Project: PI-0017

Name: RPi-GS-CAM-SER Rev. 1.0

Date: 2023-05-29

Company: PCB Arts GmbH

Phone: +49 151 58154880

Gerberformat

-------------------

Format: RS274X

NC Drill Format

-------------------

Format: Execellon

Unit: metric

Leading: 4

Trailing: 3

Zero suppress: Suppress Leading Zeroes

Name File Name Plot Type Layer Stack

---- --------- --------- -----------

X Gerber Layer RPi-GS-CAM-SER\_V1-0.GTO Top Overlay 1

X Gerber Layer RPi-GS-CAM-SER\_V1-0.GBO Bottom Overlay 4

X Gerber Layer RPi-GS-CAM-SER\_V1-0.GM29 Top Assembly 1

X Gerber Layer RPi-GS-CAM-SER\_V1-0.GM30 Bottom Assembly 4

X Gerber Layer RPi-GS-CAM-SER\_V1-0.GTL Routing 1

X Gerber Layer RPi-GS-CAM-SER\_V1-0.G1 Routing 2

X Gerber Layer RPi-GS-CAM-SER\_V1-0.G2 Routing 3

X Gerber Layer RPi-GS-CAM-SER\_V1-0.GBL Routing 4

X Gerber LSM RPi-GS-CAM-SER\_V1-0.GTS Solder Mask 1

X Gerber LSM RPi-GS-CAM-SER\_V1-0.GBS Solder Mask 4

X Gerber PSM RPi-GS-CAM-SER\_V1-0.GTP Paste Mask 1

X Gerber PSM RPi-GS-CAM-SER\_V1-0.GTP Paste Mask 4

X Gerber BOL RPi-GS-CAM-SER\_V1-0.GM1 Board Outline

X NC Drill DK RPi-GS-CAM-SER\_V1-0-

Plated.TXT Plated Drill

X NC Drill NDK RPi-GS-CAM-SER\_V1-0-

NonPlated.TXT NonPlated Drill

Special Notes:

Keine

END