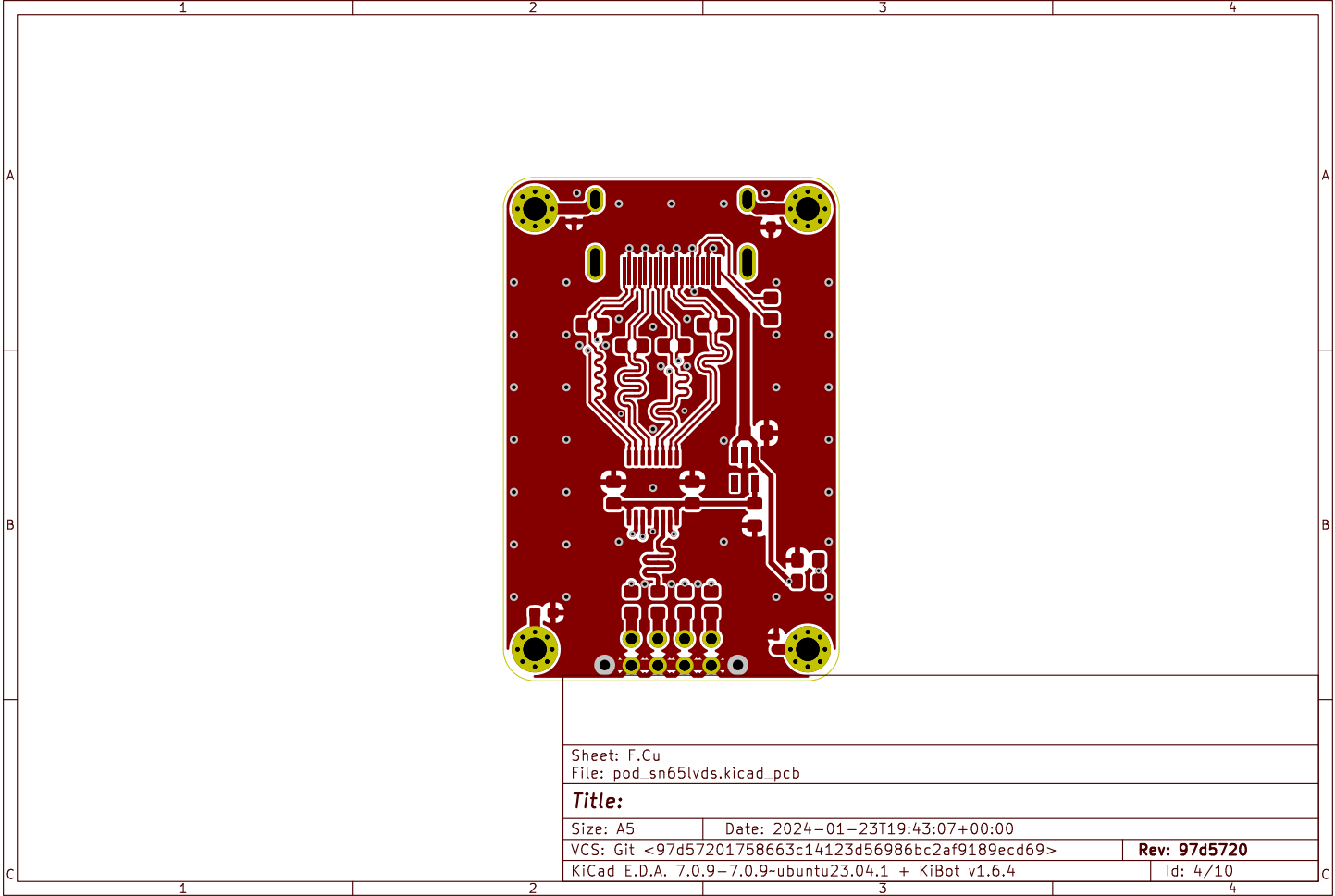


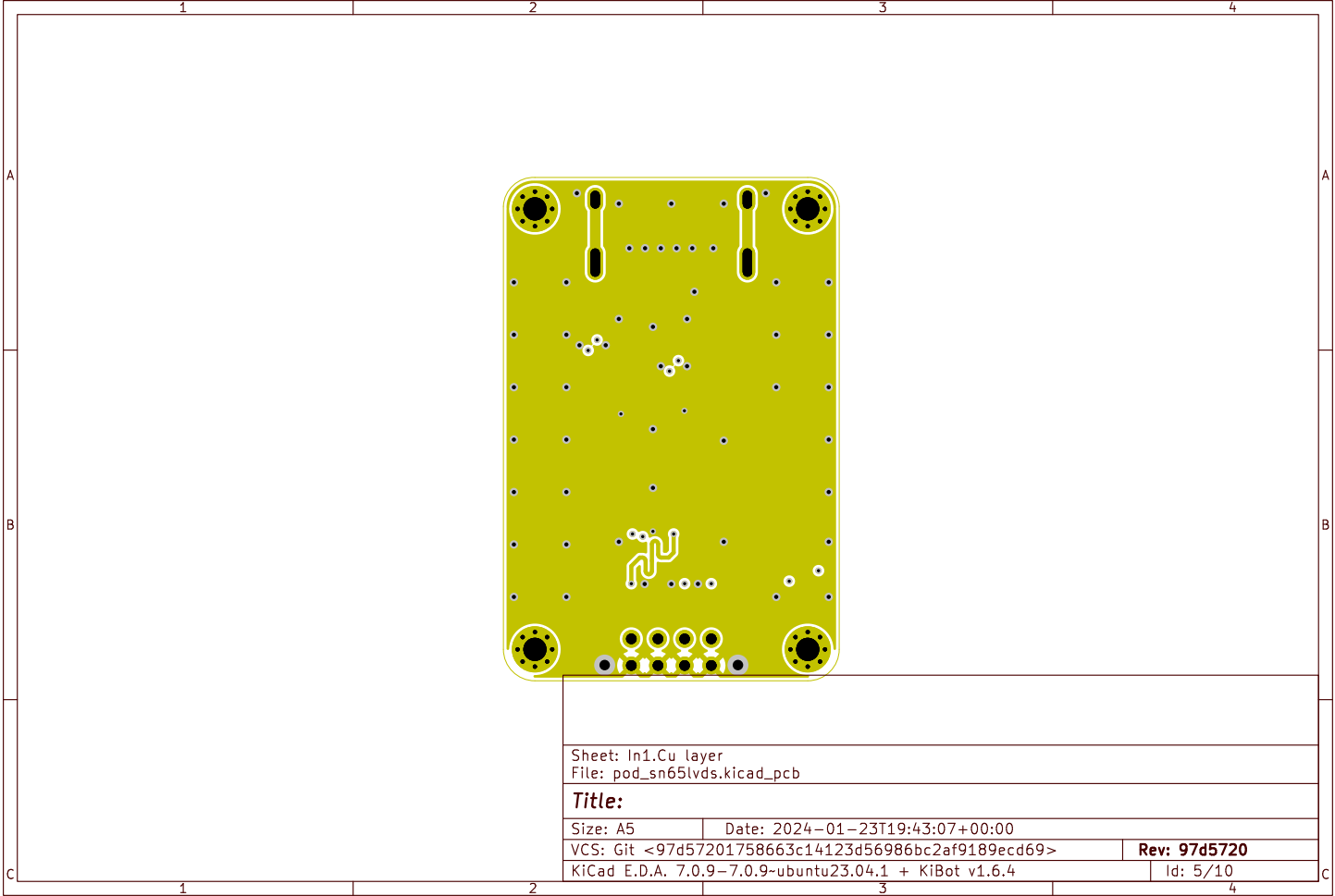
--	--	--

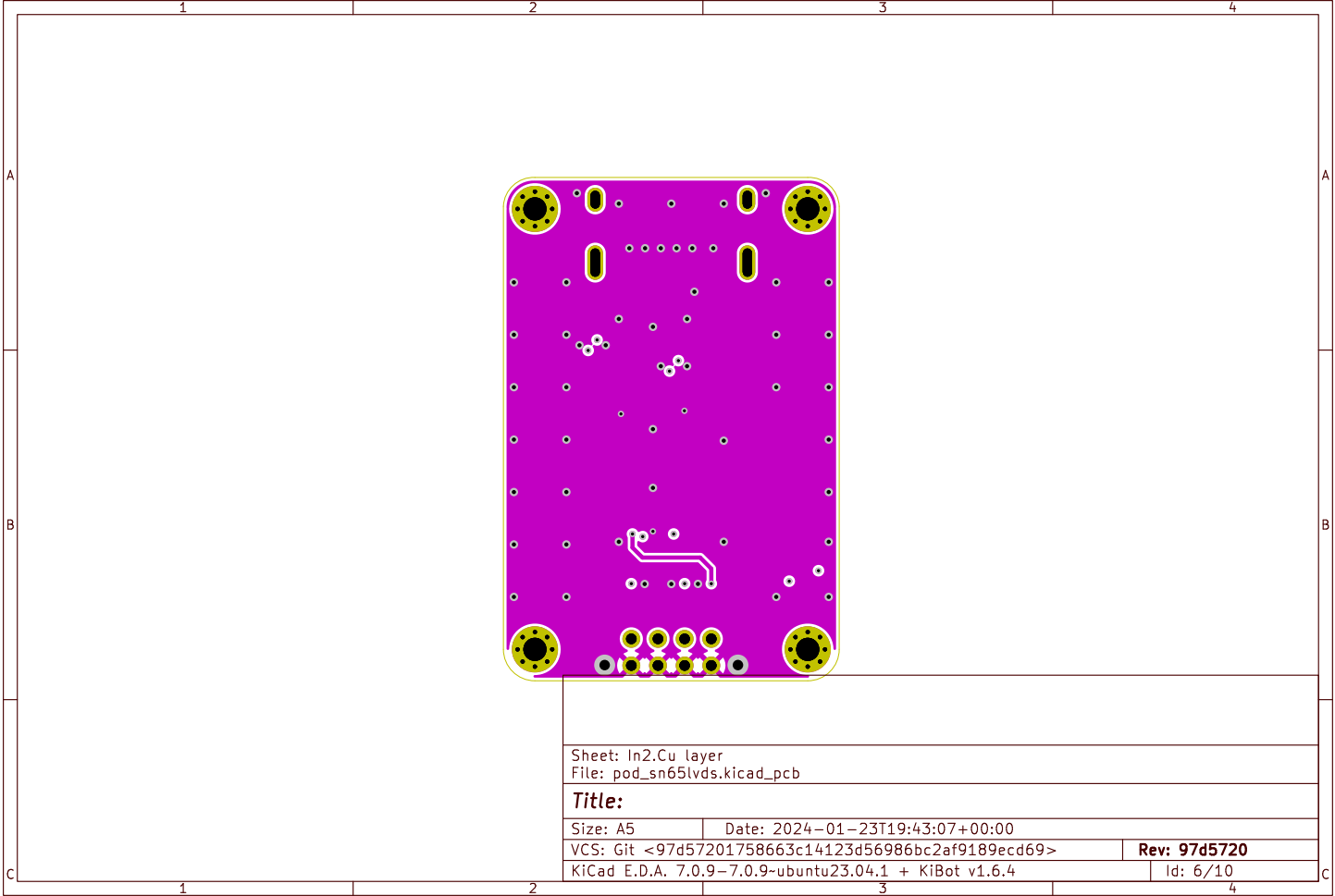
Sheet: Silkscreen top
File: pod_sn65lvds.kicad_pcb

Title:		
Size: A5	Date: 2024-01-23T19:43:07+00:00	
VCS: Git <97d57201758663c14123d56986bc2af9189ecd69>		Rev: 97d5720
KiCad E.D.A. 7.0.9-7.0.9-ubuntu23.04.1 + KiBot v1.6.4		Id: 3/10



Sheet: F.Cu			
File: pod_sn65lvds.kicad_pcb			
Title:			
Size: A5		Date: 2024-01-23T19:43:07+00:00	
VCS: Git <97d57201758663c14123d56986bc2af9189ecd69>			Rev: 97d5720
KiCad E.D.A. 7.0.9-7.0.9-ubuntu23.04.1 + KiBot v1.6.4			Id: 4/10





