3 This document is the exclusive property of Byte Lab Grupa d.o.o. Company and it should not be distributed or PCB requirements List of Gerber files reproduced into any other format without prior permission from Byte Lab Grupa d.o.o. Parameter File Description Gener at ed extension FR-4 (Tg 140) Material Number of layers GTP Top Solder Paste X 2. Board thickness 1.6 (±0.1) mm GTO Top Overlay (Silk) Black 3. GTS Top Solder Mask X Solder mask HAL (with lead) 4. X GTL Top Layer HAL (lead-free) 5. X Surface finishing Mid Layer 1 **▼** ENIG 6. G2 Mid Layer 2 other: 7. Bottom Layer 8. X Silkscreen top White Bottom Solder Mask 9. X White GBO Bottom Overlay (Silk) Silkscreen bottom VUSB 0.15 mm 10. Bottom Solder Paste X Min. track/spacing NOTECARRIER-F VBAT SDA Min. hole size 0.3 mm 11. GM10 Bottom PSM VMAIN SÇL 103x50.8 (±0.2) mm 12. X GM15 Board Layer Board dimension N_V3EO **6_D5** ■ No 13. GM8 Milling \Box Panelization VSOLAR **Б_D6** ■ 14. П V-Cut RoHS compatibility F_D9 ■ Round holes GND F⊵D10 Number of Gerber Files 15 15. TXT X N_ENo F2_D11 **■** 16. TXT Slot holes F⊵D12 N_NRST 17. Round holes F_D13 N_BOOT 18. DRR NCDrill file report N_SCL F_B0 **■** 19. Gerbers Specification X N_SDA F_TX В 20. X GM16 N_ATTN F_RX Sheet Top AUX_EN AUX_RX F_MOSI AUX_TX AUX1 Additional requests AUX2 Via in pad **AUX3**° AUX4 Half holes/castellated edges N_RX Plated edge N_TX O \Box Impedance control GND NENRST Blind vias Other: Layer stack up details Copper Dielectric Dielectric Dielectric С Layer name thickness height Top Layer (GTL) 0.035 mm FR-4 O.1 mm Prepreg 0.0175 mm Mid Layer 1 (G1) 1.265 mm FR-4 Core Mid Layer 2 (G2) 0.0175 mm O.1 mm FR-4 Prepreg Bottom Layer (GBL) SIGNATURE TITLE: M. Hamin AUTHOR: Gerber specification T. Zvonc ACCEPTOR: D PART NO.: 2200-814 Byte Lab Grupa d.o.o. Médarska 69/1 DESIGN DOCUMENT NAME: FORMAT: 10000 Zagreb 100275 Notecarier F Rev. 11 Croatia

Α

В

С