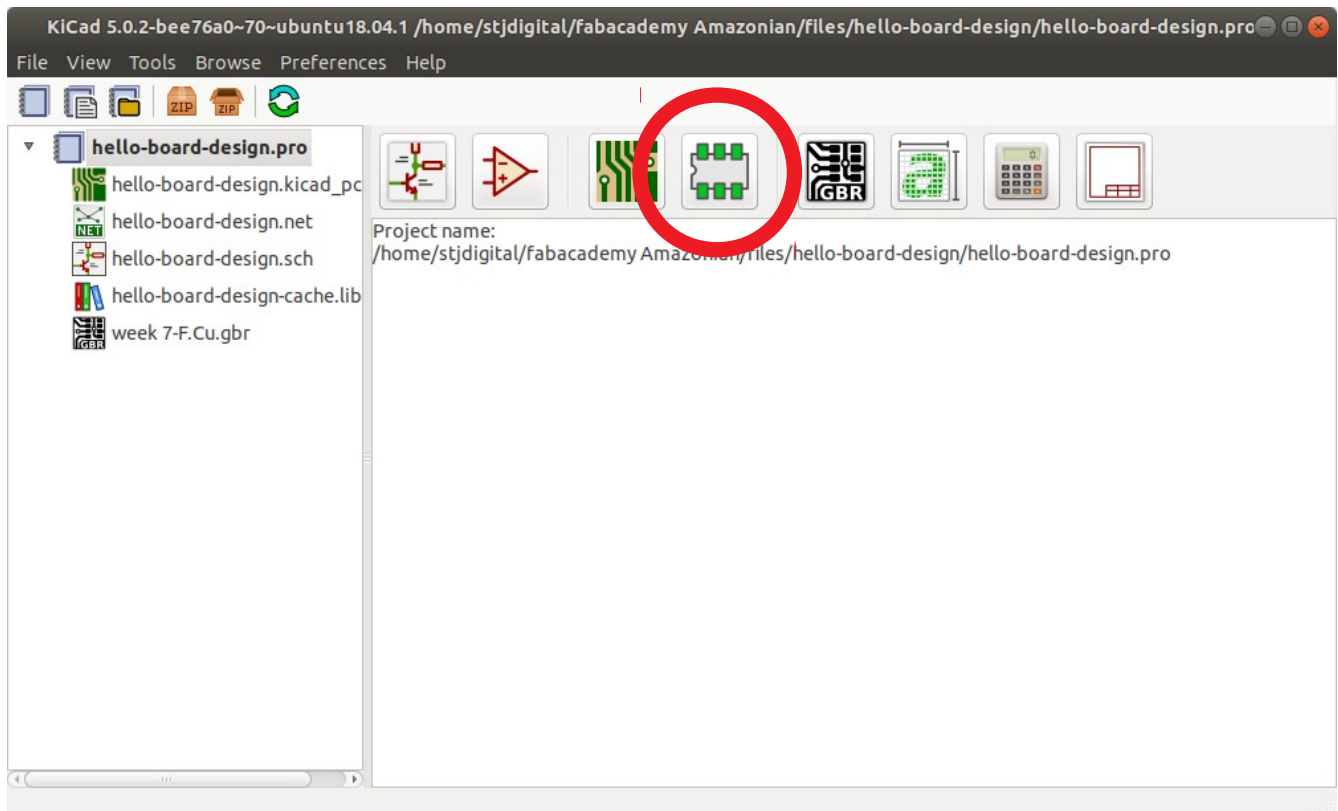