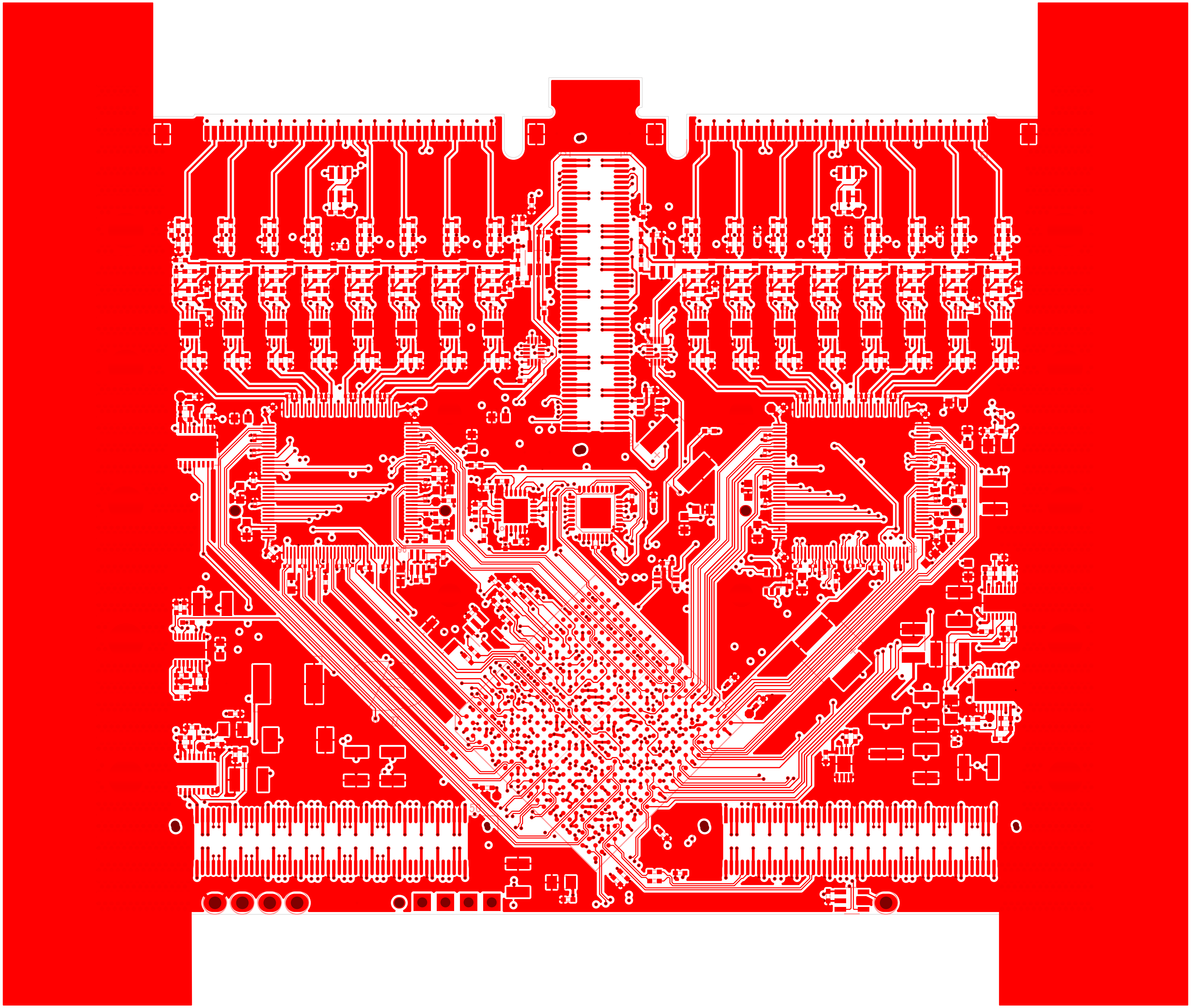
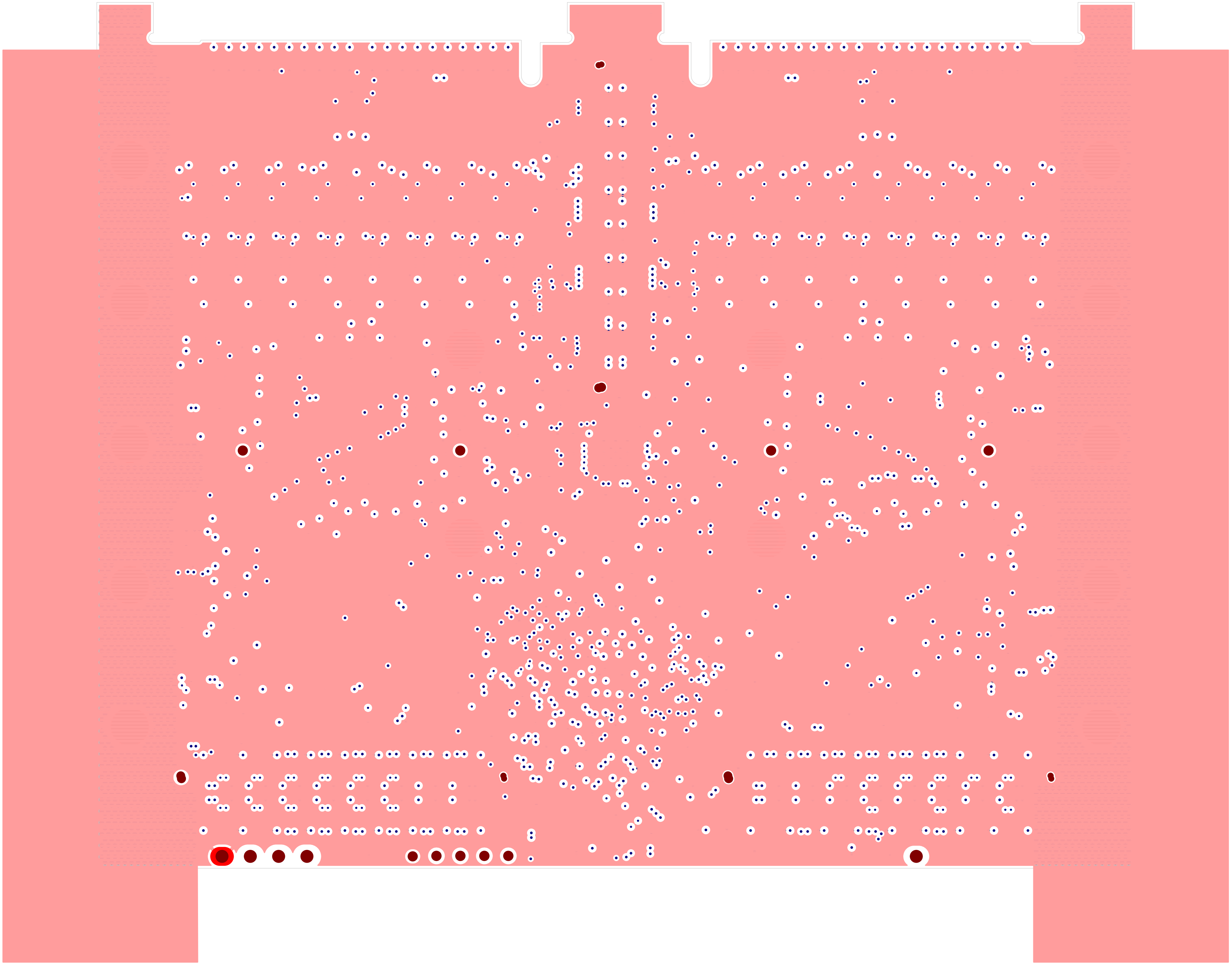
