

Alex-XNR : <https://github.com/Alex-XNR>  
martinroger : <https://github.com/martinroger>

<https://github.com/martinroger/VXDash>  
<https://cadlab.io/projects/vxdash>

Sheet:  
File: VXDash.kicad\_pcb

**Title: Main PCB**

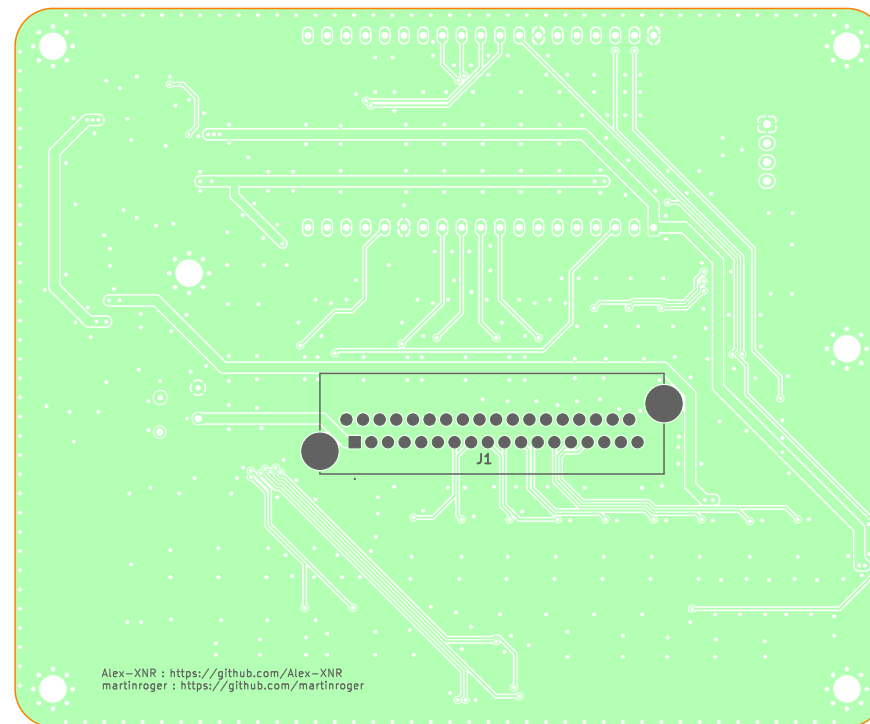
Size: A4      Date: 2023-05-27

KiCad E.D.A. kicad (7.0.0)

**Rev: 1.2.2**

Id: 1/1





Alex-XNR : <https://github.com/Alex-XNR>  
martinroger : <https://github.com/martinroger>