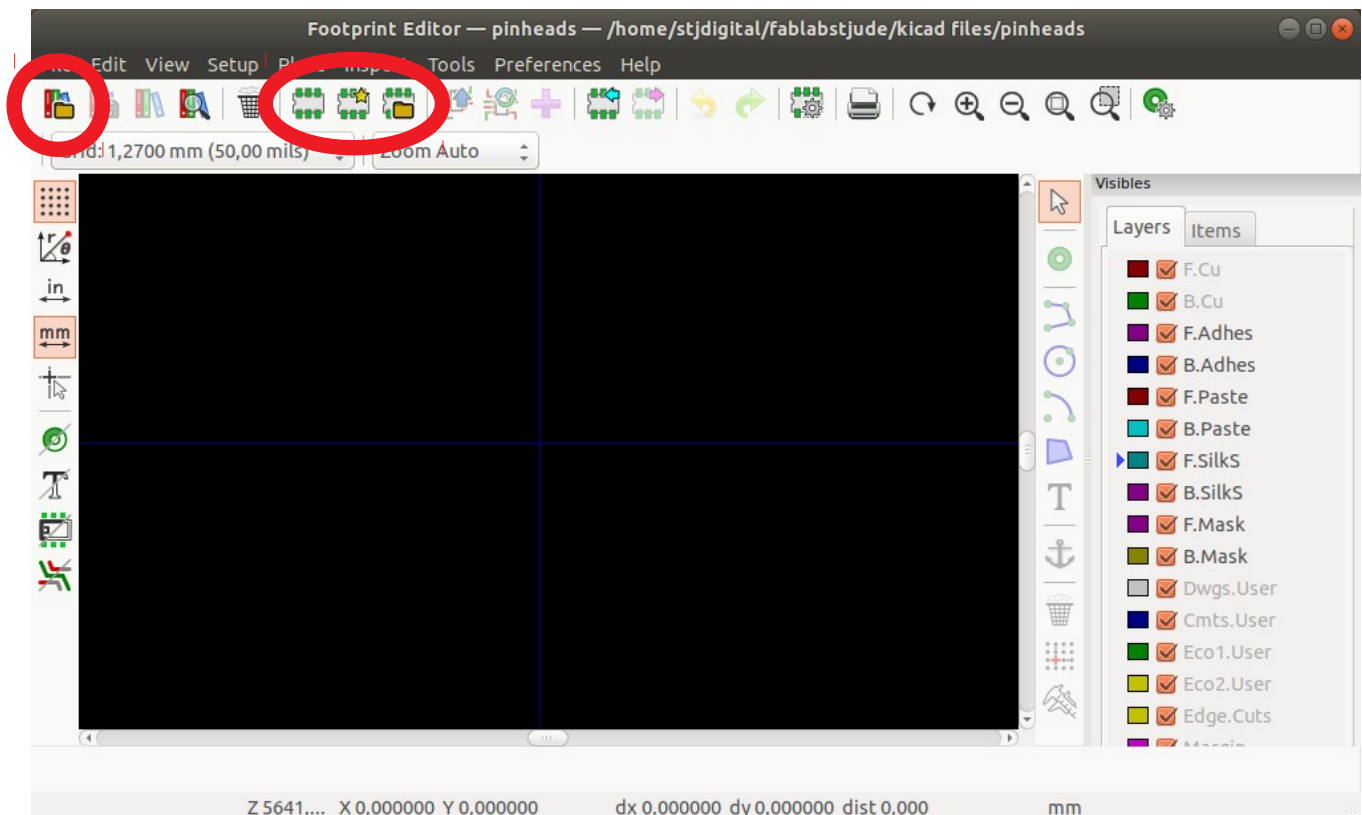
3. Soldering

