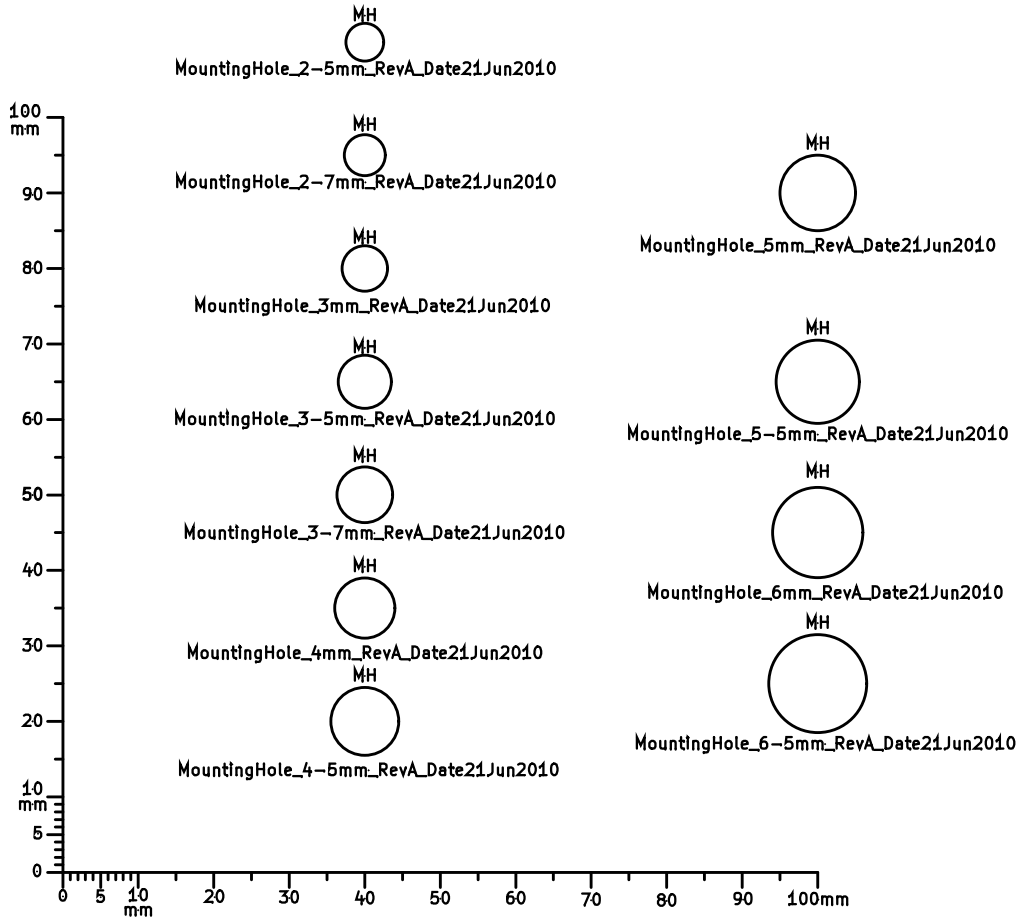
Normaly mounting holes in KiCAD are made by setting a pad, and setting the hole diameter and pad diameter to the same size. So there will be no pad. This is, because you often do not want pads at mounting holes.
But sometimes, you want a broader marking of the hole somewhere, but not in the gerber files. For this purpose, this mounting holes are created, with the pad not at the copper layer, but at the comment layer. So you do not have it at the gerber files (I suppose, you do not export your comment layer to gerber), but you can see and print it.
But be careful, because i violated some rules doing this. KiCAD normally does not accept any non copper layer as pad, and so, using this layers is blocked. I did this by editing the .mod-file with an editor, but if you now try to edit this holes, you cannot do this, because at every change KiCAD will recognises the non copper layer and blocks the editing of the pad. At moment, KiCAD (2011-04-29 BZR 2986)-stable works fine with this workaround, but this may not work at the future. You are recommended to use non-copper-plated holes, if you use holes without pads. Also, i may be more useful, to use milled holes for greater diameters.
devices

There exists also a symbol "Mounting hole". So you can put this holes on the schematic, and link it to a mounting hole footprint for PCBnew in CVpcb. This has the advantage, that you will allways keep the holes and not vorget them by re-reading the netlist at PCBnew.



1:1 Print! Check dimmension by controlling the gauge!

KiCAD Footprints of some mounting holes from the file MountingHole_RevA.mod

Author: Bernd Wiebus / Uedem / Germany / 04 July 2010

GNU-GPL NO WARRANTY!

File: MountingHole_RevA.brd		
Sheet: 1/1		
Title:		
Size: A4	Date: 4 jul 2011	Rev:
KiCad E.D.A.	pcbnew (2011-04-29 BZR 2986)-stable	Id: 1/1