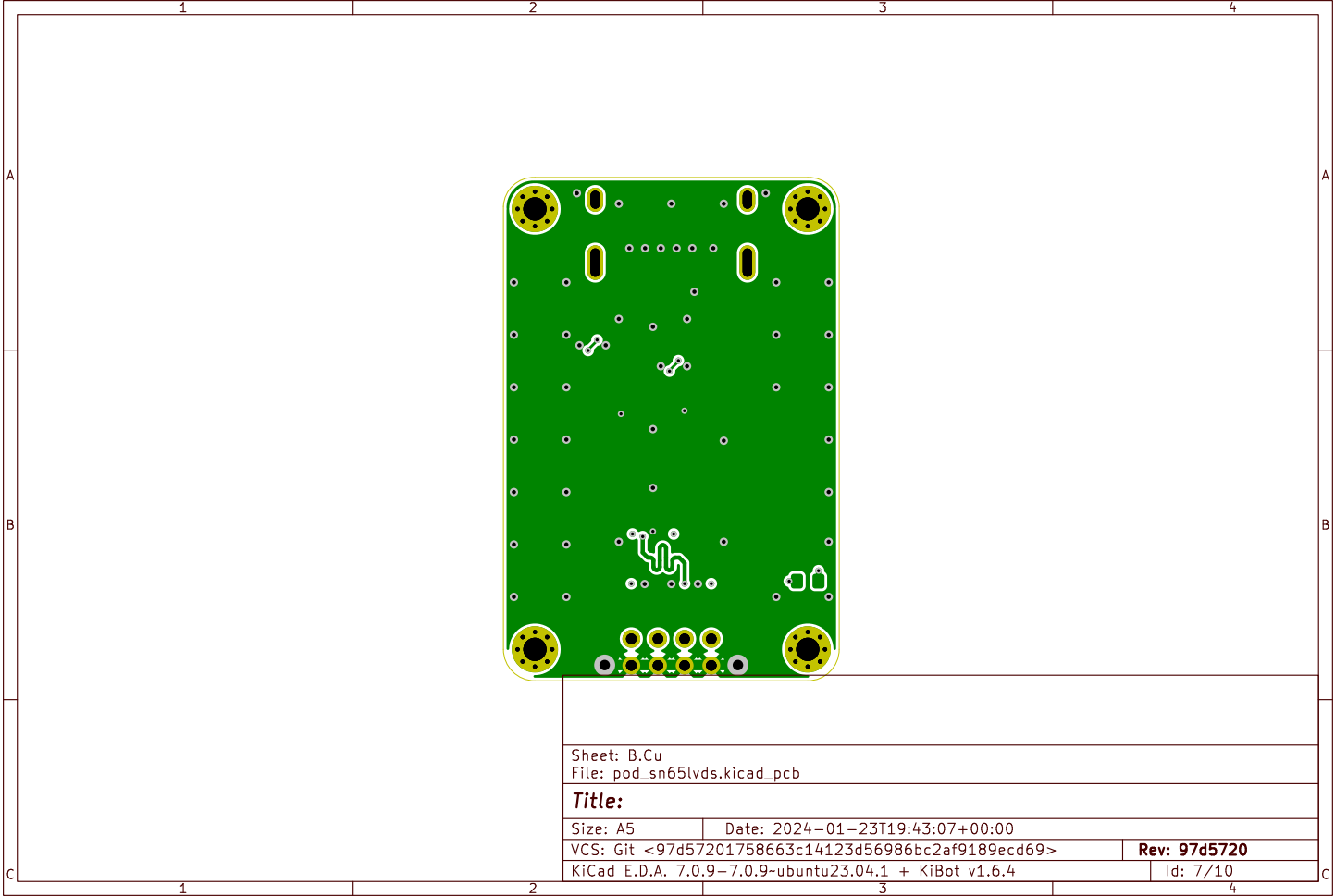
Sheet: In2.Cu layer
File: pod_sn65lvds.kicad_pcb

Title:

Size: A5 Date: 2024-01-23T19:43:07+00:00

VCS: Git <97d57201758663c14123d56986bc2af9189ecd69> **Rev: 97d5720**

KiCad E.D.A. 7.0.9-7.0.9-ubuntu23.04.1 + KiBot v1.6.4 Id: 6/10



Sheet: B,Cu
File: pod_sn65lvds.kicad_pcb

Title:

Size: A5	Date: 2024-01-23T19:43:07+00:00		
VCS: Git <97d57201758663c14123d56986bc2af9189ecd69>			Rev: 97d5720
KiCad E.D.A. 7.0.9-7.0.9-ubuntu23.04.1 + KiBot v1.6.4			Id: 7/10

