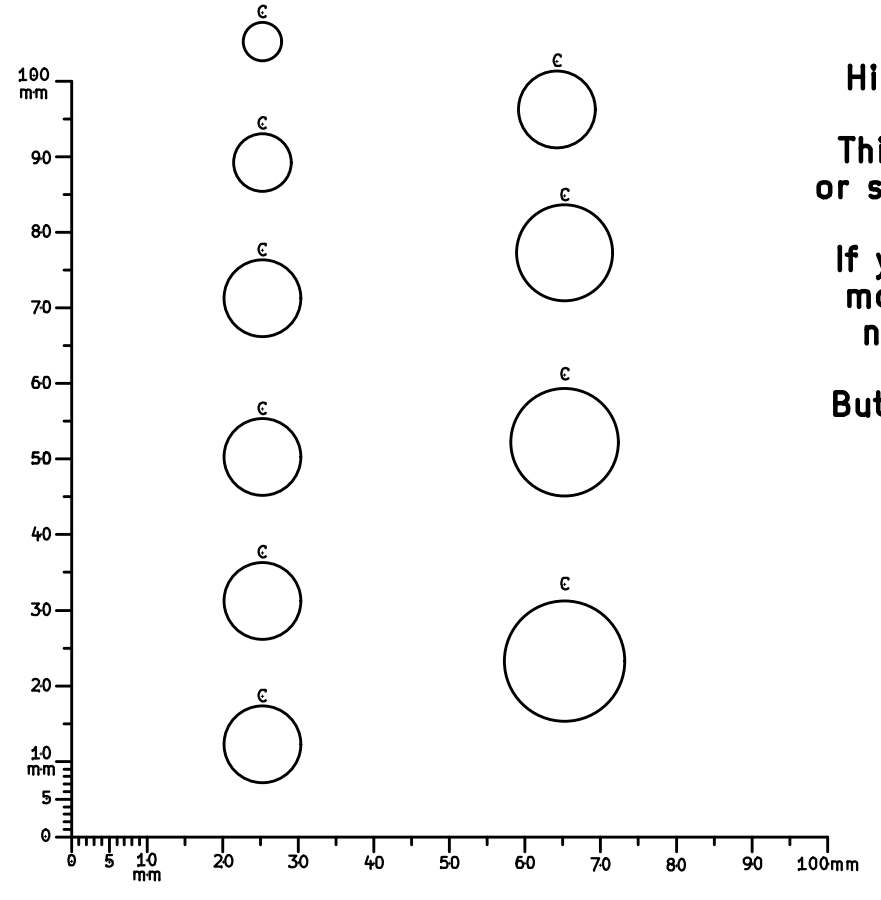

**Hint/Warning: The Plus symbol "+" is at the silk screen AND copper layer, too!
This may cause some problems with isolations or shortcuts, because a grafical line at a copper layer is NOT recognised by the DRC!
If you want to avoid this problems, choose the modules with the suffix "_CopperClear" in 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!**

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

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

Author: Bernd Wiebus / Uedem / Germany / 29 Dec 2011

GNU–GPL NO WARRANTY!