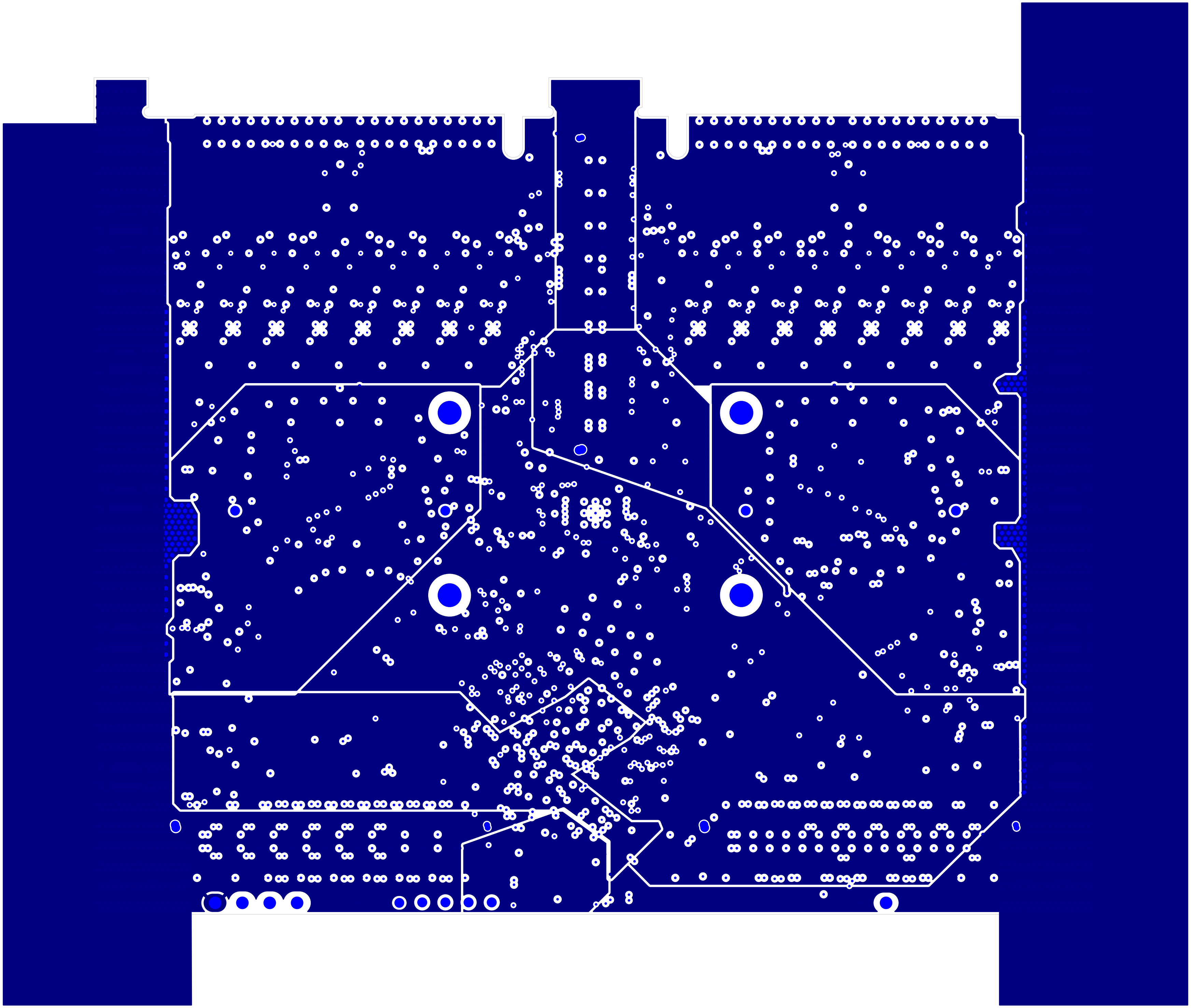


