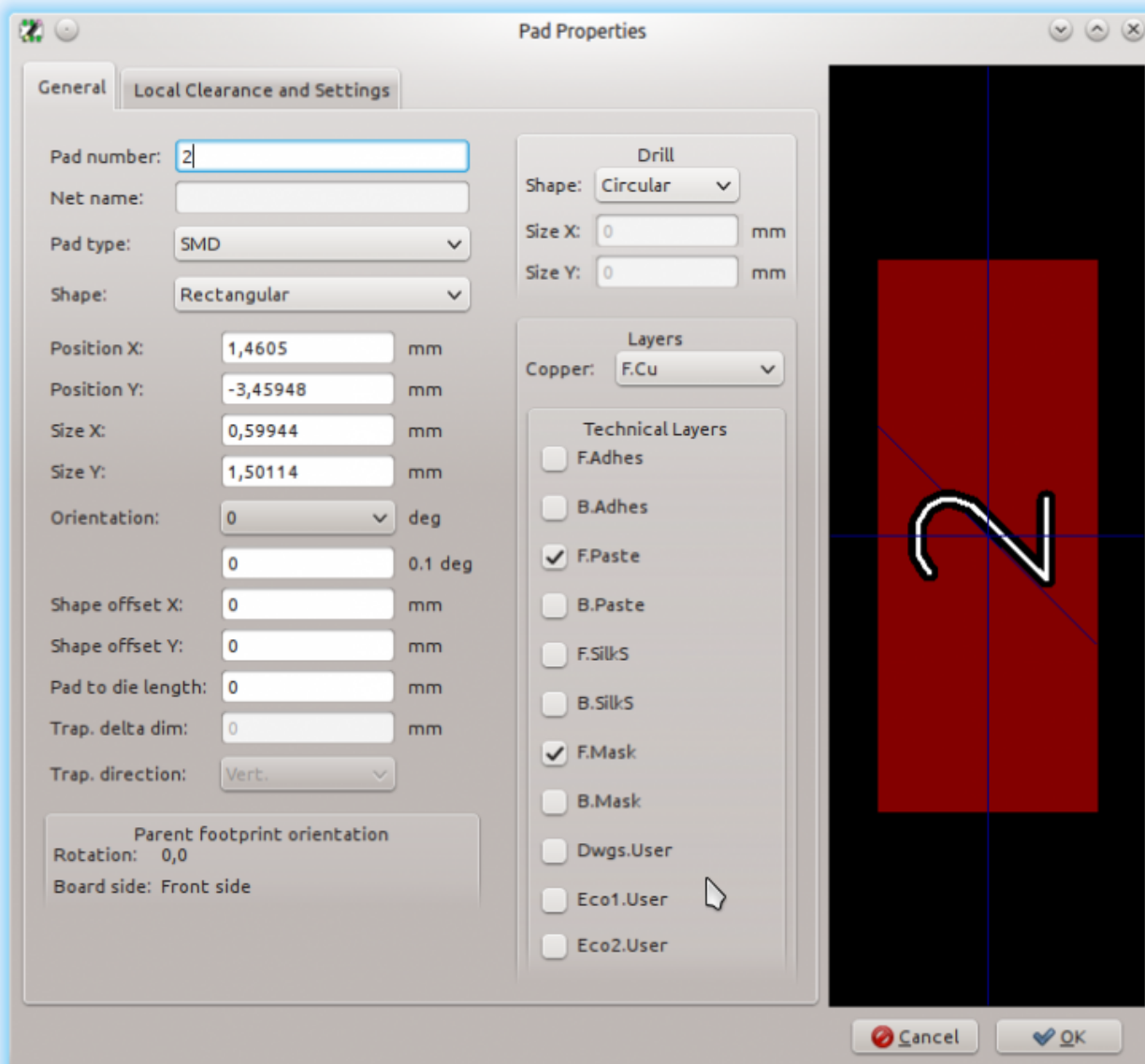


KiCad Blue Prints

Layer Settings in Footprint Editor