You also need to know if the component is SMD (surface mount device) ot THT (through-hole technology).

Open KiCAD and go to Footprint Editor:

Now you have to define a library, to which your footprint will be added, then choose whether to create a
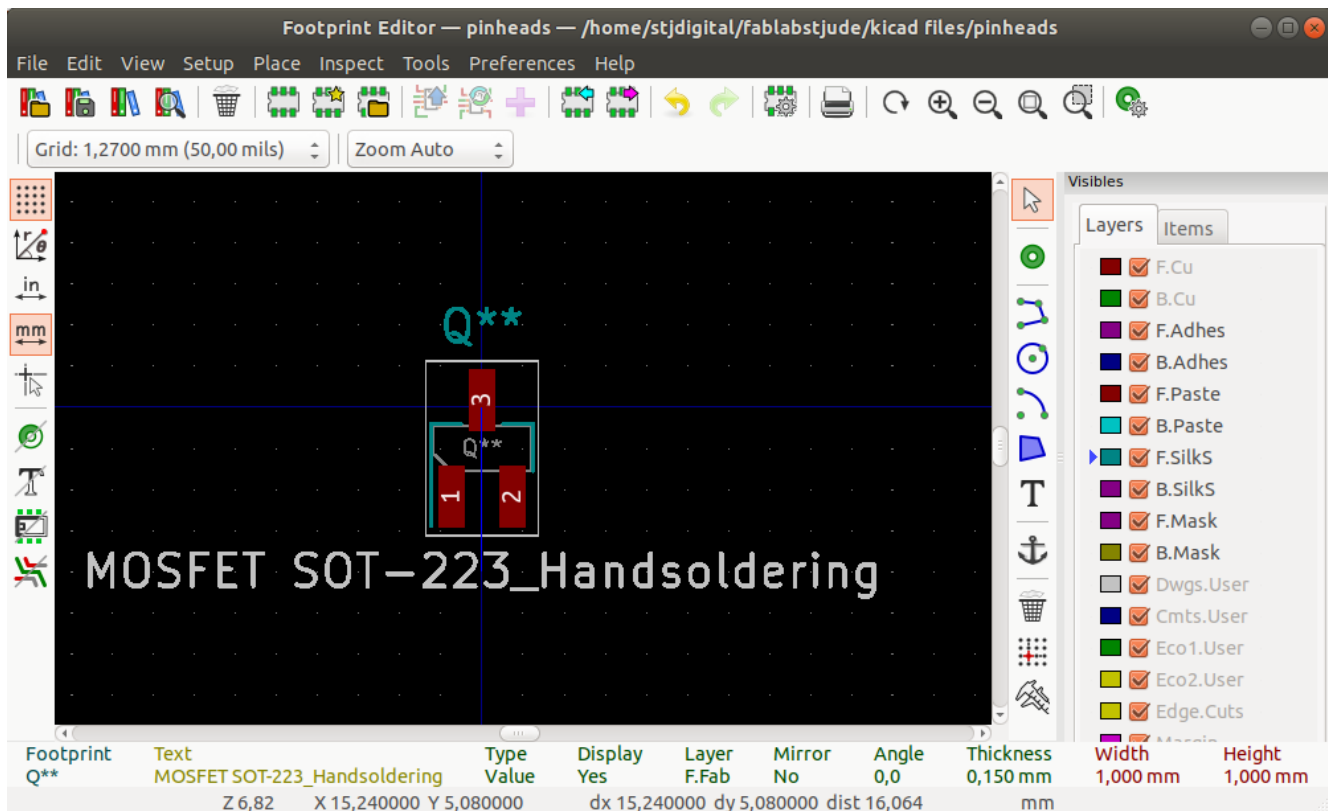

new footprint, use footprint wizard, or use another footprint as template.