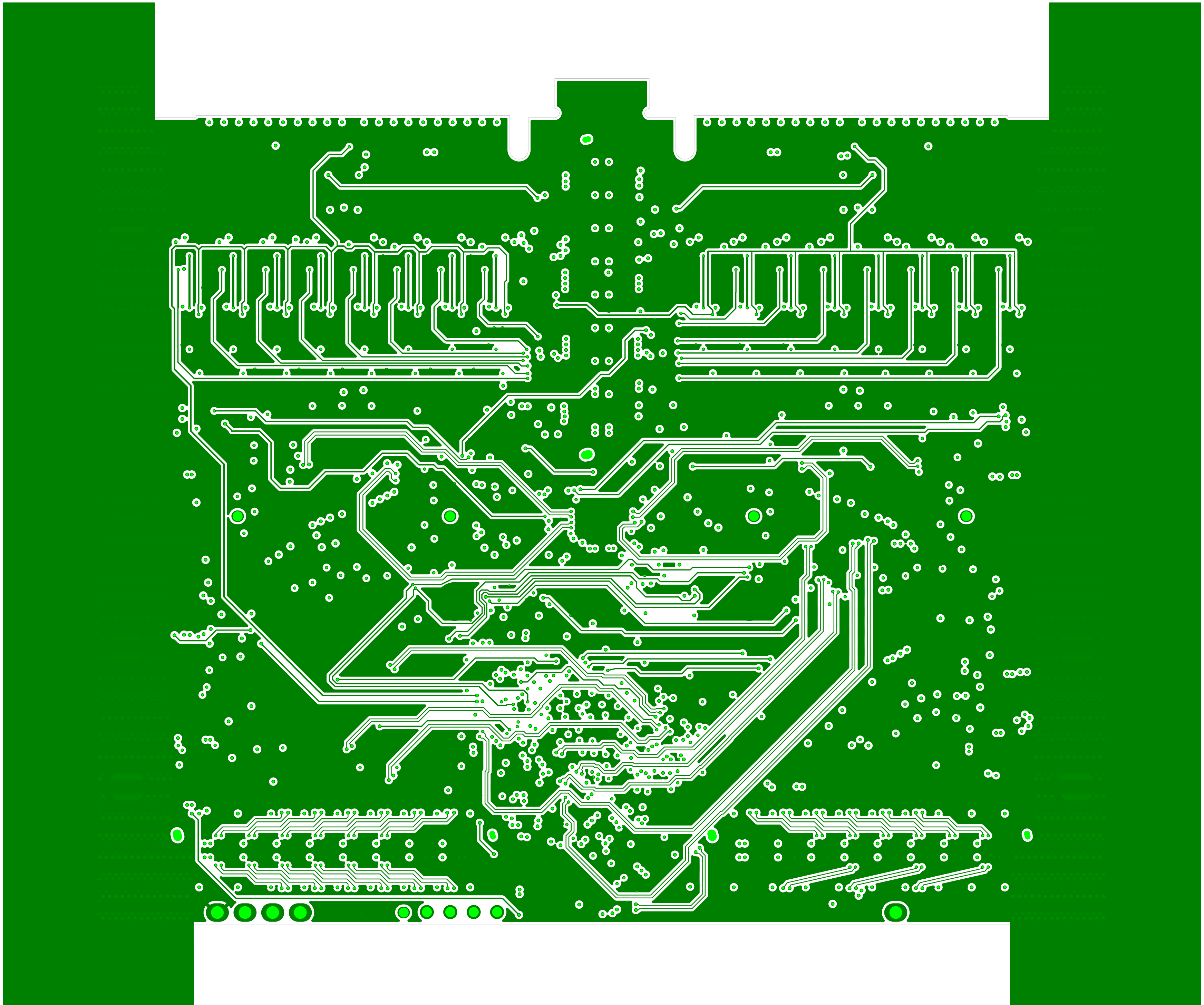
top



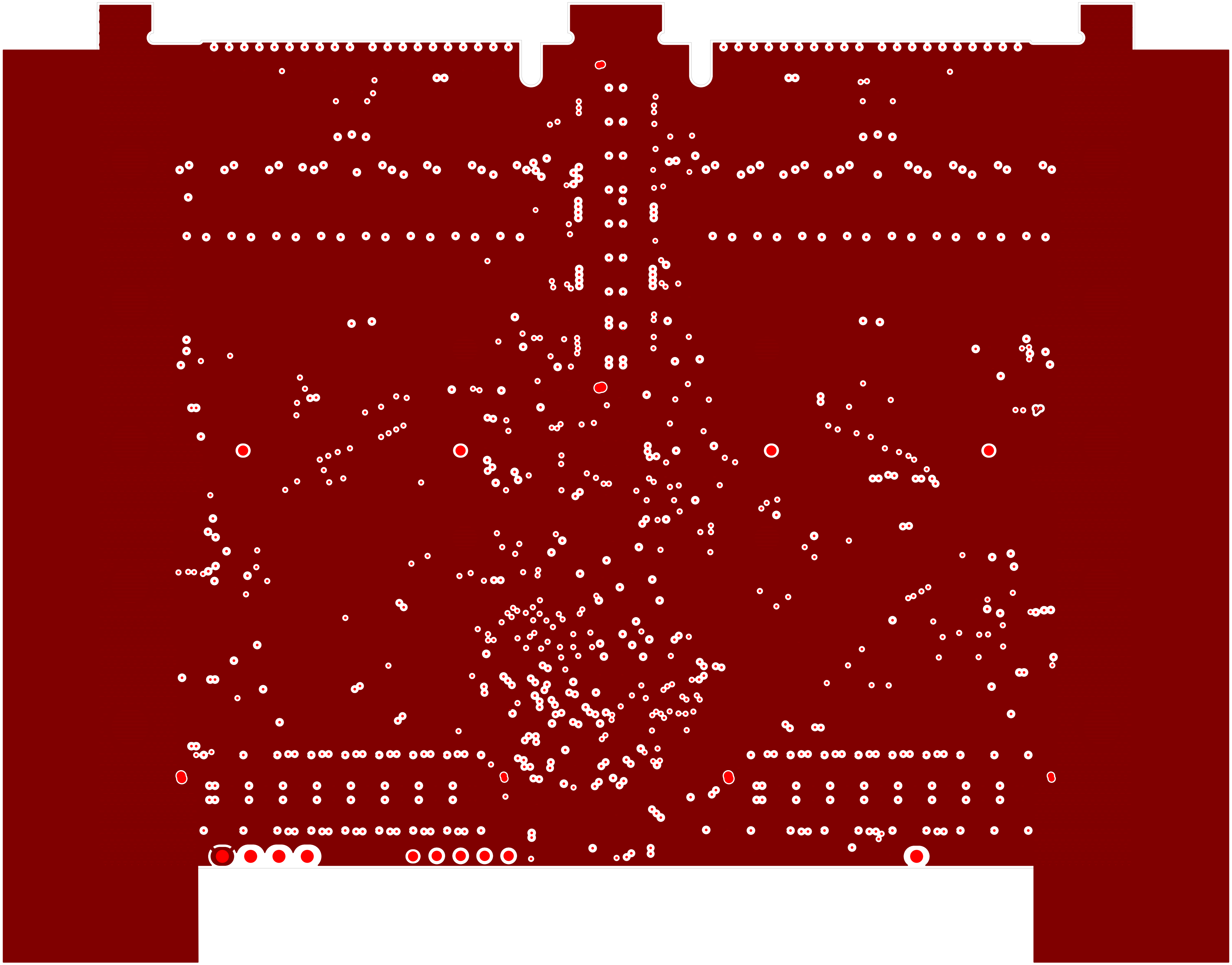
02-GND



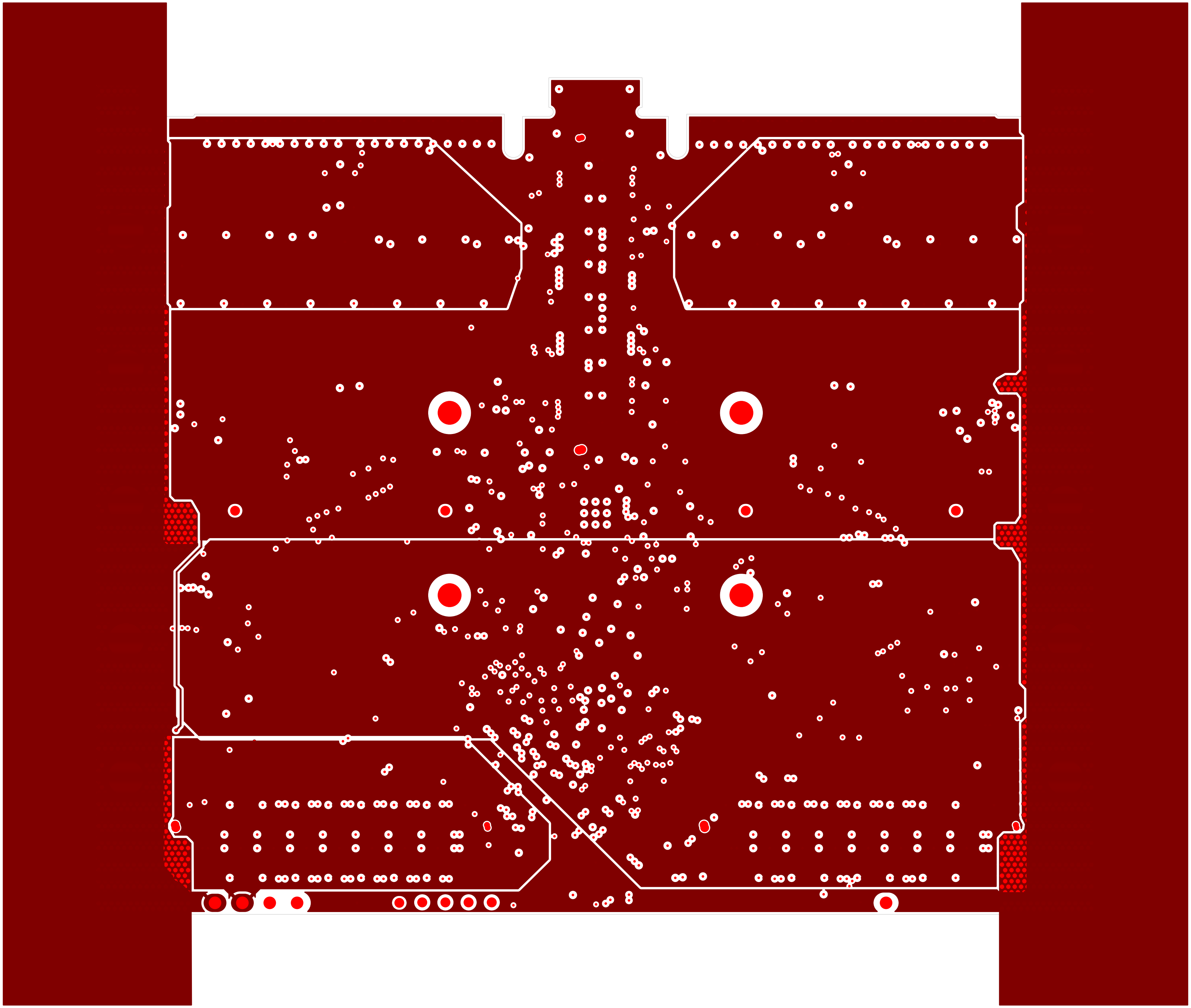
03-power



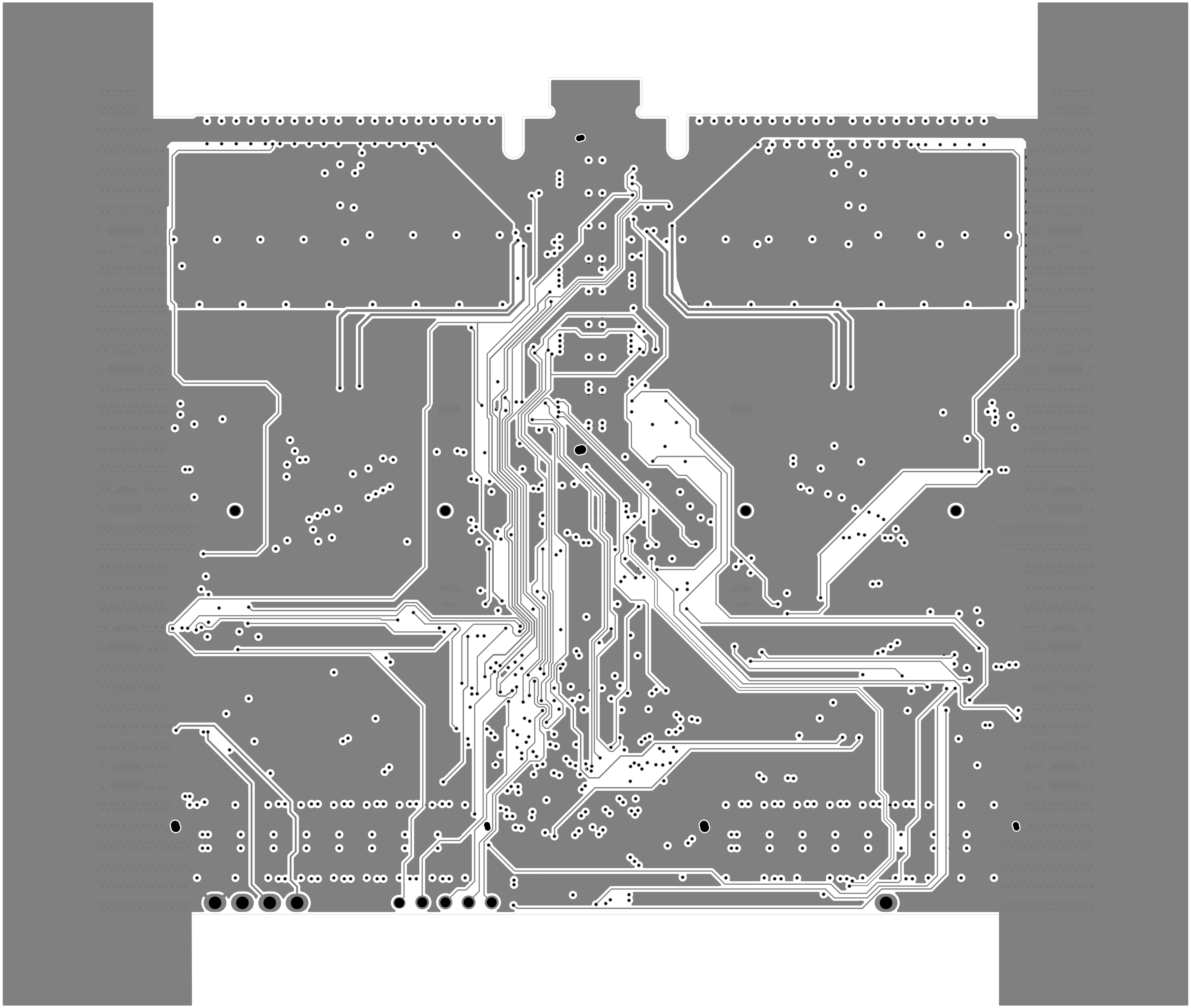
04-routing



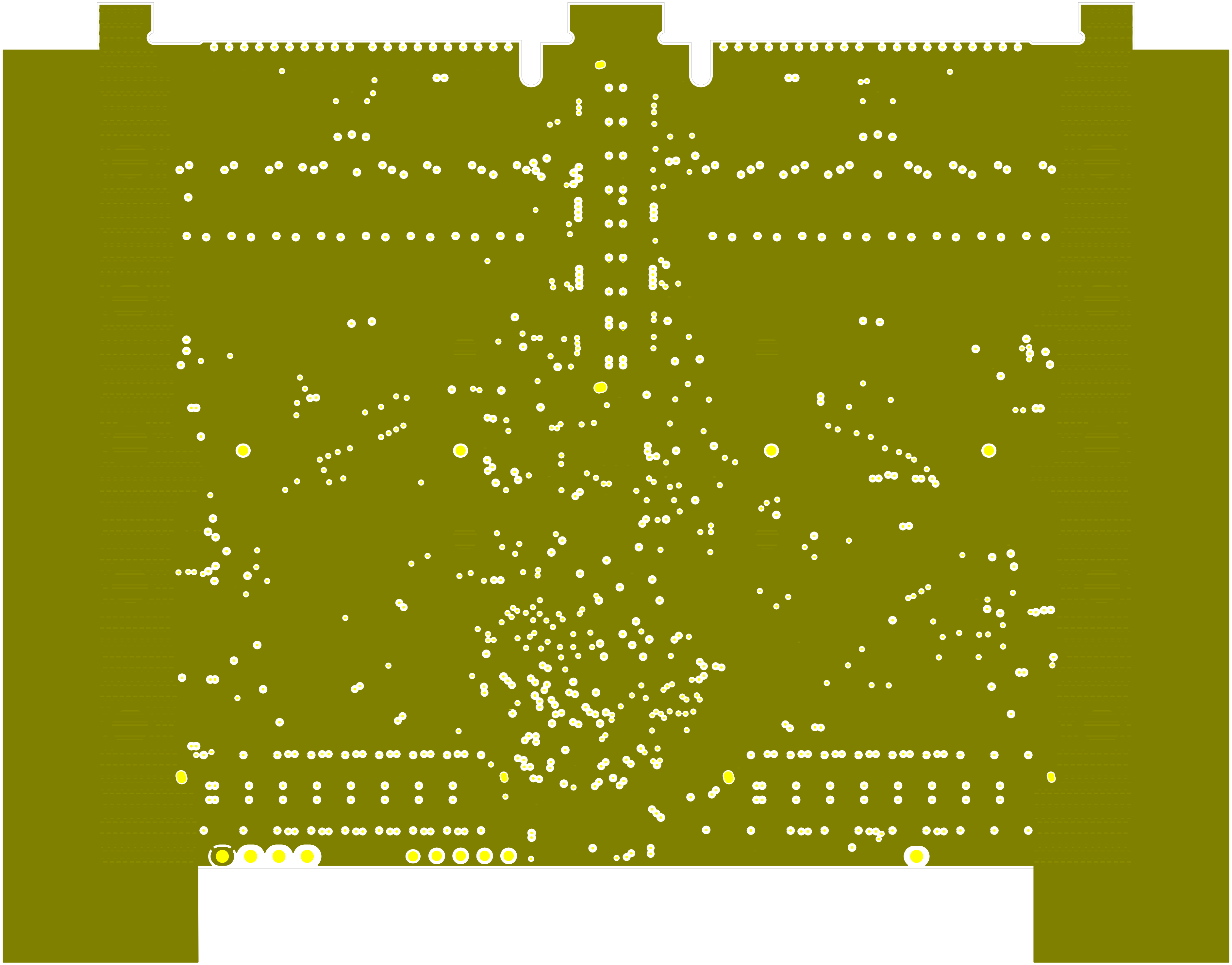
05-GND



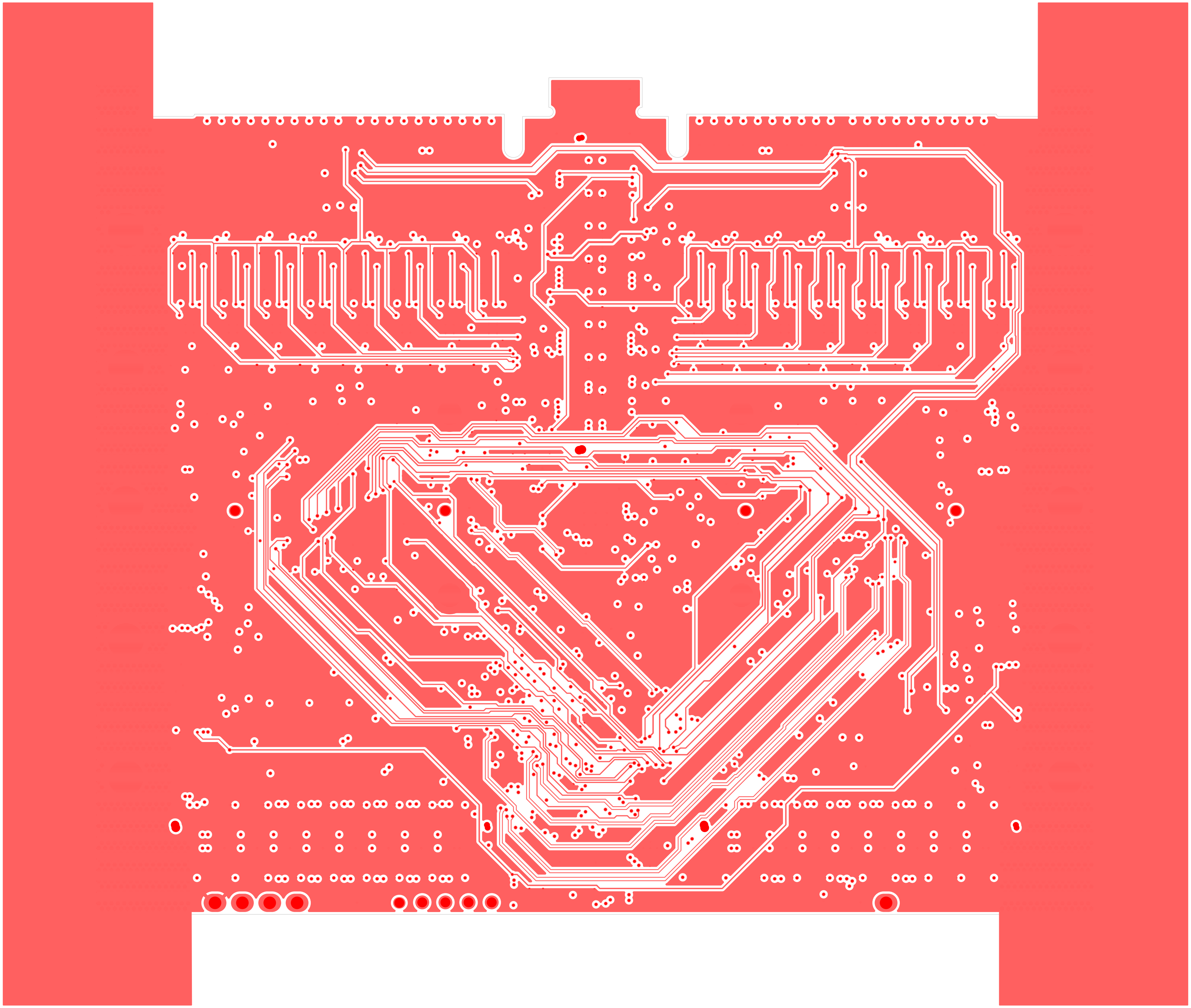
06-power



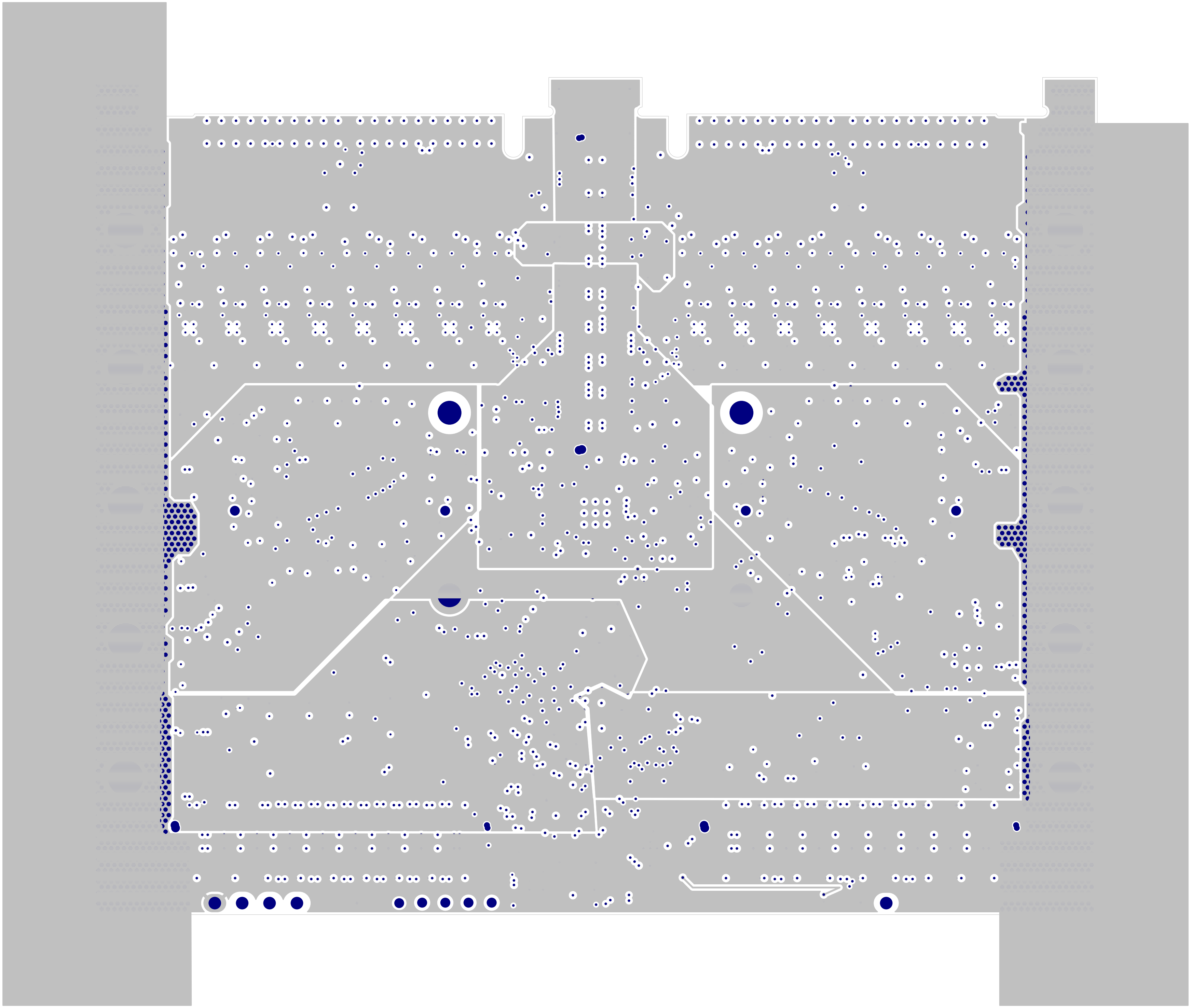
07-routing



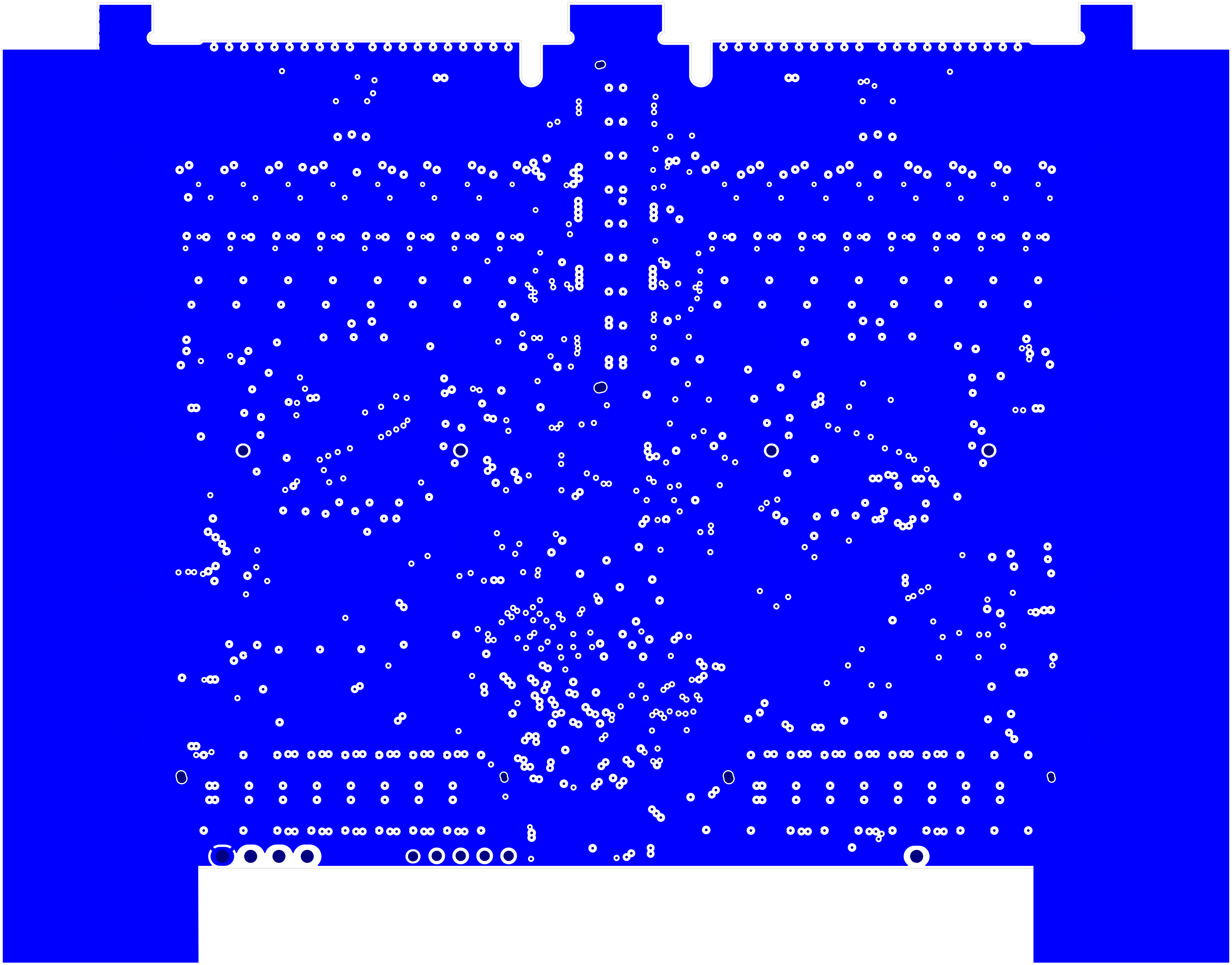
08-GND



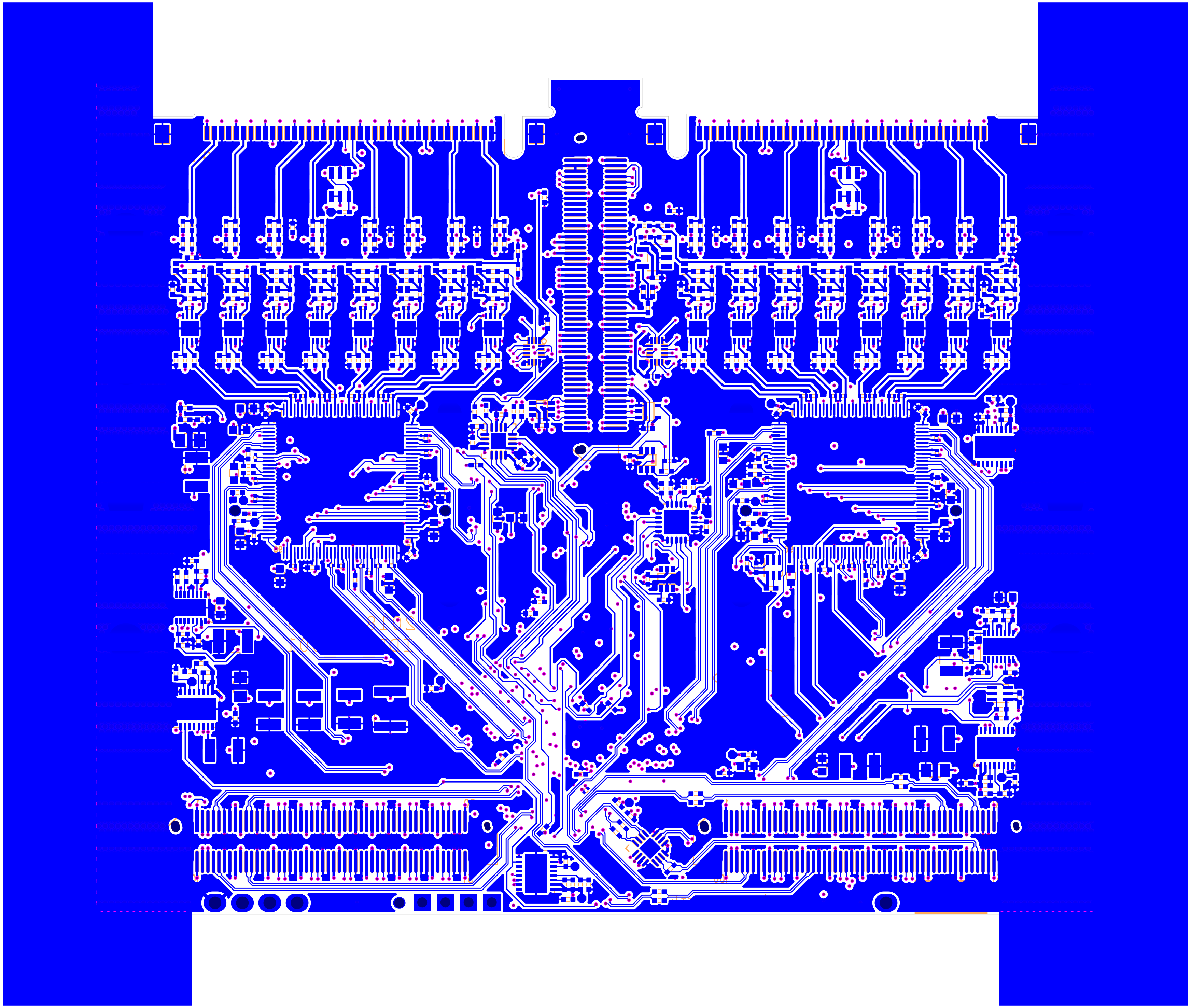
09-routing



10-power



11-GND



bottom