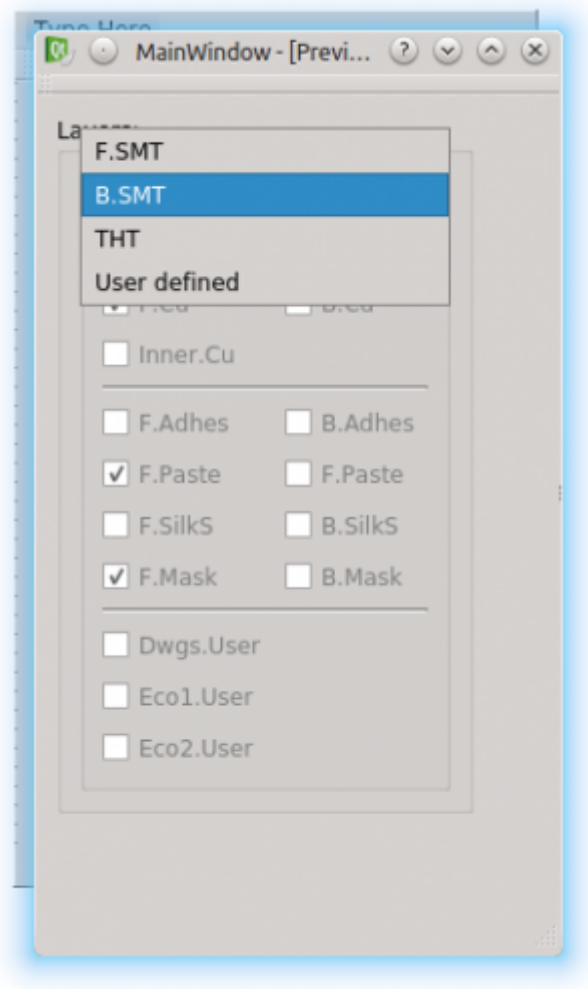
Current User Interface:

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.