The best option is to use another footprint as template, so you'll have an idea of which items must be created for the new footprint. I will use the SOT-23_Handsoldering as a base for my new footprint:
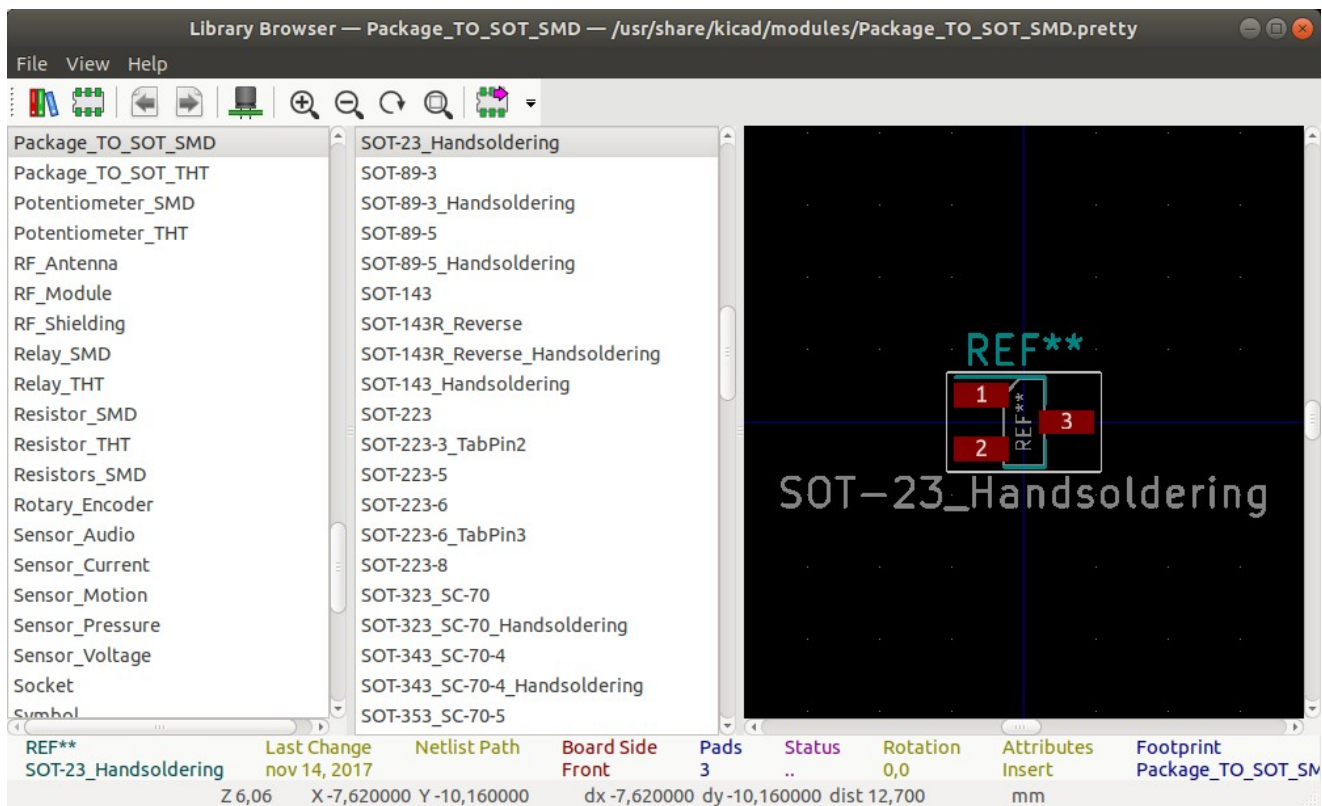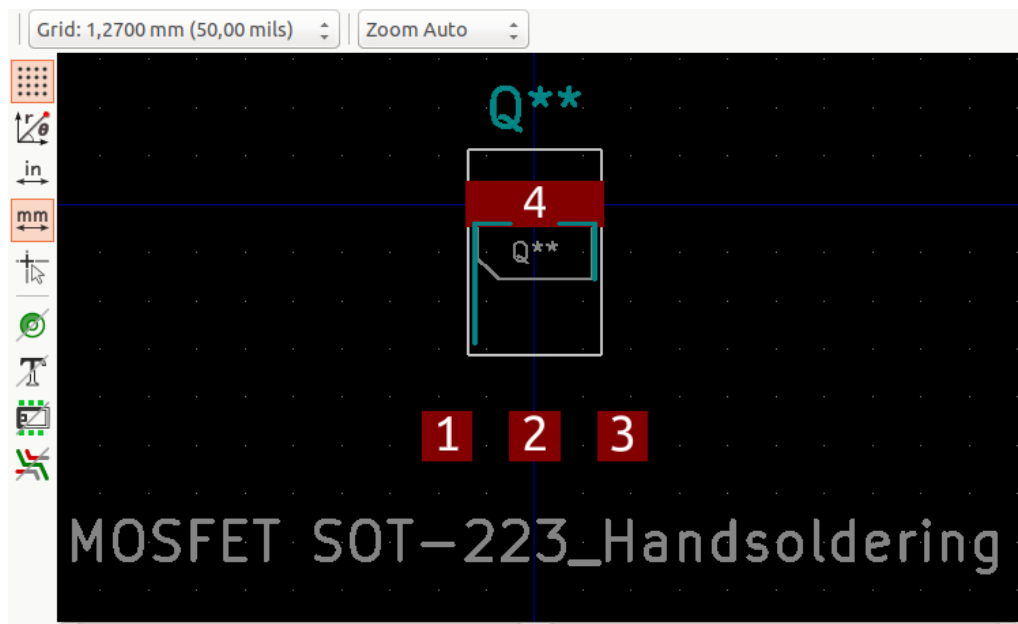

