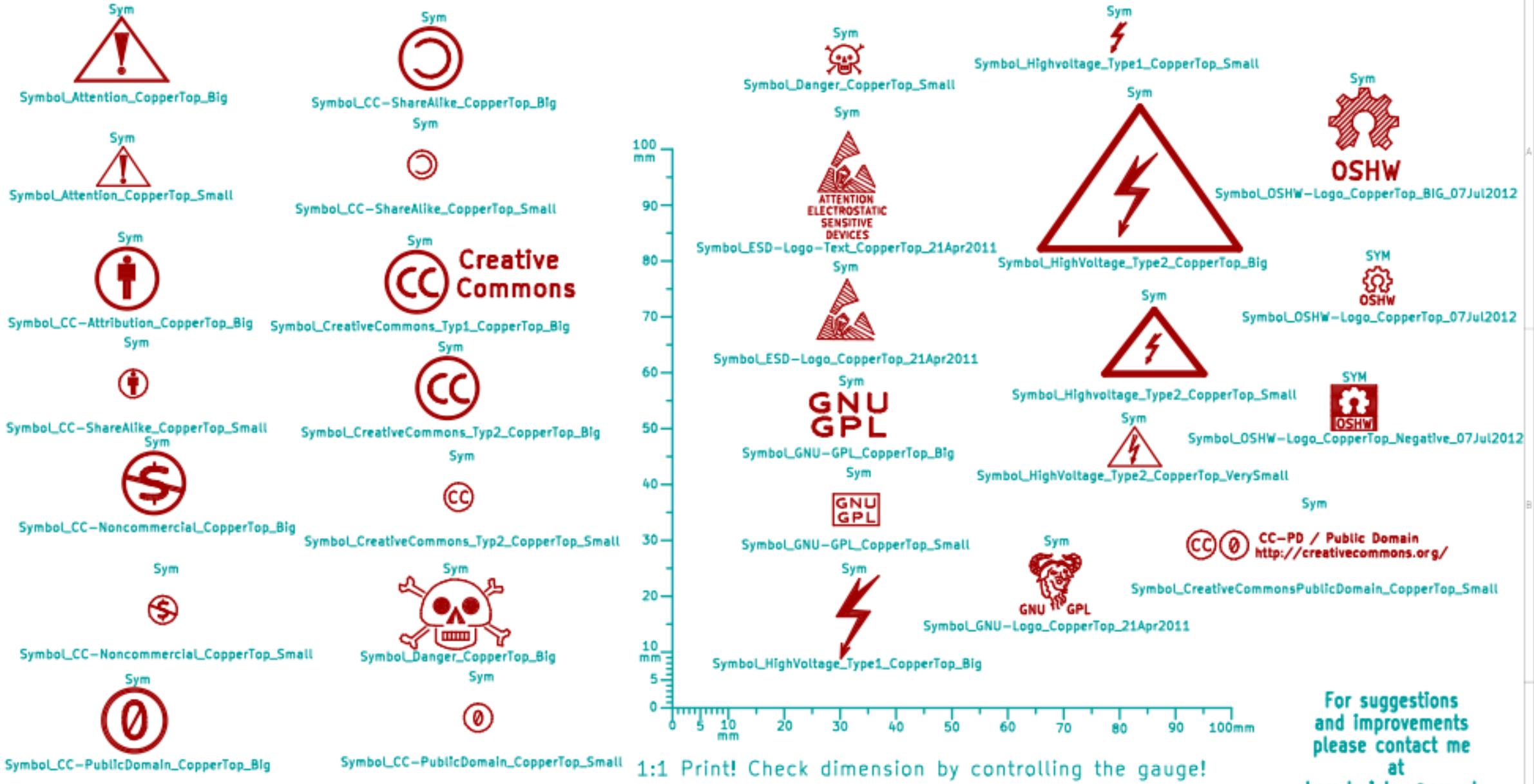

1:1 Print! Check dimension by controlling the gauge!

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

KiCAD Footprints of some Symbols from the file Footprint-Symbols_RevD_28Aug2012.mod

Author: Bernd Wiebus / Uedem / Germany / 28 Aug 2012

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

File: Footprint-Symbols_RevD_28Aug2012.kicad_pcb		
Sheet:		
Title:		
Size: A4	Date: 6 jun 2013	Rev:
KiCad E.D.A.		Id: 1/1