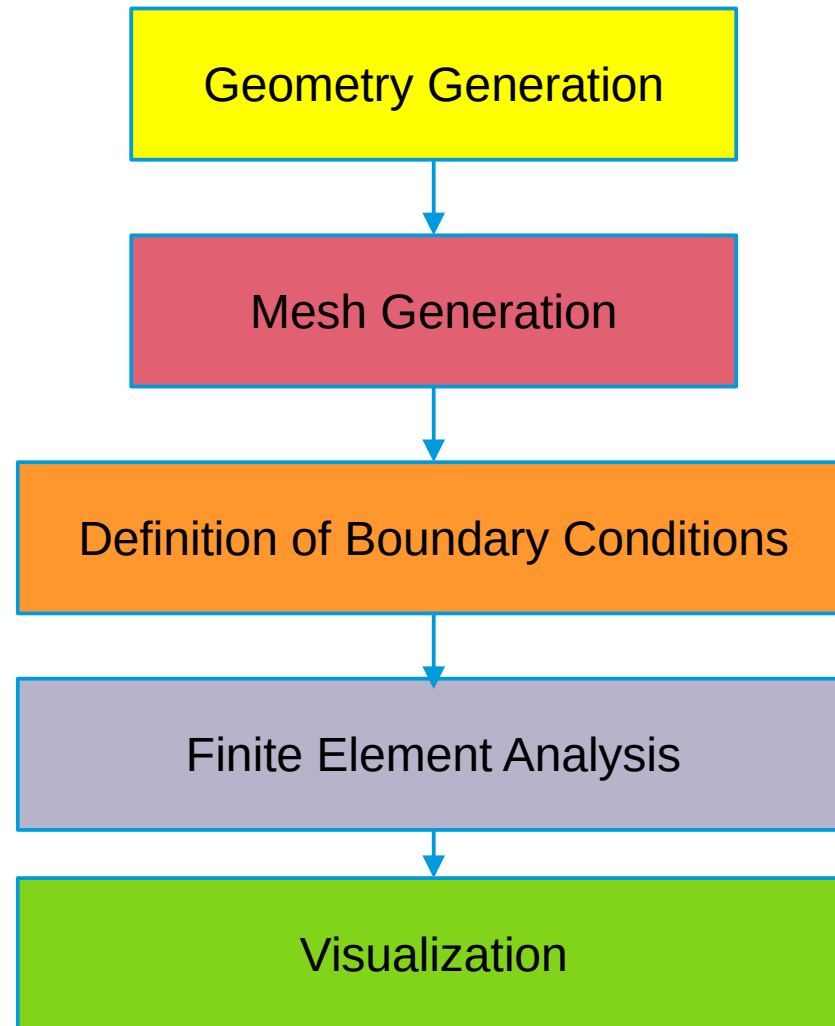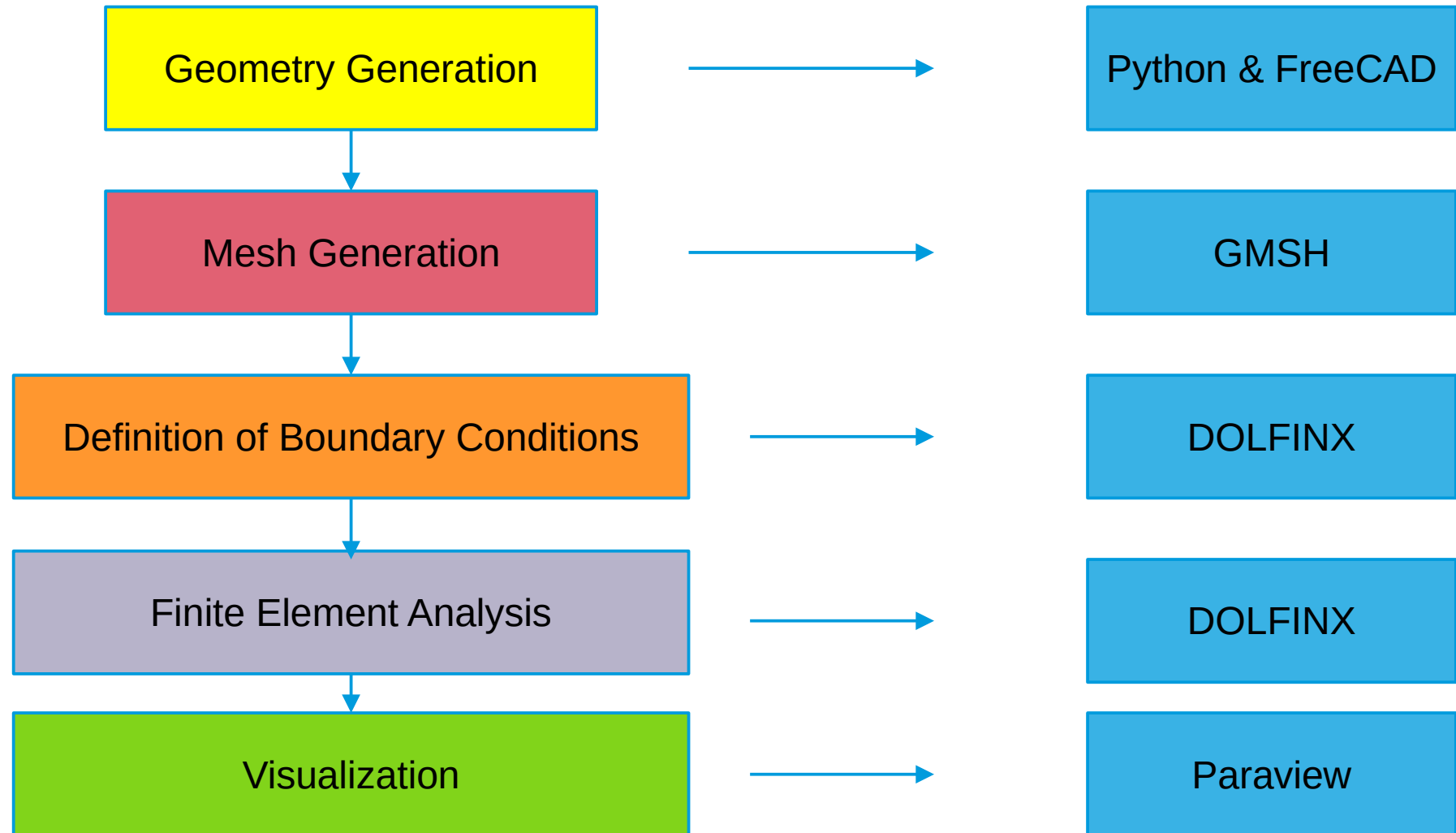# Finite Element Modelling of Gyroid-based Honeycomb