Now, you can edit each part of the footprint to meet your needs. Double-click any item to edit it.
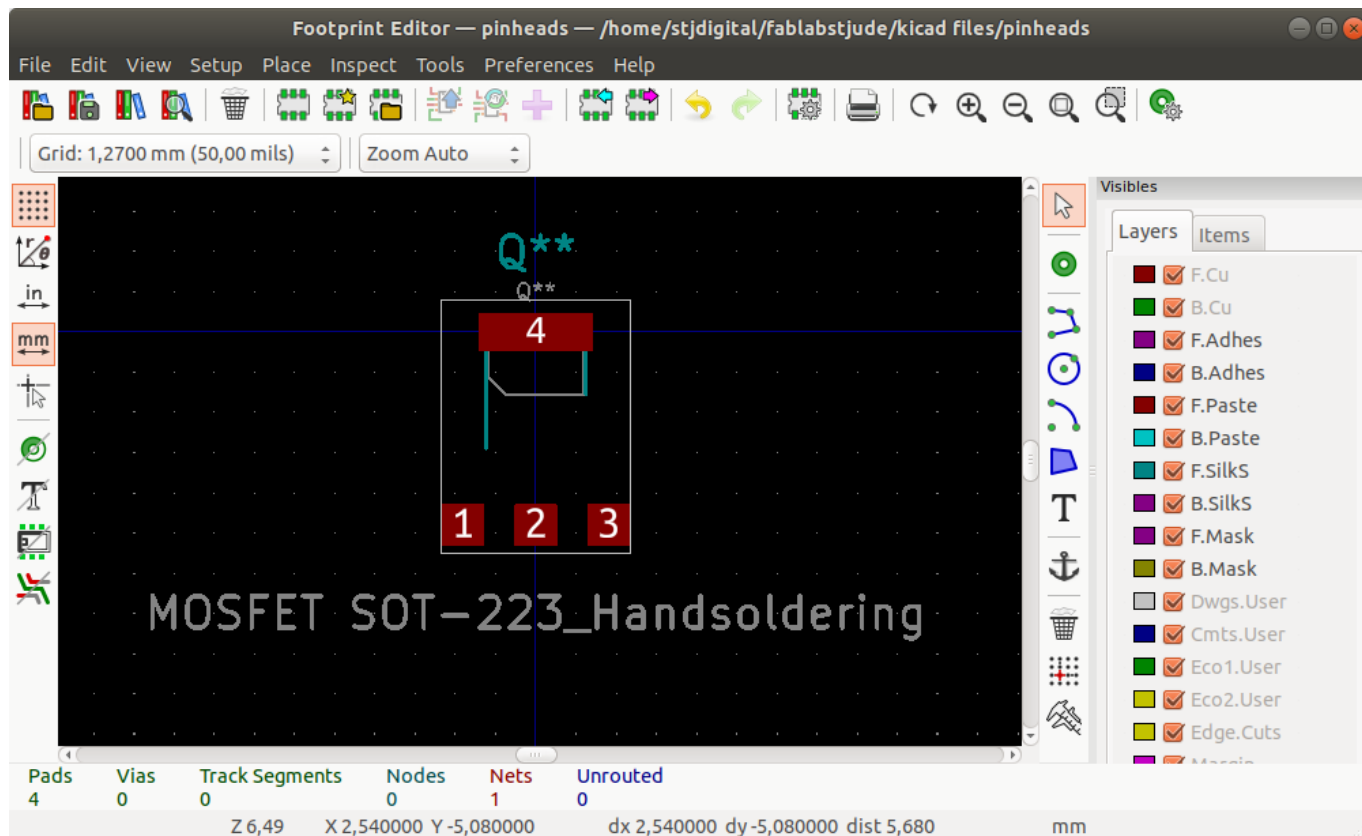

