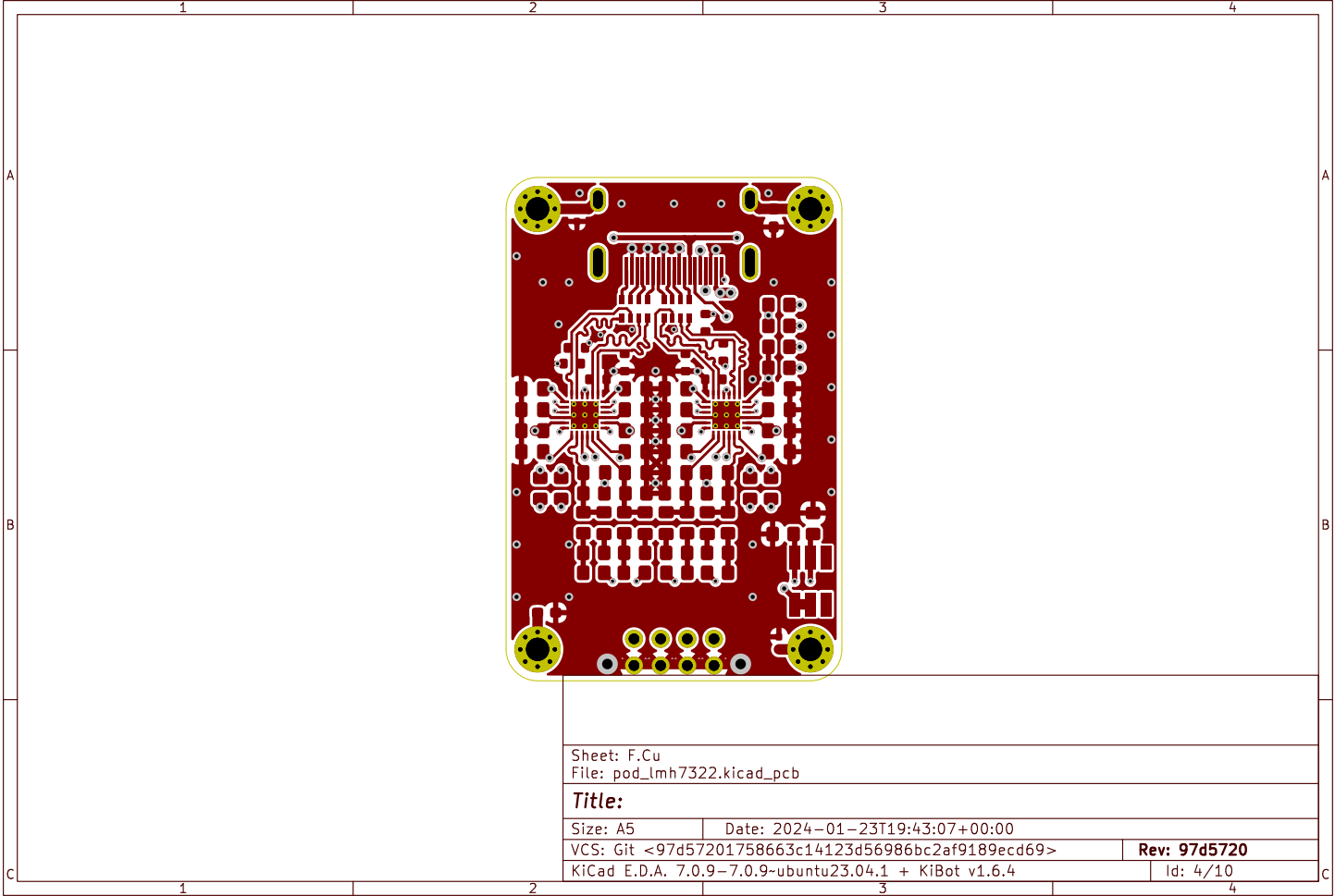




Title:

