



Note: Resistor network footprint datasheed says 1206, but dimensions are equal to 0603 in KICAD

Magjack ze stejne strany jako soucastky TME.CZ

Sheet: /	
File: stm32f4_ethernet.sch	
<b>Title:</b>	
Size: A4	Date:
KiCad E.D.A. kicad (2016-08-20 BZR 7083)-product	Rev:
	Id: 1/1