<https://github.com/martinroger/VXDash>  
<https://cadlab.io/projects/vxdash>

Sheet:  
File: VXDash.kicad\_pcb

**Title: Main PCB**

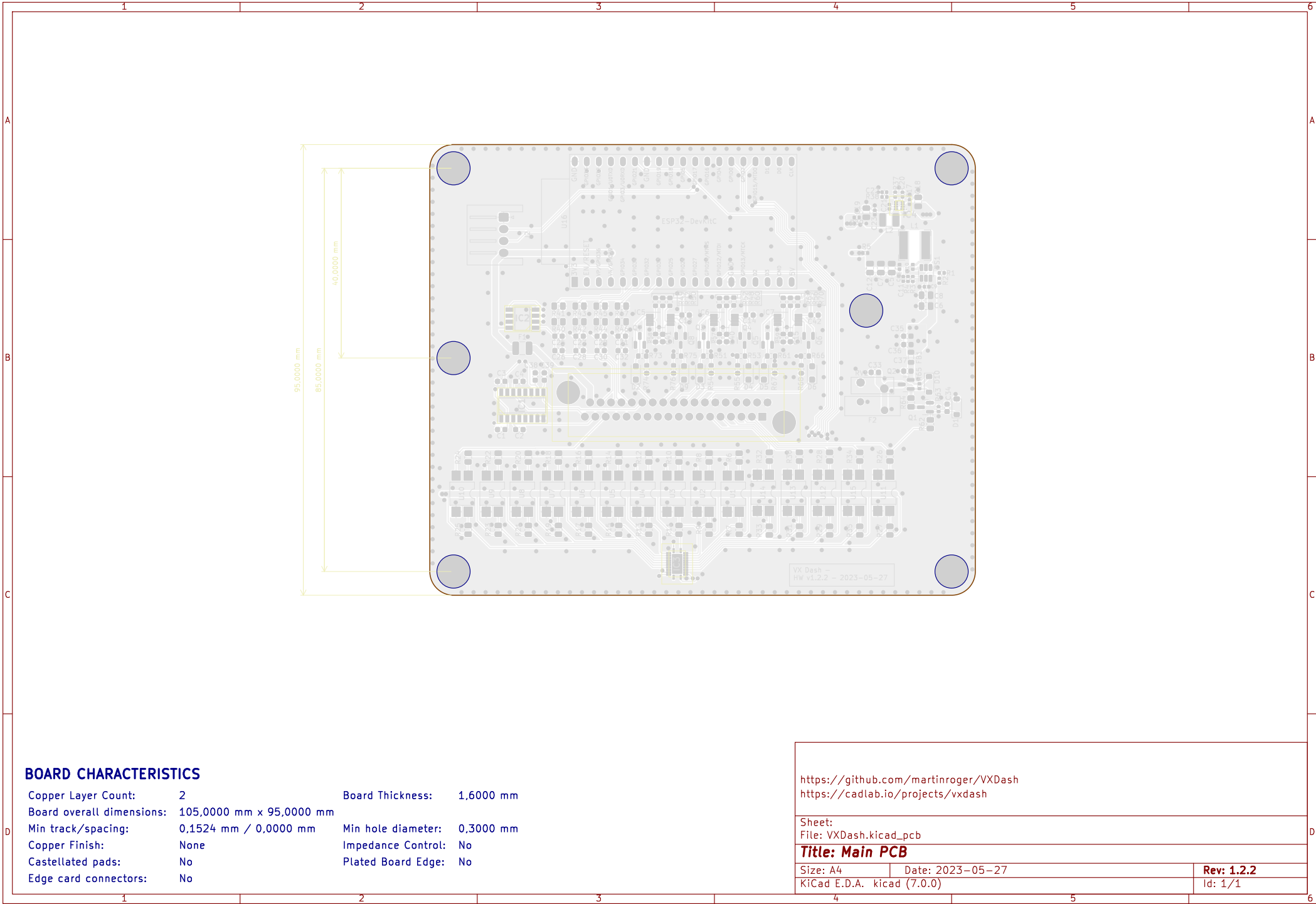
Size: A4 Date: 2023-05-27

KiCad E.D.A. kicad (7.0.0)

**Rev: 1.2.2**

Id: 1/1





BOARD CHARACTERISTICS

Copper Layer Count:	2	Board Thickness:	1,6000 mm
Board overall dimensions:	105,0000 mm x 95,0000 mm		
Min track/spacing:	0,1524 mm / 0,0000 mm	Min hole diameter:	0,3000 mm
Copper Finish:	None	Impedance Control:	No
Castellated pads:	No	Plated Board Edge:	No
Edge card connectors:	No		