Aims and Objectives

- Complete Finite Element analysis of proposed gyroid geometry will be performed.

- Open-source tools are exploited.

- Finite Element framework is established for analysing displacement and stress fields.

- Open-source frameworks are indispensable tools to handle state of the art geometries for FE modelling.

- This statement wanted to be proven in this study.

Flowchart

Geometry Generation → Mesh Generation → Definition of Boundary Conditions → Finite Element Analysis → Visualization

Flowchart - Tools

|		
Geometry Generation	→	Python & FreeCAD
Mesh Generation	→	GMSH
Definition of Boundary Conditions	→	DOLFINX
Finite Element Analysis	→	DOLFINX
Visualization	→	Paraview

Geometry Generation

Here are the steps;

- Python script generates points.

- Reading these points from txt file and import them into FreeCAD.

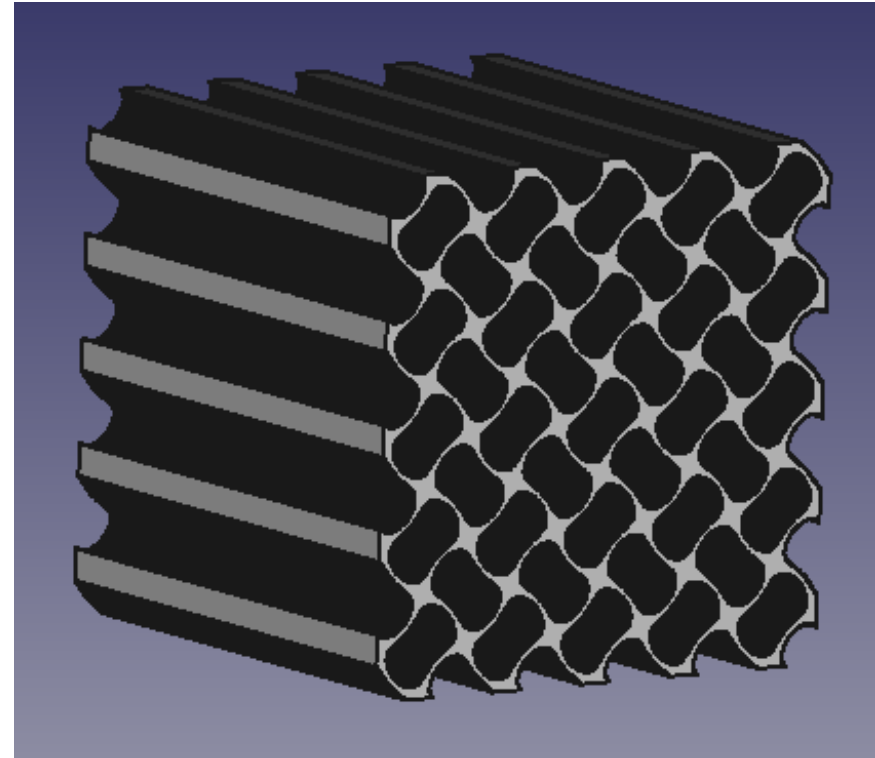- Manipulate geometry to get Gyroid-based honeycomb.

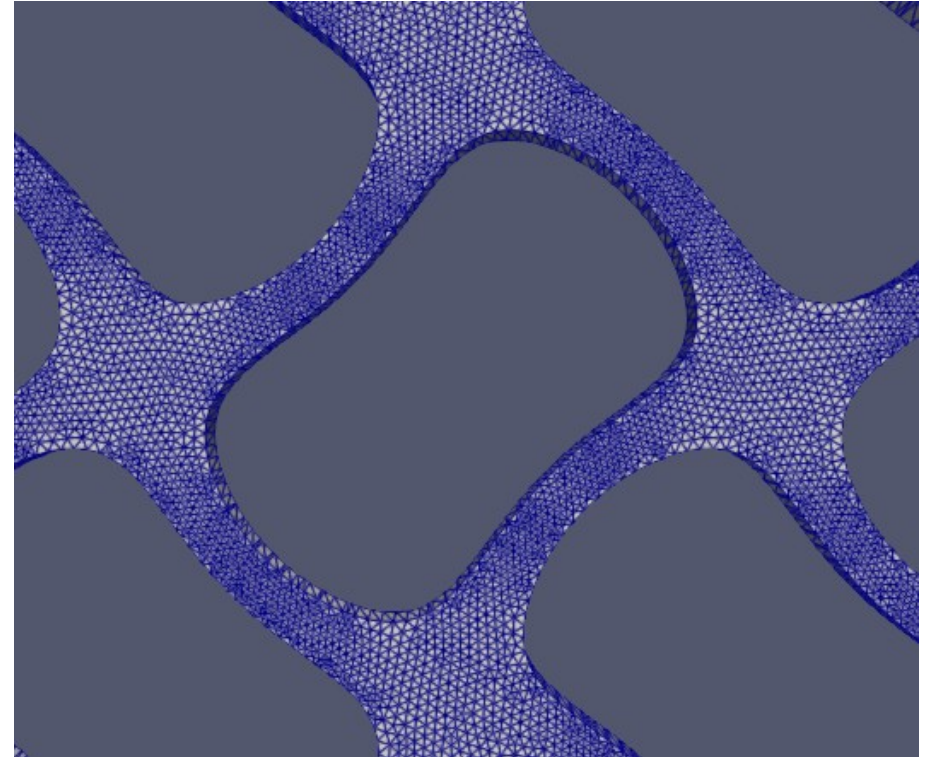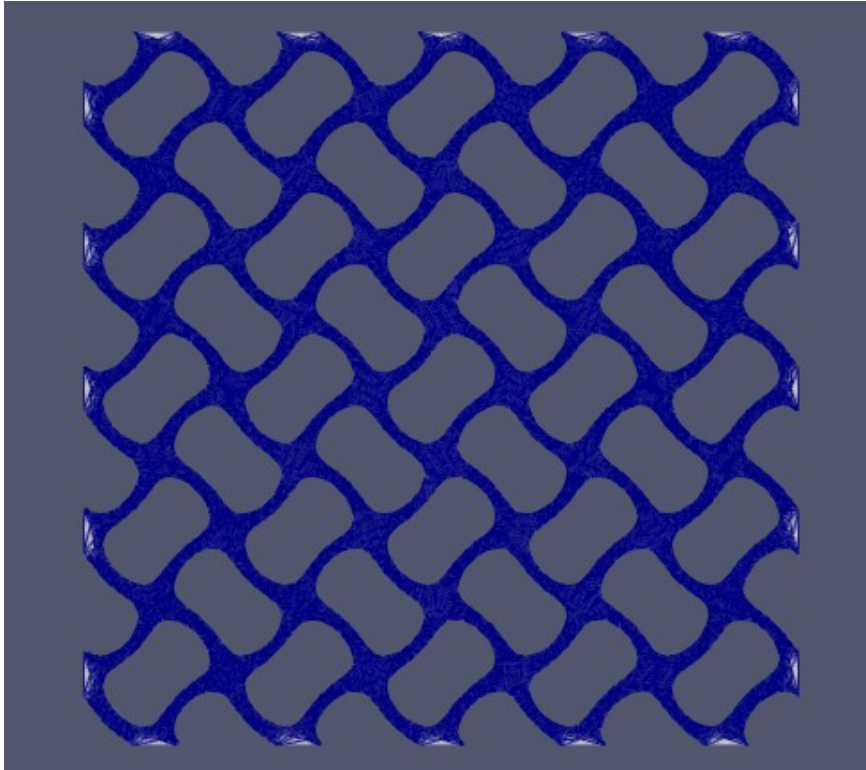The process is fully automated.

