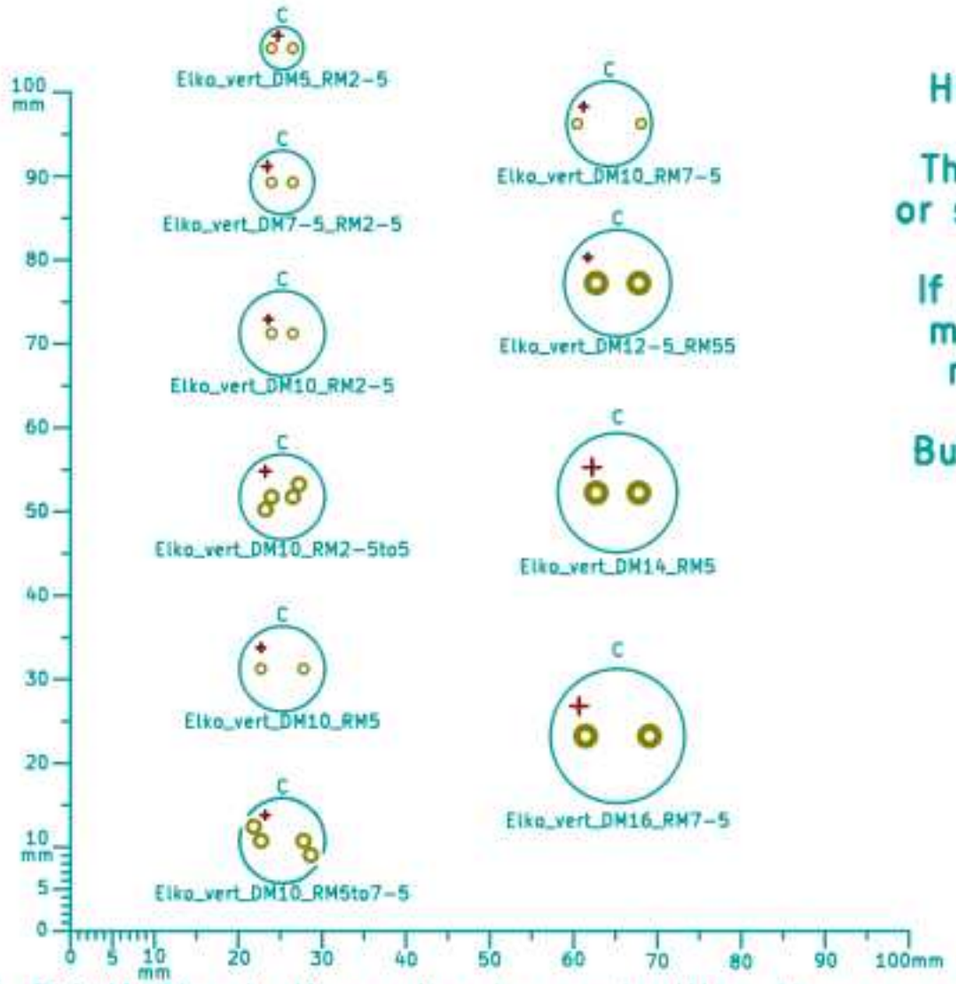
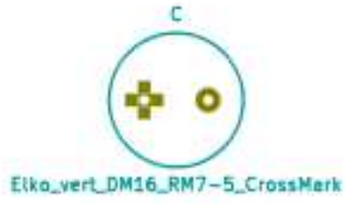


**Hint/Warning: The Plus symbol "+" is at the silk screen AND copper layer, too!
This may cause some problems with isolations or shortcuts, because a grafical line at a copper layer is NOT recognised by the DRC!
If you want to avoid this problems, choose the modules with the suffix "_CopperClear" in the name. There the "+" symbol is only at the silk screen layer.
But if you get boards without silk screen layer, you will have no orientation information!**



Experimental Footprint
with "cross-marked" +.

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

KiCAD Footprints of some vertical mounted through hole electrolytic capacitors from the file Elko_ThroughHole_RevB-3_30Dec2011.mod

Author: Oliver Kirsch + Bernd Wiebus / Uedem / Germany / 07 Feb 2011

GNU-GPL NO WARRANTY!
Licence: CC-Zero / Public Domain
NO RESTRICTION ! NO WARRANTY!
http://creativecommons.org/

For suggestions and improvements please contact me at bernd.wiebus@gmx.de

File: Elko_ThroughHole_RevB-3_30Dec2011.kicad_pcb
Sheet:
Title:
Size: A4 | Date: | Rev:
KiCad E.D.A. | Id: 1/1