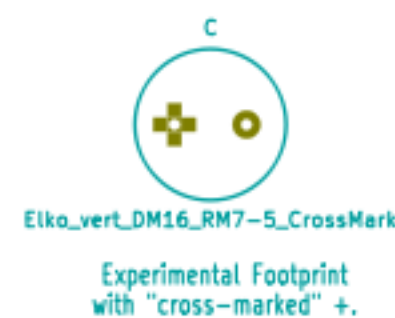


Hint/Warning: The Plus symbol "+" is at the silk screen AND copper layer, too!
 This may cause some problems with isolations or shortcuts, because a grafical line at a copper layer is NOT recognised by the DRC!
 If you want to avoid this problems, choose the modules with the suffix "_CopperClear" in 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 dimension by controlling the gauge!
 KiCAD Footprints of some vertical mounted through hole electrolytic capacitors from the file Elko_ThroughHole_RevB-3_30Dec2011.mod

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

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/>



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