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

	3	4
--	---	---



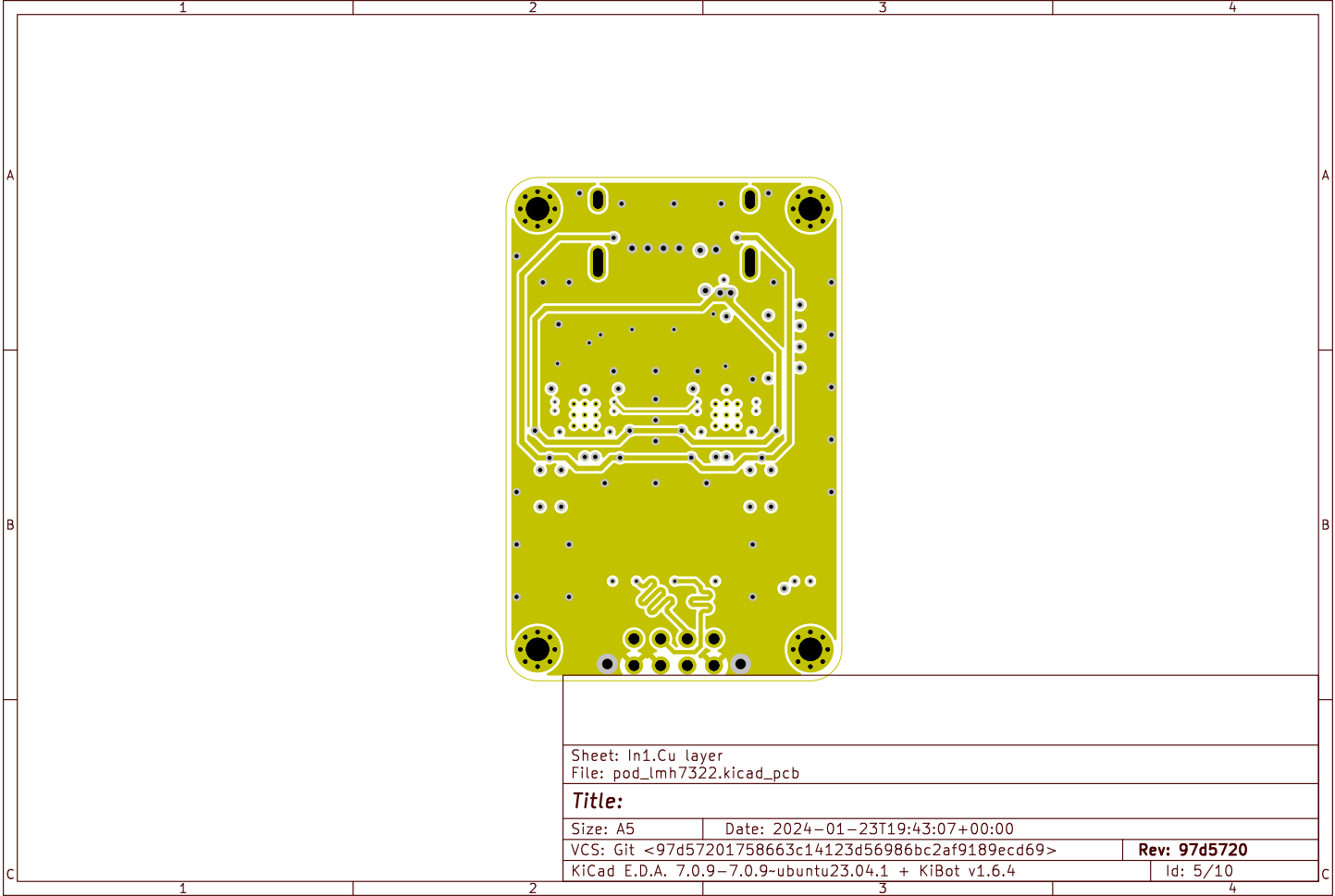