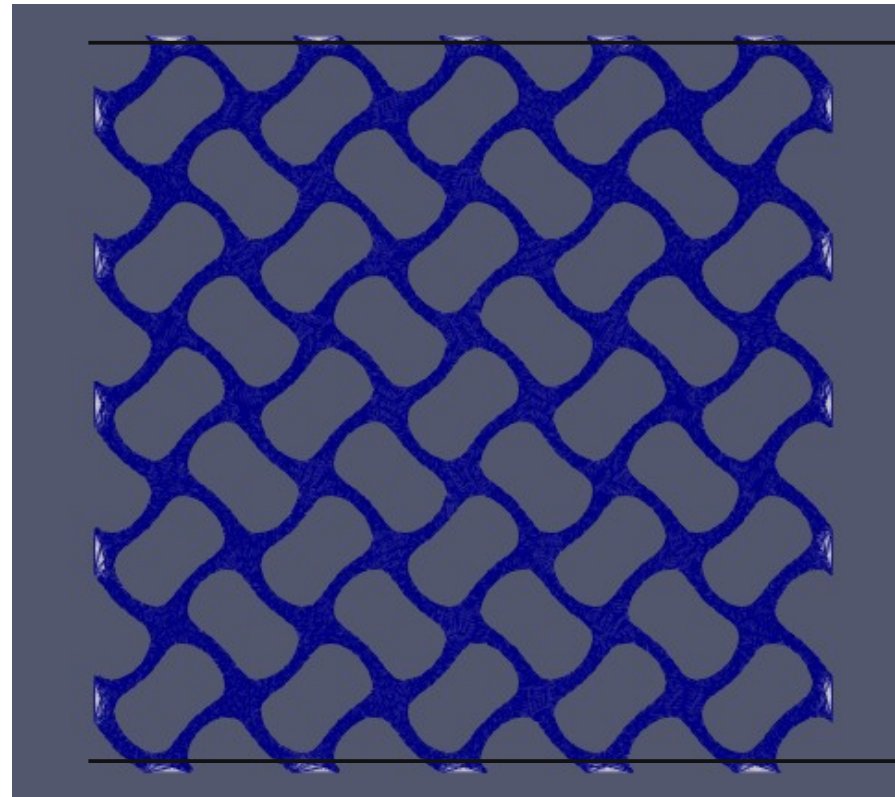# Geometry Generation

50mm
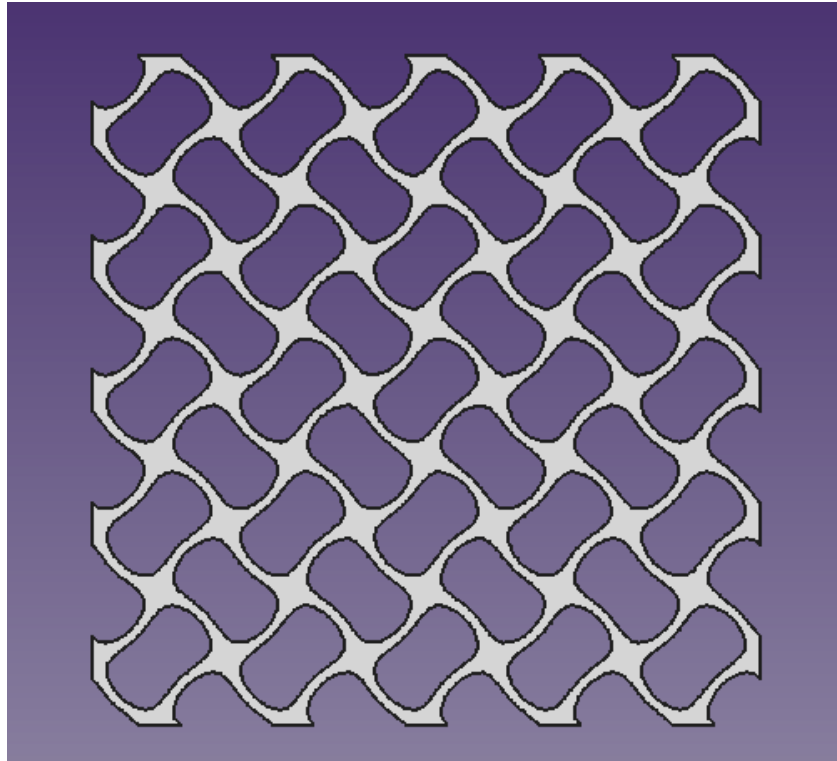
50mm

Mesh Generation

Around ~667k cells

Definition of Boundary Conditions

Rest of the boundaries;

