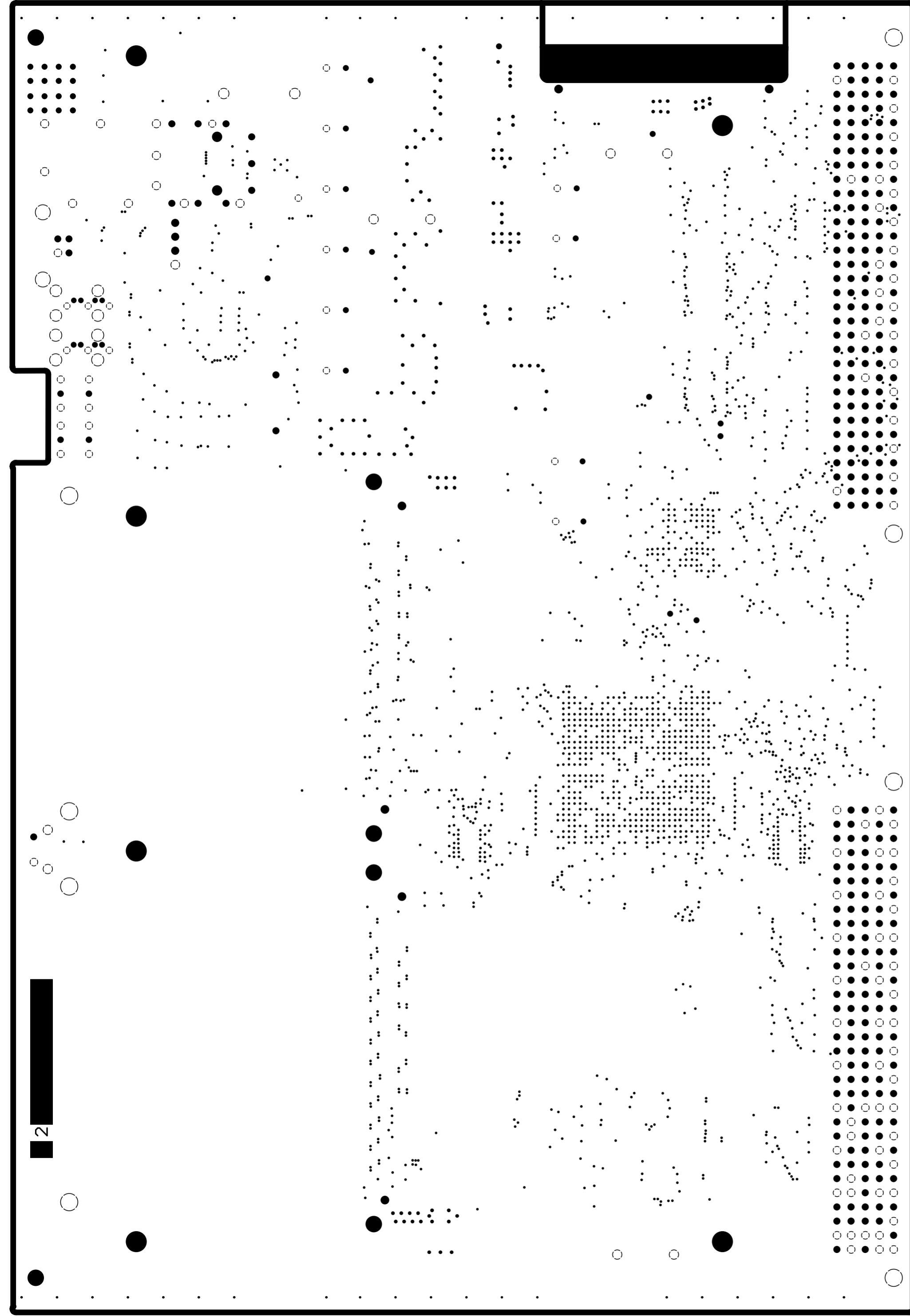
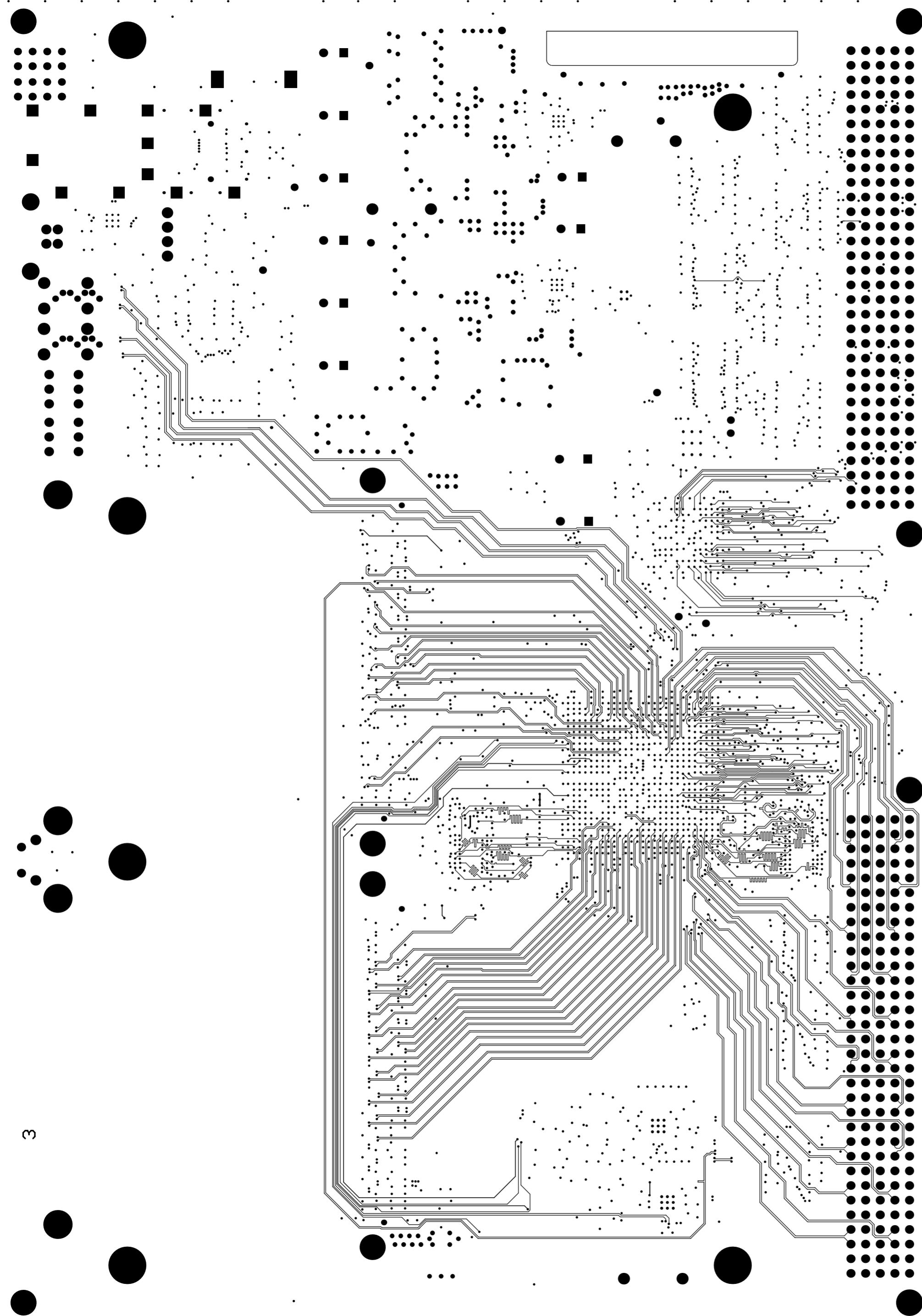
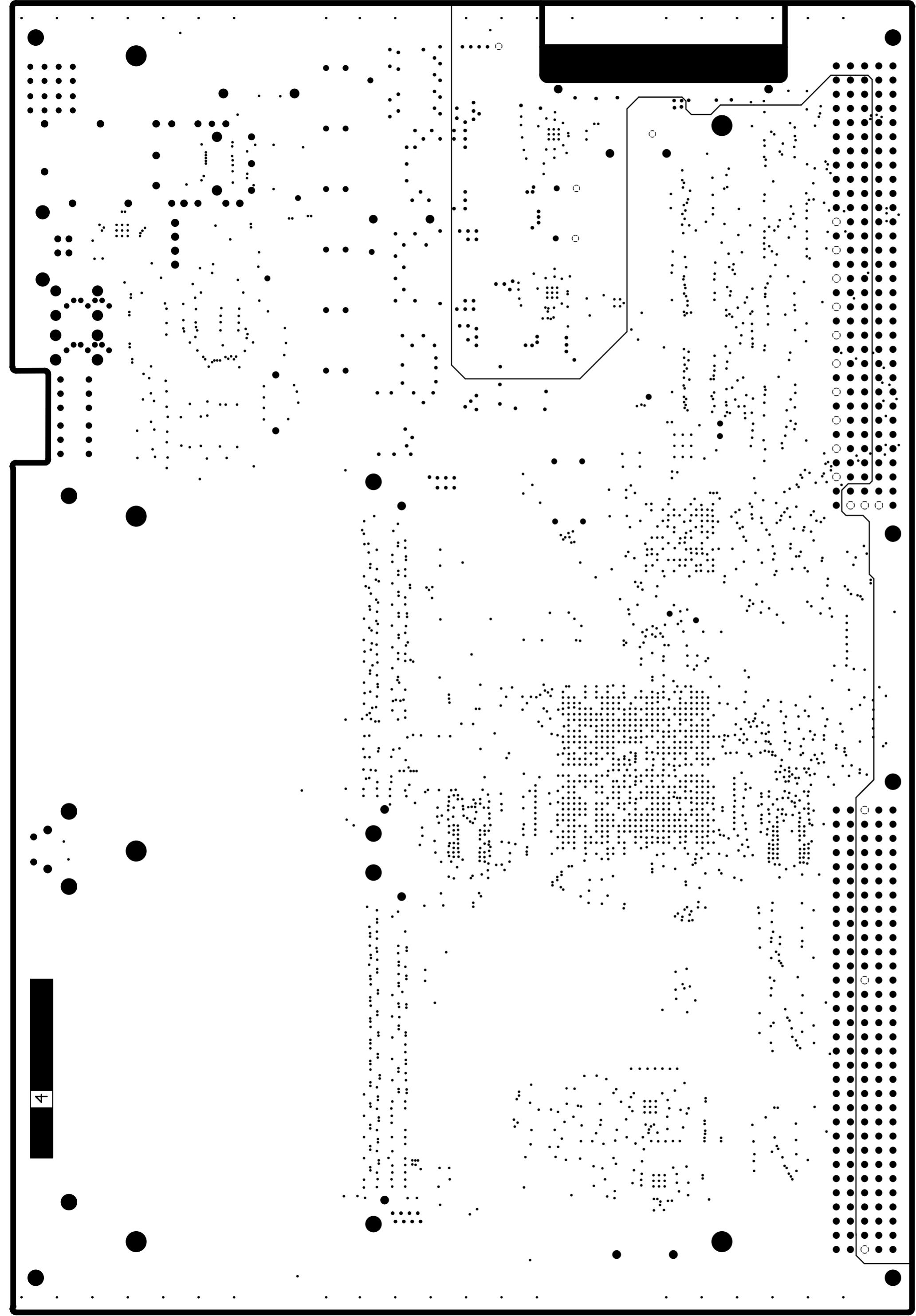
