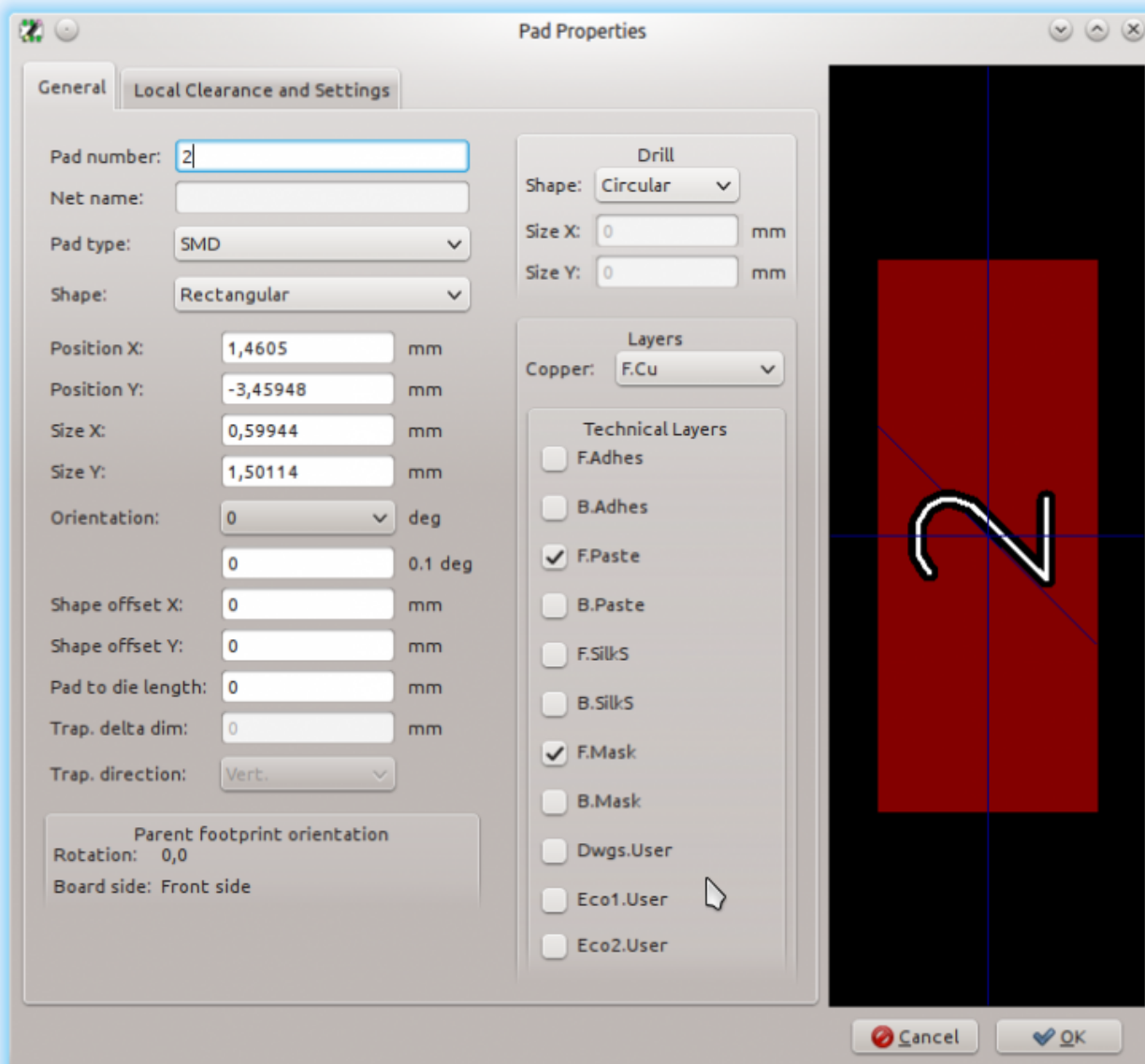


KiCad Blue Prints

Layer Settings in Footprint Editor

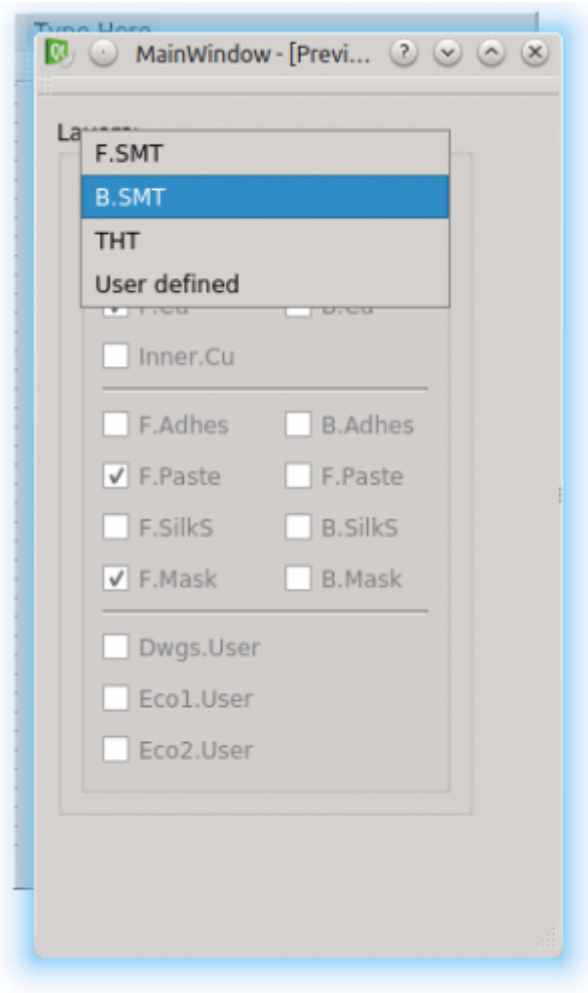
Current User Interface:

The Problem is here, that if you change the copper layer, the according technical layers do not change. This could lead for example to SMT pads on the bottom side, which are covered with solder mask. That should be avoided in it's root.