Traction-Free

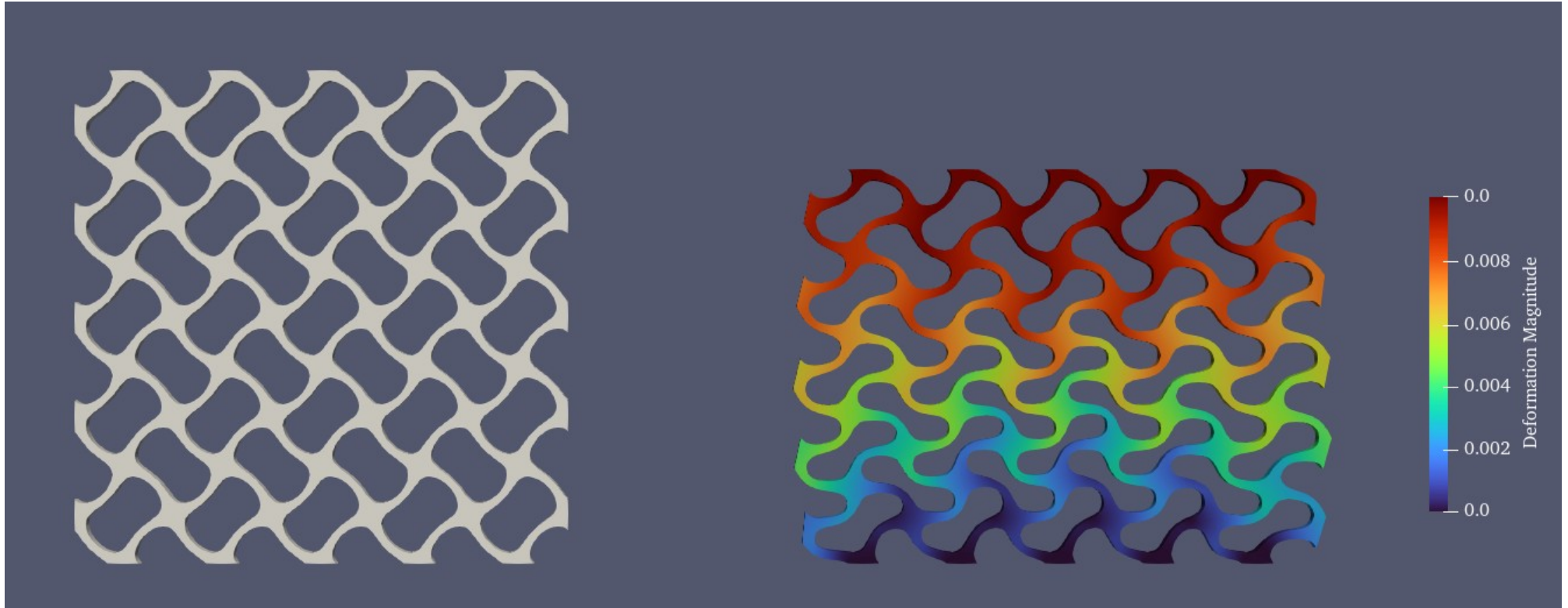Displacement boundary condition

Fixed Boundary

Finite Element Analysis

- Dolfinx Framework has been used.

  https://github.com/FEniCS/dolfinx

- Linear Elasticity equations are solved.

- Python interface has been utilized.

Material Properties

Table 4. The mechanical properties of the Polyamide PA 2200 [29].

|	
Tensile Modulus	1650 MPa
Tensile Strength	48 MPa
Elongation at Break	18%
Melting Temperature (20°C/min)	176°C
Heat Deflection Temperature (1.80 MPa)	70°C
Heat Deflection Temperature (0.65 MPa)	154°C
Density	930 kg/m³

Visualization – Displacement Field

We are imposing y=-0.01m displacement from top boundary
(Compression Test)

