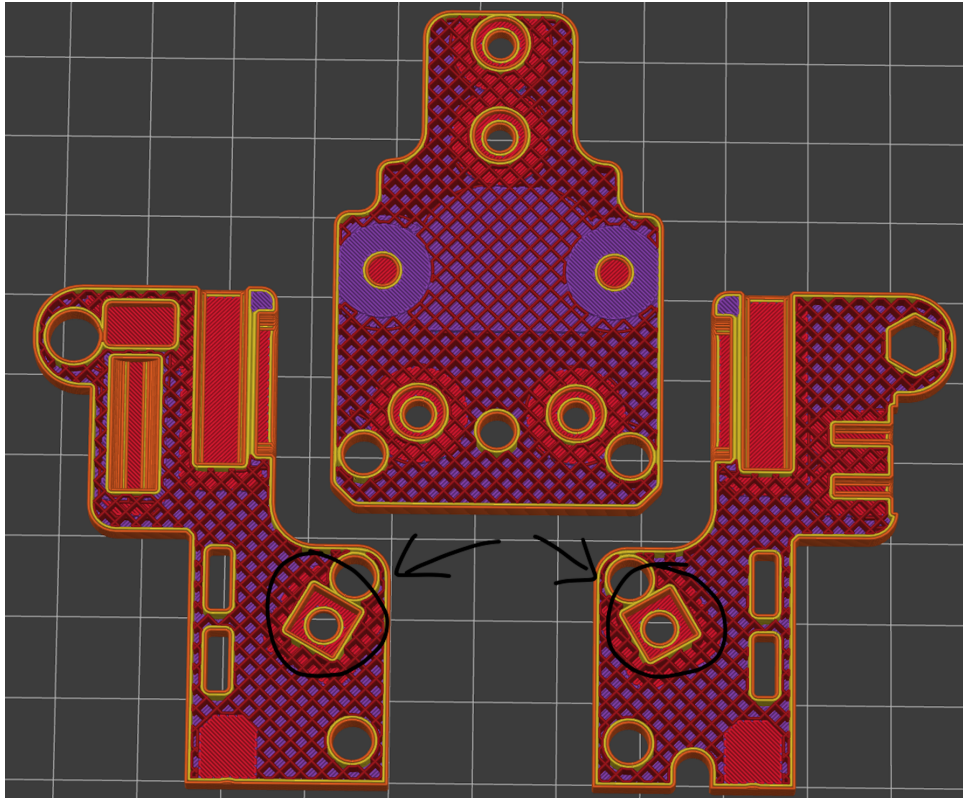


You need to pause the print at 3.6mm if you are printing with 0.2mm layer height!!!
And add 2x M3 square nuts.



Parts in the image are printed with PA12-CF15,
But you could print it with ABS or ASA. **PLA and PETG not supported or tested.**

PA12-CF15

Layer height: 0.2mm

Perimeters: 4

Solid layers Top: 5

Solid layers Bottom: 5

Infill: 40%

Infill pattern: Grid

Nozzle Temp.: 280C

Bed Temp.: 100C

Chamber Temp.: 48-50C

Outer wall: 40 mm/s

Inner wall 60 mm/s

Infill 80 mm/s

No support needed!!!

ABS or ASA

Layer height: 0.2mm

Perimeters: 4

Solid layers Top: 5

Solid layers Bottom: 5

Infill: 40%

Infill pattern: Grid

Nozzle Temp.: 255C

Bed Temp.: 90C

Chamber Temp.: 40-45C

Outer wall: 45 mm/s

Inner wall 120 mm/s

Infill 140 mm/s

No support needed!!!

As you are printing the parts without support. After the print is done, take Knipex Knips and cut out all the supports in red.