Sheet: F.Cu
File: pod_lmh7322.kicad_pcb

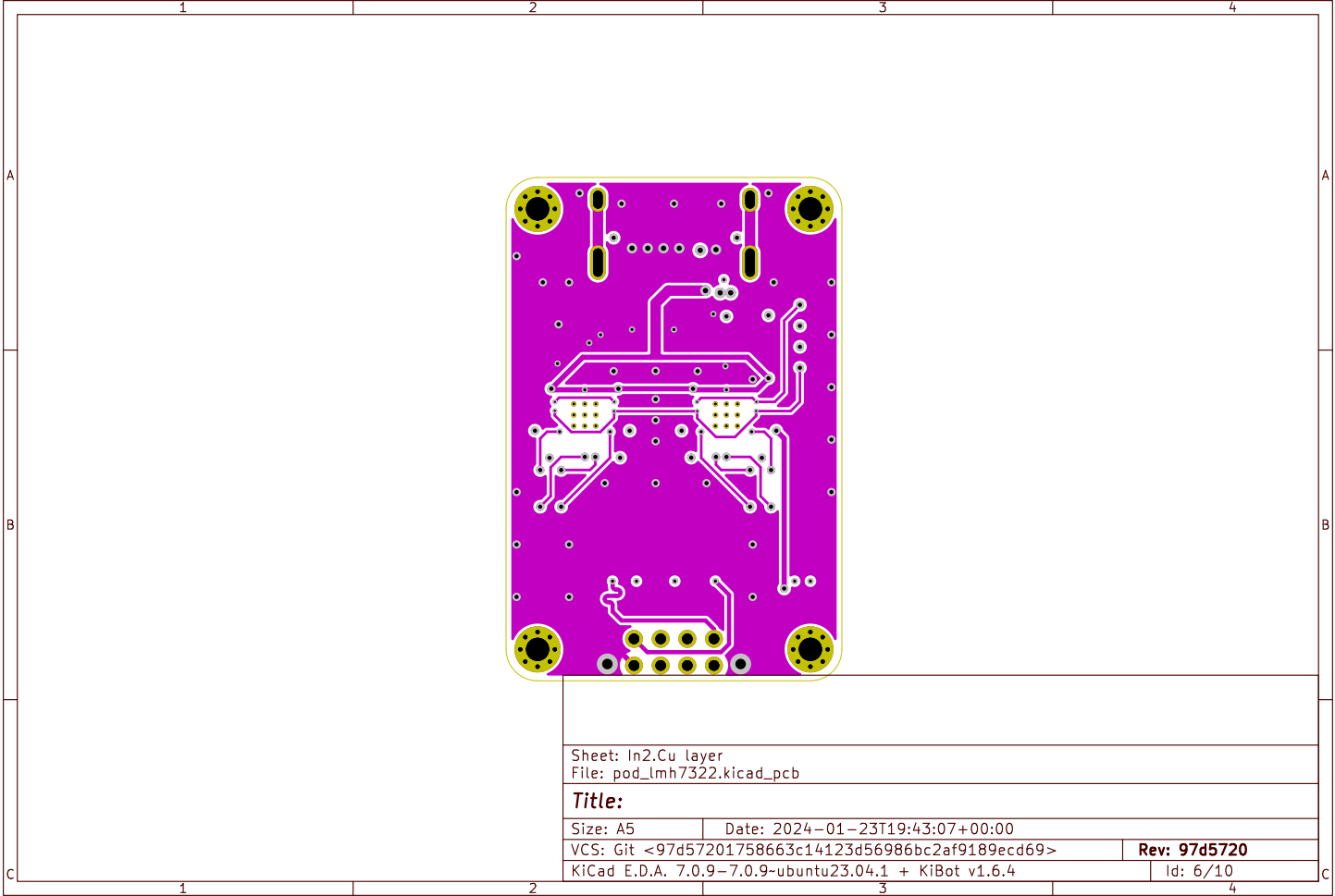
Title:

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

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

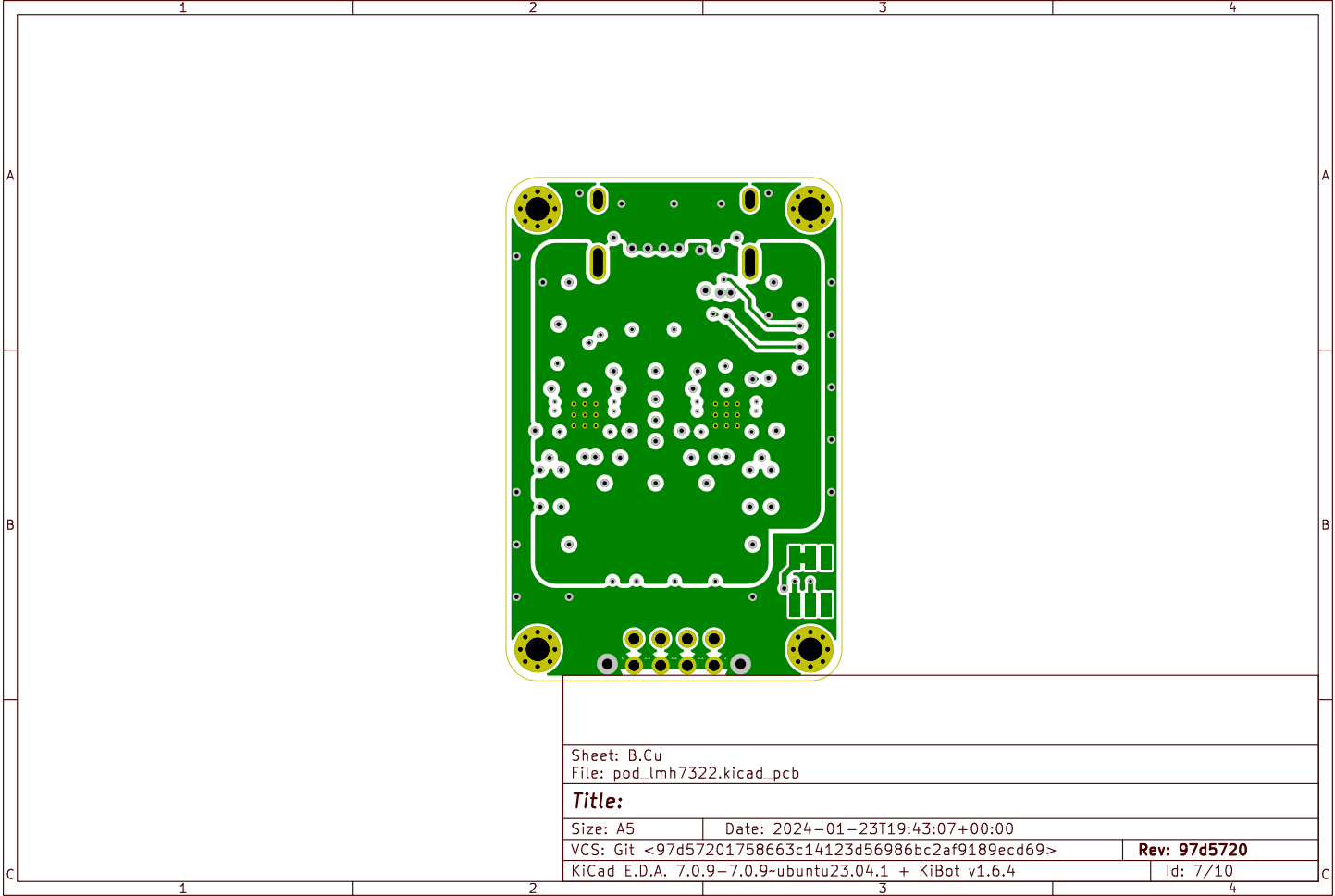
KiCad E.D.A. 7.0.9-7.0.9-ubuntu23.04.1 + KiBot v1.6.4	Id: 4/10
-------------------------------------------------------	----------





Sheet: In2.Cu layer
File: pod_lmh7322.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_lm7322.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
-------------------------------------------------------	----------