board material = FR4 180 C Tg
overall board dimensions = 110 mm * 91.75 mm
soldermask color = red
silkscreen color = white
plating = ENIG
plated edges: left and right (see file edge-plating.gerber)
left/right edge plating should connect electrically to all inner layers
all plated holes plated to at least 1 mil copper wall thickness, or more if possible
**via-in-PAD plating on top and bottom layers (see files top-via-in-pad.gerber and bottom-via-in-pad.gerber for locations)**
**board to be fabricated using sequential lamination (layers 1-4, layers 9-12, then entire board)**

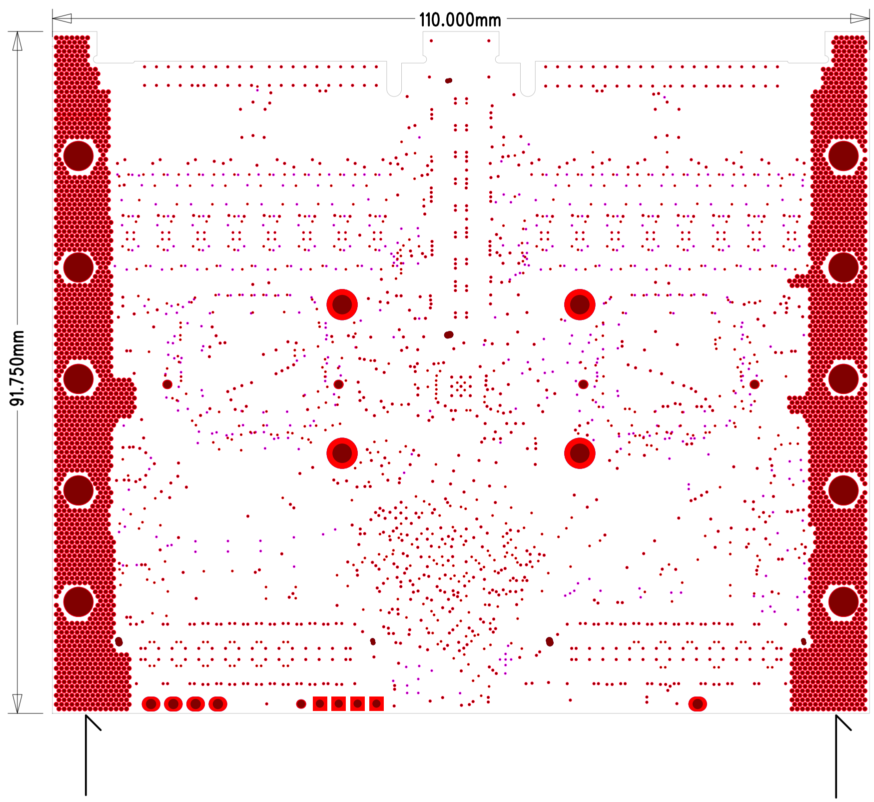
| dielectric thicknesses: | copper gerber layers: |
|---|---|
| | top-copper.GTL - copper foil (1 oz finished) |
| prepreg 0.127 mm | |
| | 2-copper.G2L - copper (2 oz) |
| core 0.080 mm | |
| | 3-copper.G3L - copper (2 oz) |
| prepreg 0.152 mm | |
| | 4-copper.G4L - copper foil (0.5 oz finished) |
| prepreg 0.152 mm | |
| | 5-copper.G5L - copper (1 oz) |
| core 0.080 mm | |
| | 6-copper.G6L - copper (1 oz) |