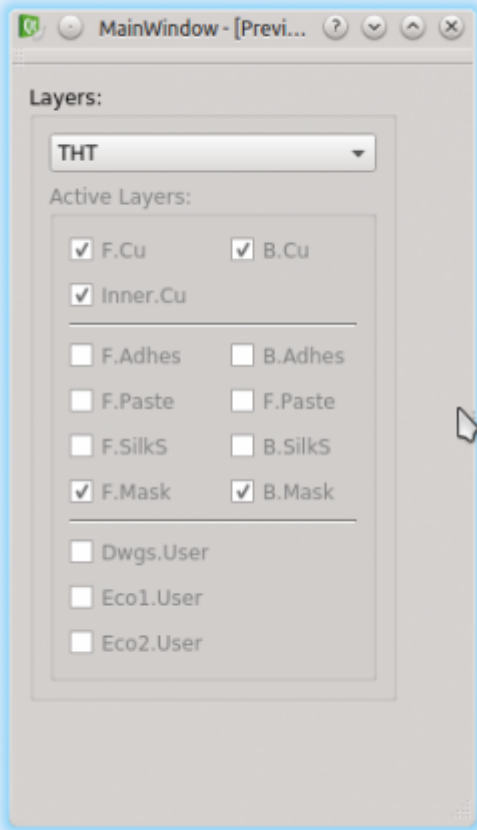
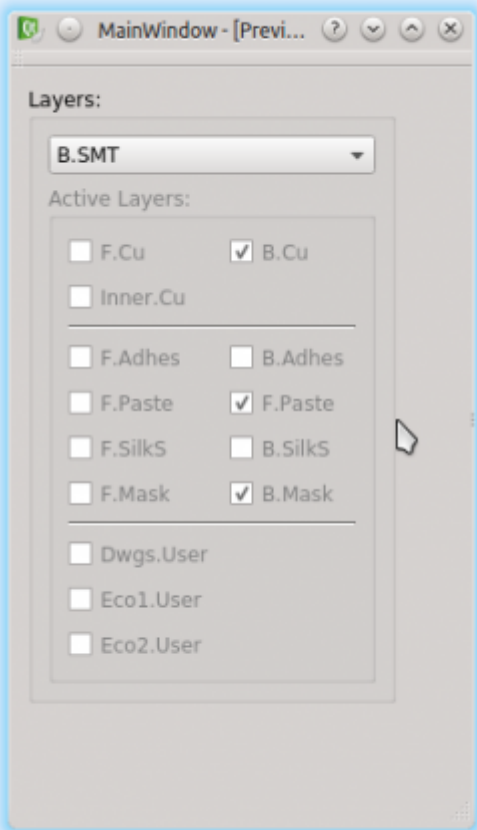
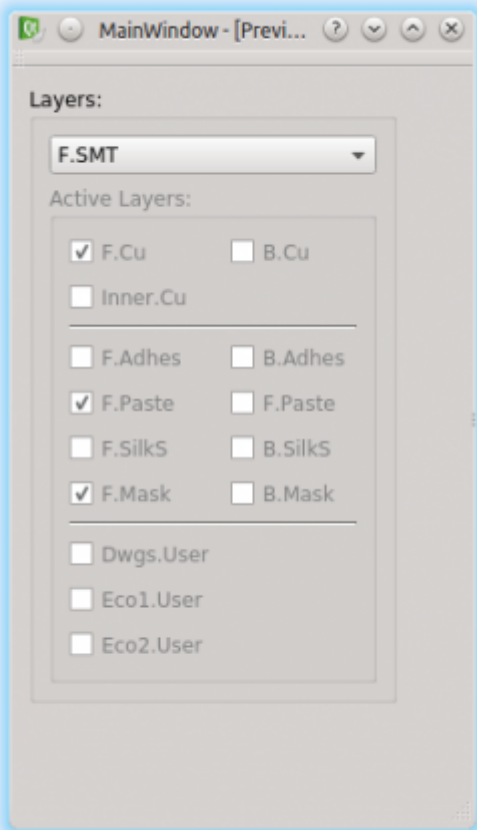
The options:

We have 4 options for predefined layer sets.



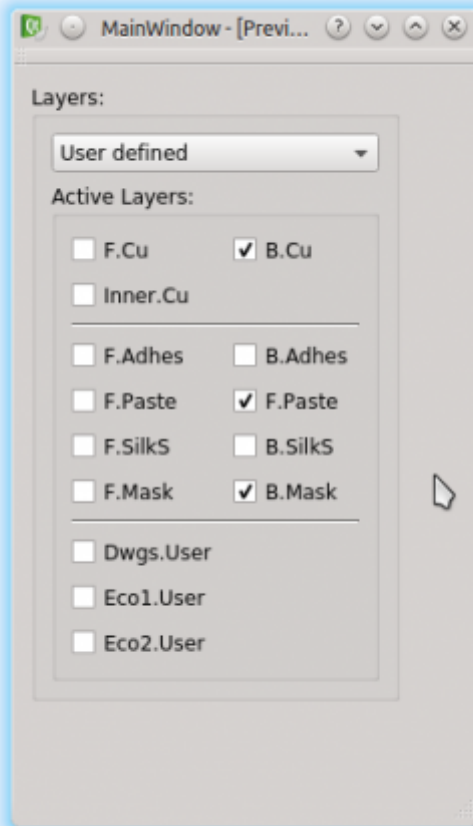
Fixed Layers for common templates:

In the first 3 sets, the layers can not be changed by the user, this is, that he get not confused by automatic cahanging checkboxes, which, her perhaps has changed manually before.



And user defined layers:

For flexibility we provide a user defined set of layers, where the checkboxes are enabled and editable



by the user:

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=1425845664

Last update: **2015/03/08 21:14**