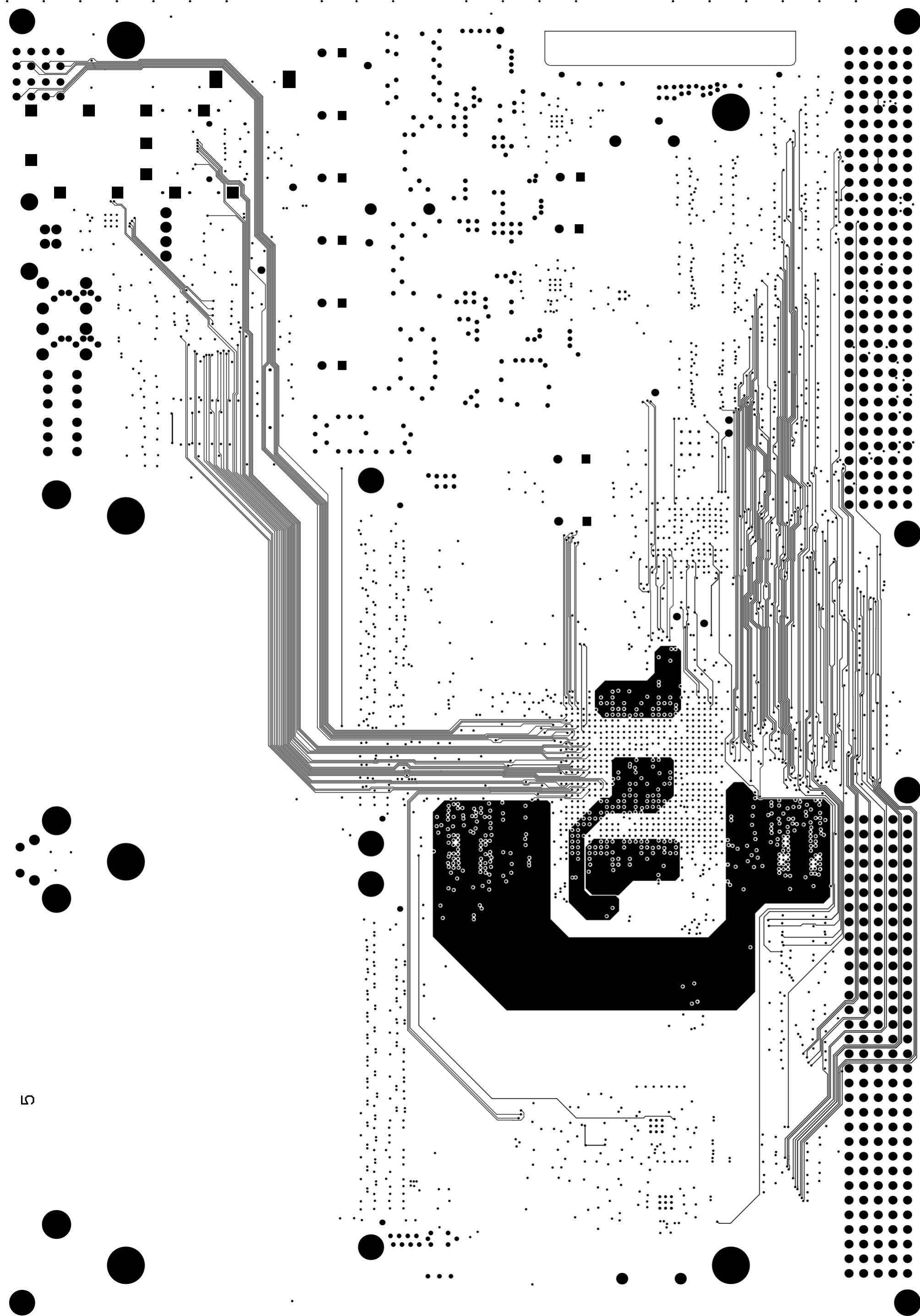


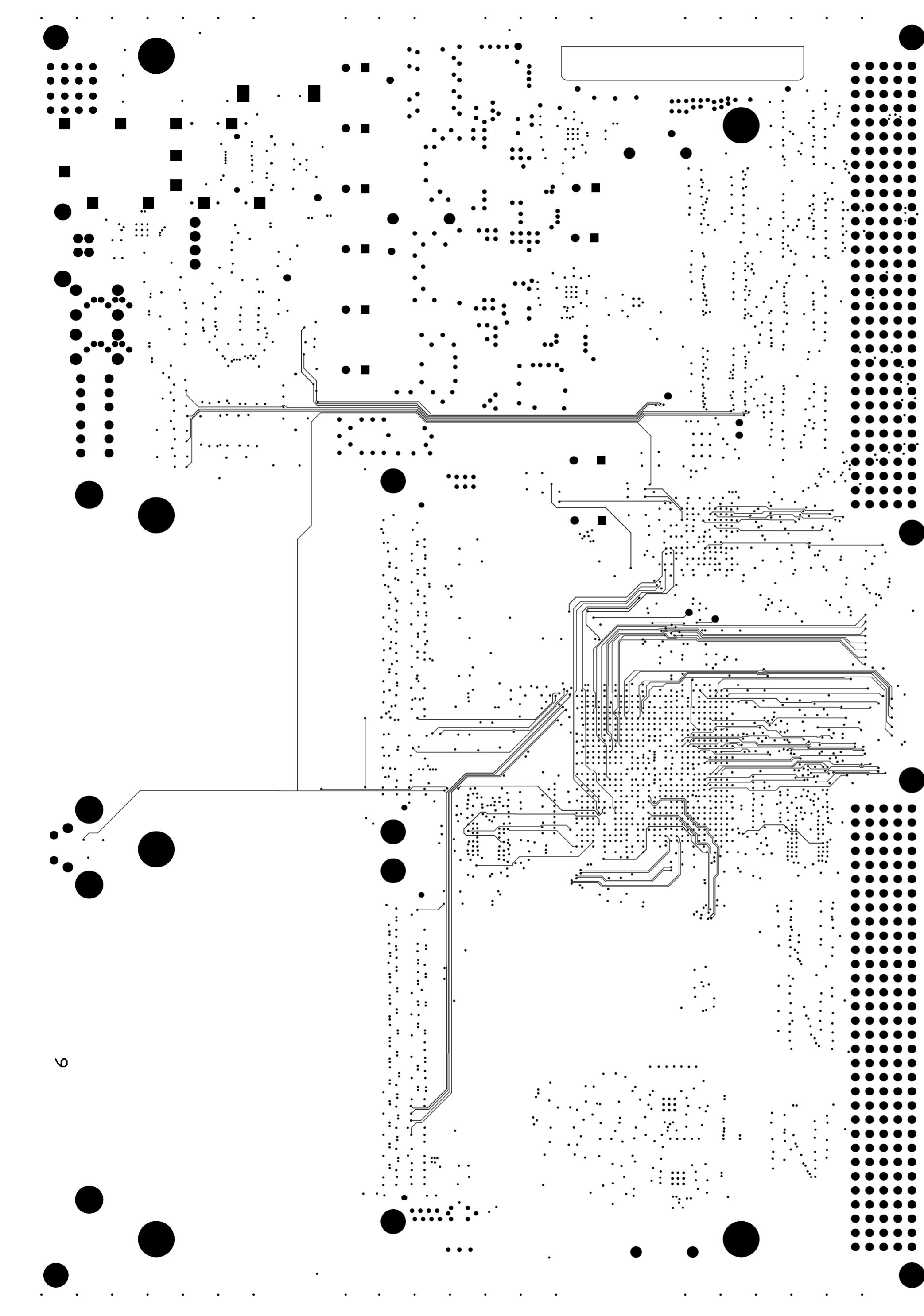
Layer Stack Up Detail for: SUEC.pcbdoc					
Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant
Top Layer	(.GTL)	0.709mil	3.937mil	FR-4	4.60
L2 GND	(.GP1)	0.709mil	5.905mil	FR-4	4.80
L3	(.G1)	0.709mil	5.905mil	FR-4	4.80
L4 PWR3.3	(.GP2)	0.709mil	3.937mil	FR-4	4.80
L5	(.G2)	0.709mil	11.811mil	FR-4	4.80
L6	(.G3)	0.709mil	3.937mil	FR-4	4.80
L7 GND	(.GP3)	0.709mil	5.905mil	FR-4	4.80
L8	(.G4)	0.709mil	5.905mil	FR-4	4.80
L9 PWR	(.GP4)	0.709mil	3.937mil	FR-4	4.80
Bottom Layer	(.GBL)	0.709mil	0.394mil	Solder Resist	3.50
Bottom Solder Mask	(.GBS)				

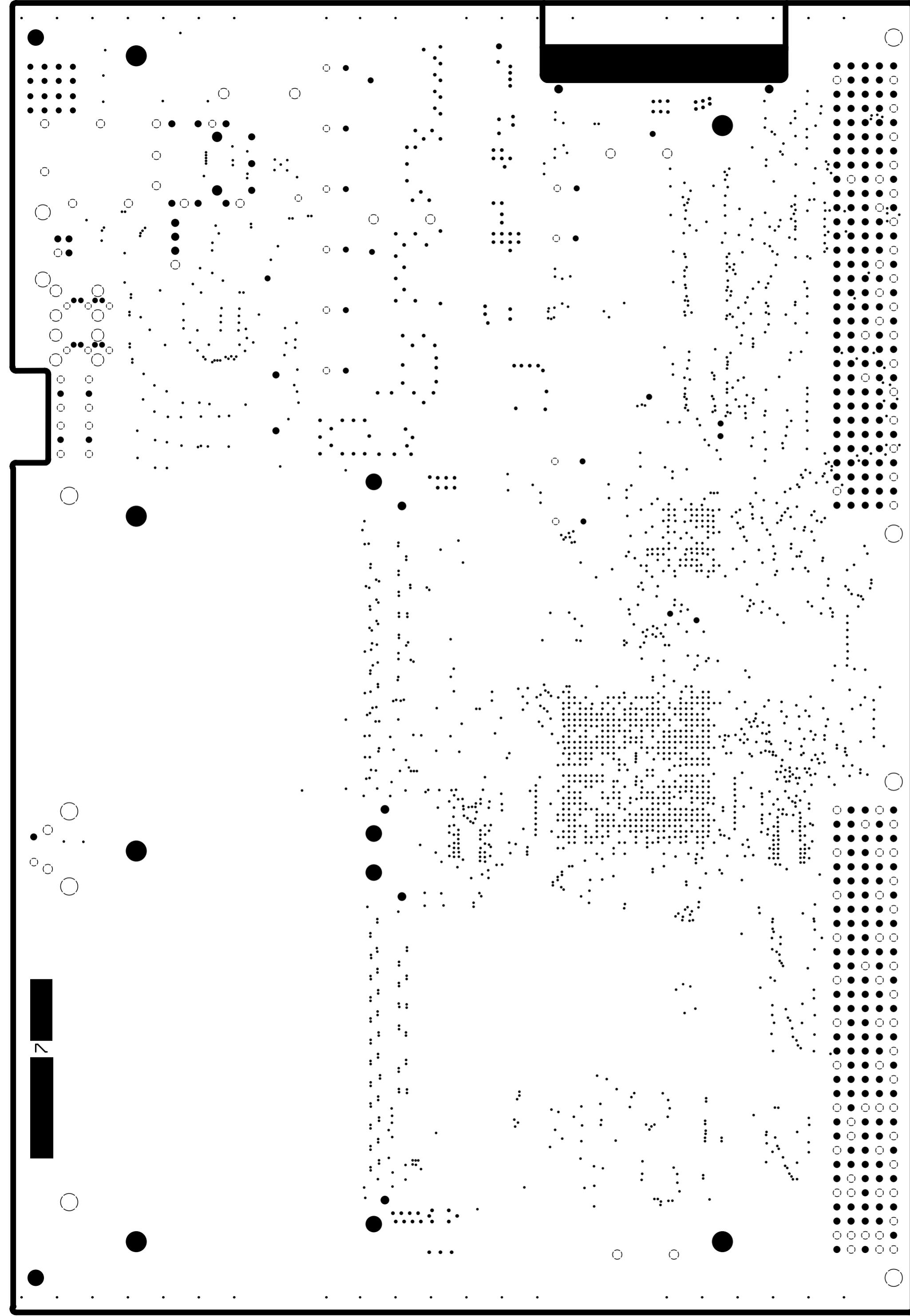


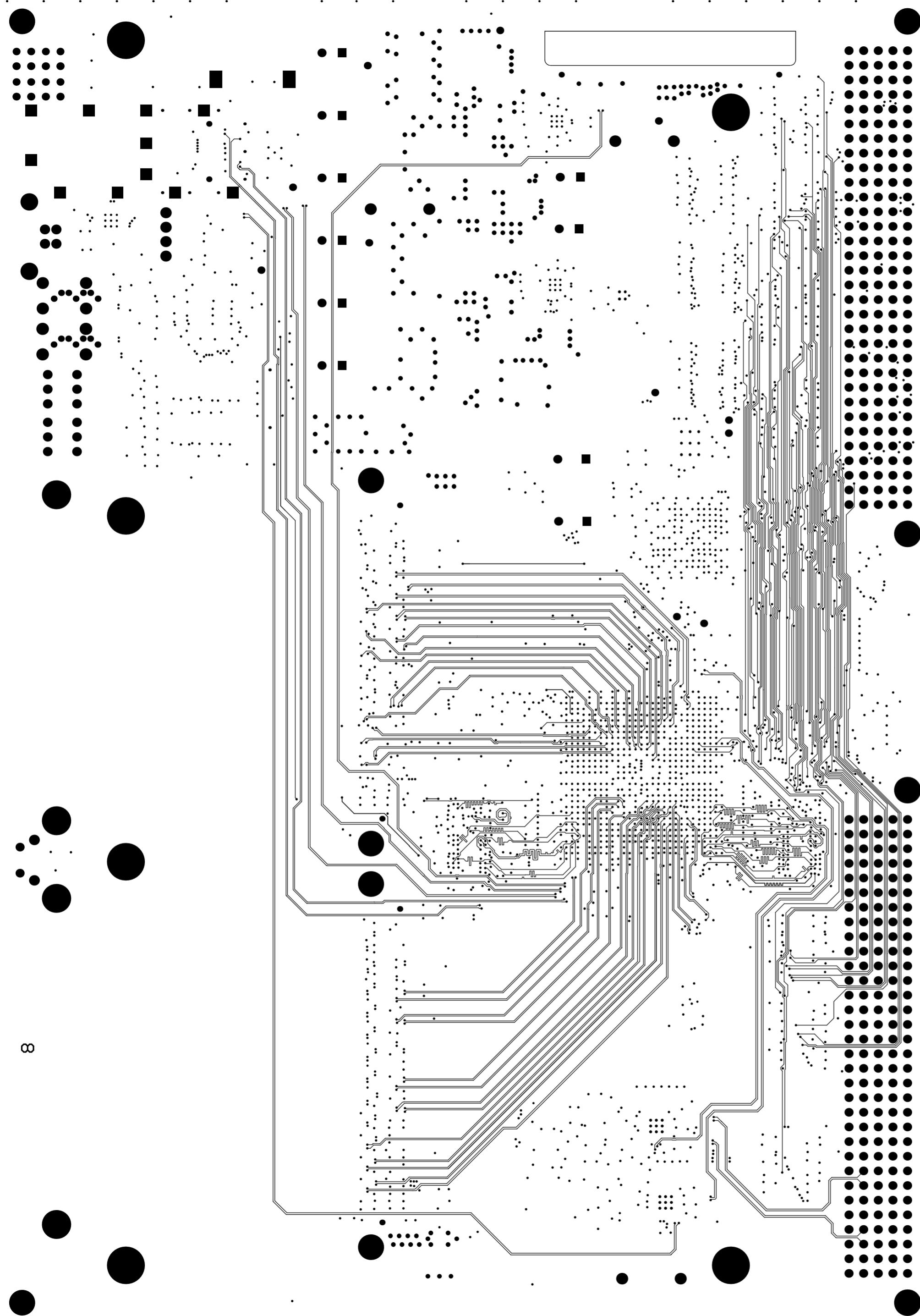