| prepreg 0.051 mm | |
| | 7-copper.G7L - copper (1 oz) |
| core 0.080 mm | |
| | 8-copper.G8L - copper (1 oz) |
| prepreg 0.152 mm | |
| | 9-copper.G9L - copper foil (0.5 oz finished) |
| prepreg 0.152 mm | |
| | 10-copper.G10L - copper (2 oz) |
| core 0.080 mm | |
| | 11-copper.G11L - copper (2 oz) |
| prepreg 0.127 mm | |
| | bottom-copper.GBL - copper foil (1 oz finished) |

silkscreen gerber layers:
top-silkscreen.GTO
bottom-silkscreen.GBO

soldermask gerber layers:
top-soldermask.GTS
bottom-soldermask.GBS

other gerber layers:
board-outline.GKO - showing outline of board
drill-drawing.gerber - this file
edge-plating.gerber - showing where edge plating should be added (see note to left
top-via-in-pad.gerber
bottom-via-in-pad.gerber
soldermask-plugged-vias.top.gerber - shows location where we want vias plugged with soldermask

drill files:
drill-through.DRL - plated through holes/vias
drill-01-04.DRL - blind vias (layers 1 to 4 inclusive)
drill-09-12.DRL - blind vias (layers 9 to 12 inclusive)
drill-unplated-pins.top.DRL - unplated blind holes (see note below)
drill-unplated-pins.bottom.DRL - unplated blind holes (see note below)




overall thickness = 69 mils after ENIG plating, measured metal to metal on both board edges

* controlled-depth drill on bottom side drilled to 0.8763 mm depth
* controlled-depth drill on top side drilled to 0.8763 mm depth
* controlled-depth drill on bottom side drilled to 0.8763 mm depth
* controlled-depth drill on top side drilled to 0.8763 mm depth

# drill drawing