<https://github.com/martinroger/VXDash>  
<https://cadlab.io/projects/vxdash>

Sheet:  
File: VXDash.kicad\_pcb

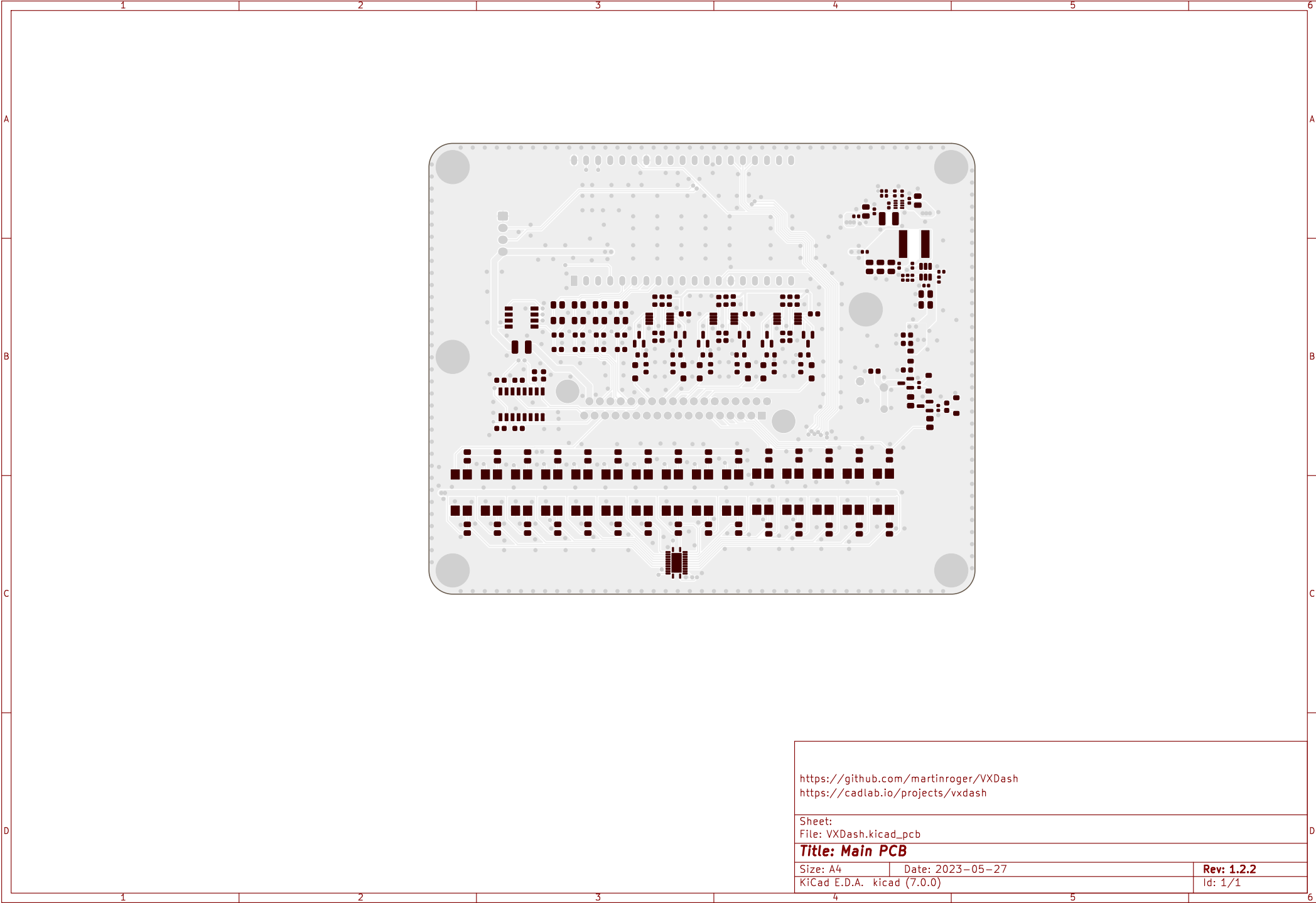
Title: Main PCB

Size: A4 Date: 2023-05-27

KiCad E.D.A. kicad (7.0.0)

Rev: 1.2.2

Id: 1/1



<https://github.com/martinroger/VxDash>  
<https://cadlab.io/projects/vxdash>

Sheet:  
File: VxDash.kicad\_pcb

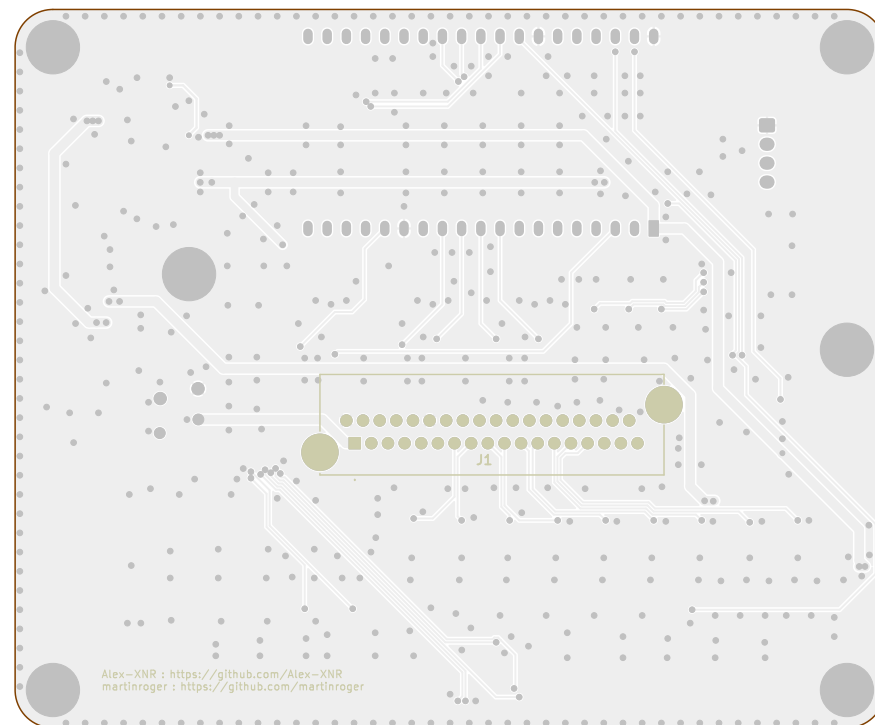
**Title: Main PCB**

Size: A4 Date: 2023-05-27

KiCad E.D.A. kicad (7.0.0)

**Rev: 1.2.2**

Id: 1/1



Alex-XNR : <https://github.com/Alex-XNR>  
martinroger : <https://github.com/martinroger>

<https://github.com/martinroger/VXDash>  
<https://cadlab.io/projects/vxdash>

Sheet:  
File: VXDash.kicad\_pcb

**Title: Main PCB**

Size: A4 Date: 2023-05-27

KiCad E.D.A. kicad (7.0.0)

**Rev: 1.2.2**

Id: 1/1