Notice that I changed the REF to Q (for a transistor), rotated the pads and changed the description text. Now, I will go to the datasheet to change the pads and the diemnsions. If you double-click the pad number 3, you enter the pad properties. I will change the size to 1,2 height and 3,6 length:
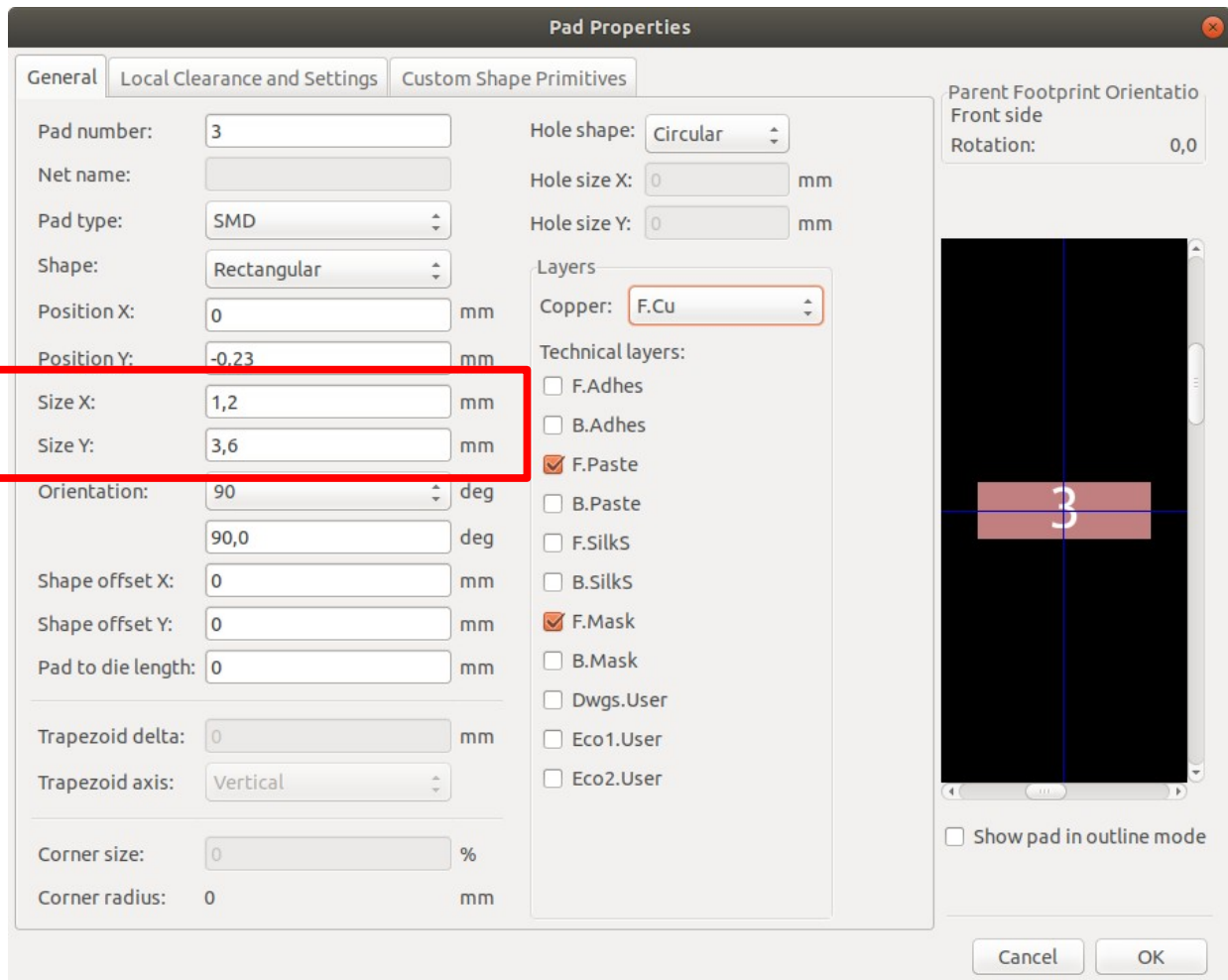
Also, according to the datasheet, there are three smaller pads on the lower side. I'll copy and paste one of the smaller pads, also, change their sizes, numbers and position:

I put the pad number 4 in the position x = 0 and y 0, so, the position for each other pad is:

      pad number 1: x = -2,3 and y = 6,1

      pad number 2: x = 0 and y = 6,1

      pad number 1: x = 2,3 and y = 6,1

Just for this guide's purposes, if I superpose the datasheet diagram over the footprint, they should match:

Finally, change the the enclosing square (not absolutely necessary).

Now you can save your work and use the new footprint.