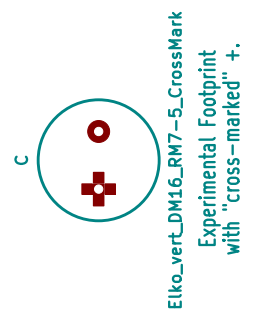


Hint/Warning: The Plus symbol "+" is at the silk screen AND copper layer, too!
 This may cause some problems with isolations or shortcuts, because a grafical line at a copper layer is NOT recognised by the DRC!
 If you want to avoid this problems, choose the modules with the suffix "_CopperClear" in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



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

KiCAD Footprints of some vertical mounted
 through hole electrolytic capacitors from the folder
 Elko_ThroughHole_RevB-3_30Dec2011.pretty
 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/>



Sheet:		Elko_ThroughHole_RevB-3_30Dec2011.kicad_pcb
File:		
Title:		
Size: A4	Date:	
KiCad E.D.A. kicad (2014-06-29 BZR 4957) - product		
Rev:		Id: 1/1