Visualization – Von Mises Stresses
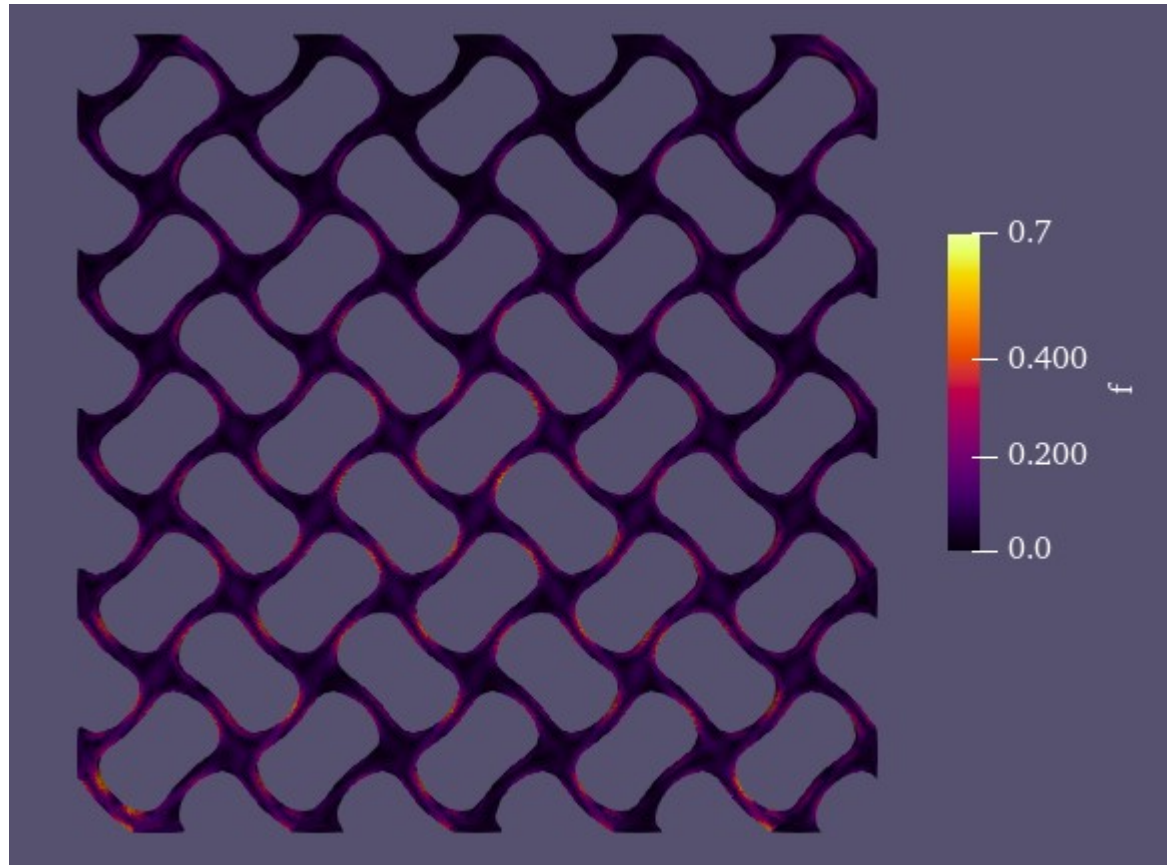
Issue - Overlapping
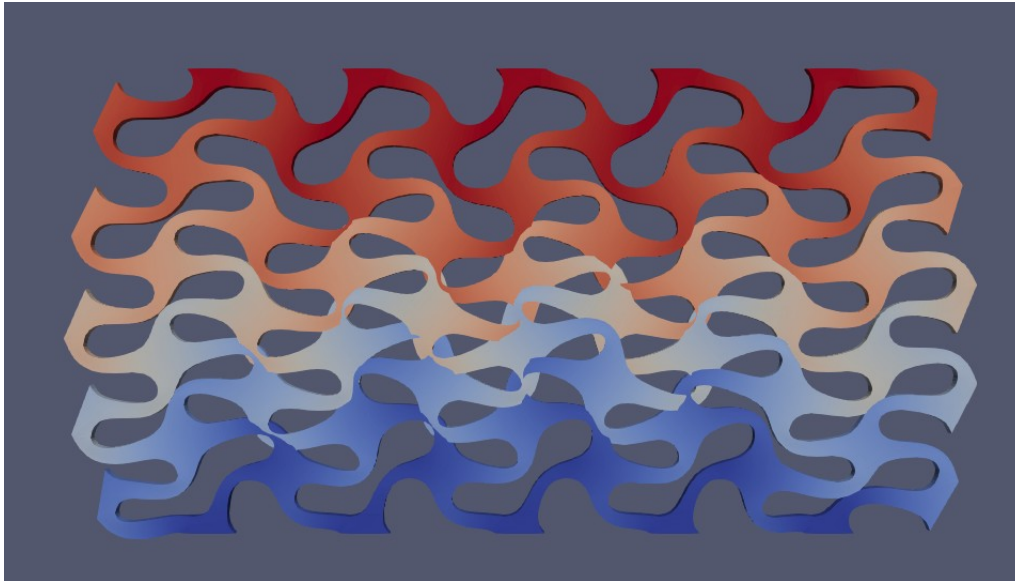
When displacement of the top boundary exceeds y = -0.02m, the mesh overlaps.

Contacts between internal boundaries should be defined to avoid overlapping.

Possible solutions;
1) This problem can be treated as postprocessing problem.
- As we now the displacements of the each node in the mesh, we can manipulate the coordinates accordingly.
- At the moment, this is not implemented in **Dolfinx**,
- But we can implement -> it will take some time for sure (it is already ongoing PostDoc project of friend of mine).

Conclusion

- Framework of Finite Element Analysis for Gyroid-based honeycomb geometry is established.

- Material parameters are not well-defined yet, but easily doable.

- More complex structures (graded, periodic) can be implemented using Python scripts.