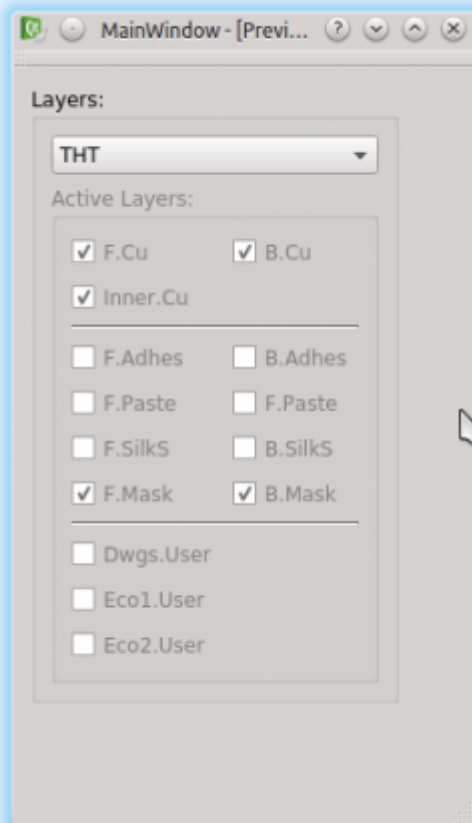
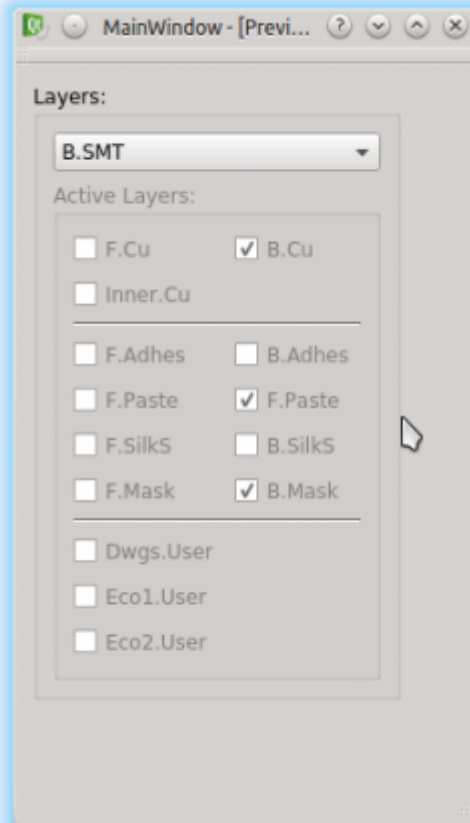
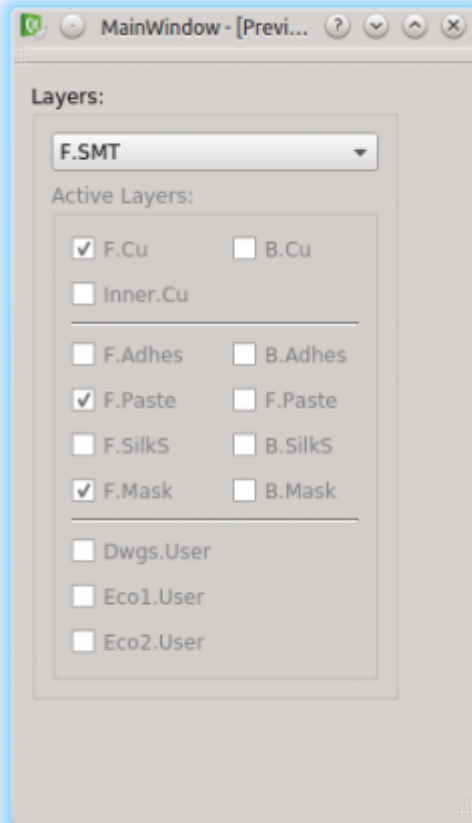


The options:

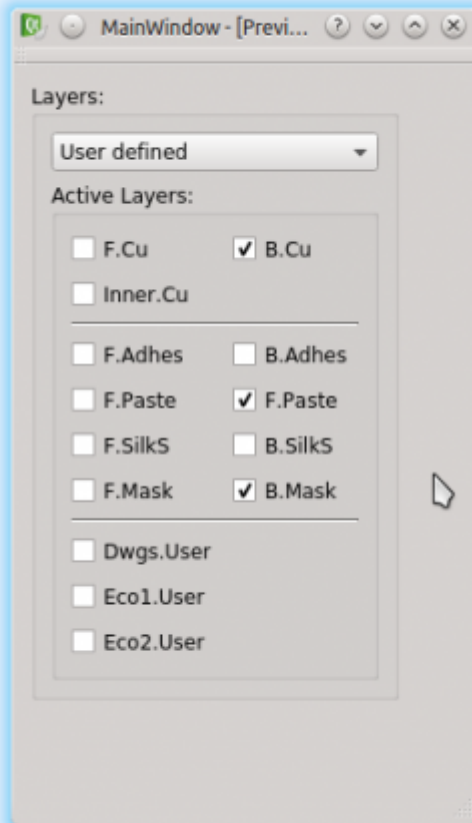
We have 4 options for predefined layer sets.



Fixed Layers for common templates:



And user defined layers:



From:

<http://www.zeilhofer.co.at/wiki/> - **Verschiedenste Artikel von Karl Zeilhofer**

Permanent link:

http://www.zeilhofer.co.at/wiki/doku.php?id=kicad_blueprints&rev=1425845530

Last update: **2015/03/08 21:12**