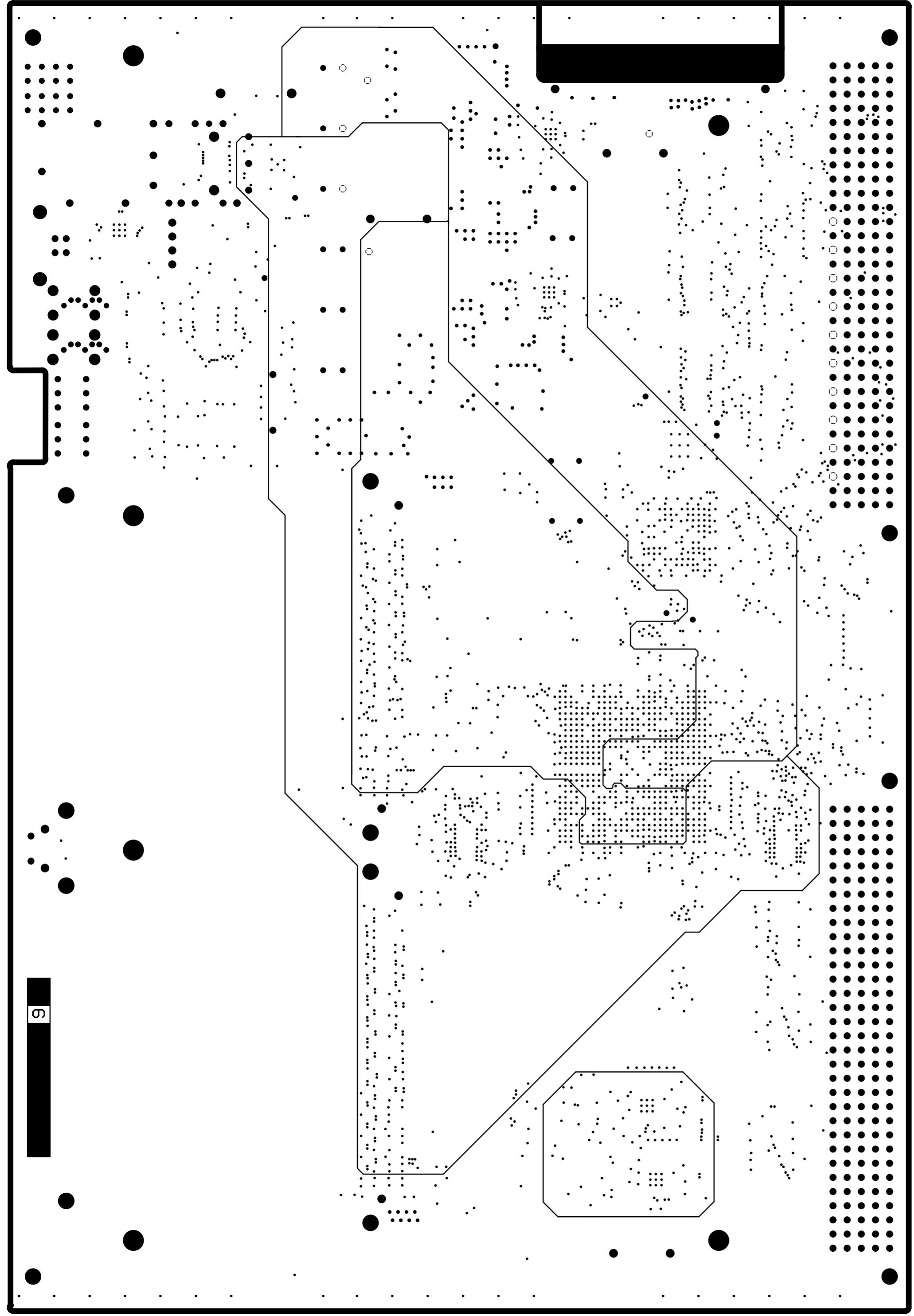


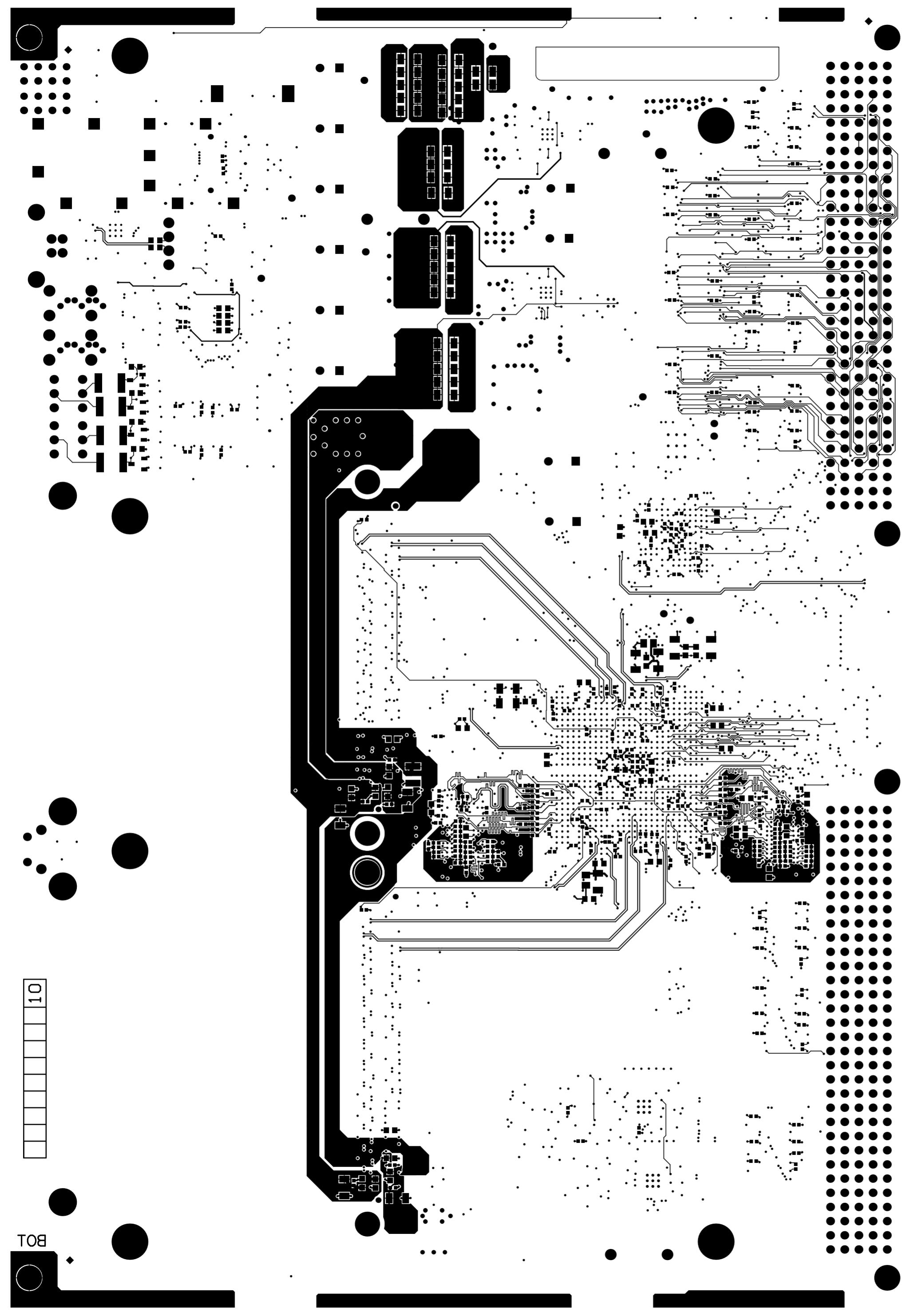


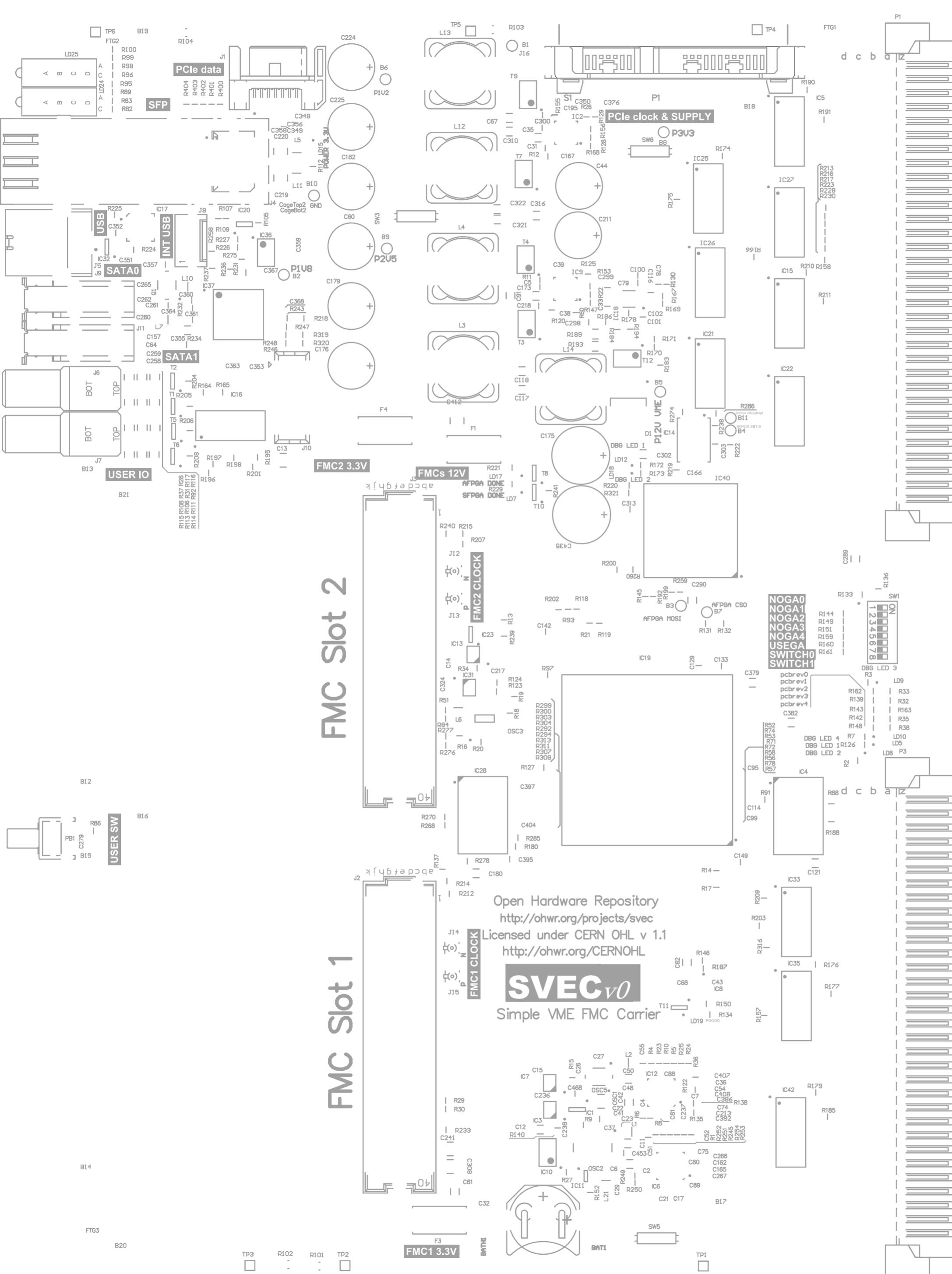












LOG

