



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

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

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

Author: Bernd Wiebus / Uedem / Germany / 28 Aug 2012
GNU-GPL NO WARRANTY!

File: Footprint-Symbols_RevD_28Aug2012.brd	
Sheet: 1/1	
Title:	
Size: A4	Date: 28 aug 2012
KiCad E.D.A. pcbnew (2012-01-19 BZR 3256) -stable	
Rev:	Id: 1/1