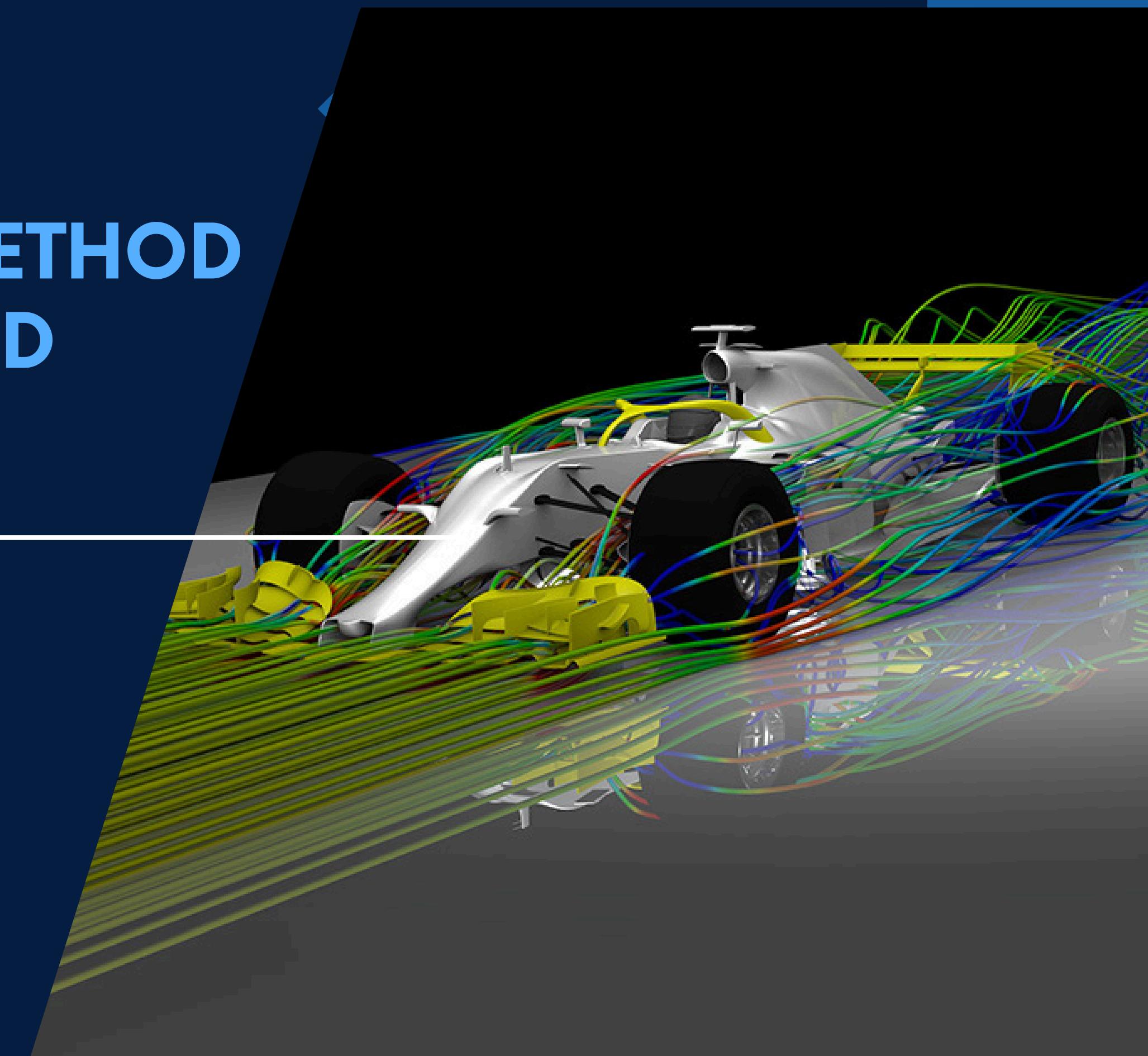


CFD

AN OVERVIEW OF VOF METHOD IN COMPUTATIONAL FLUID DYNAMICS

Presented by :

Dharmansh Vyas
Arnav Maitreya
Jahnvi Soni



MULTIPHASE FLOW

In fluid mechanics, multiphase flow is the simultaneous flow of materials with two or more thermodynamic phases.

- **Method to solve**

1. Volume of Fluid (VOF) Method

VOF is used to track the interface between immiscible fluids by solving a transport equation for the volume fraction of one of the fluids.



2. Eulerian-Eulerian Model

Treats each phase as an interpenetrating continuum, solving separate sets of conservation equations (mass, momentum, and energy) for each phase.

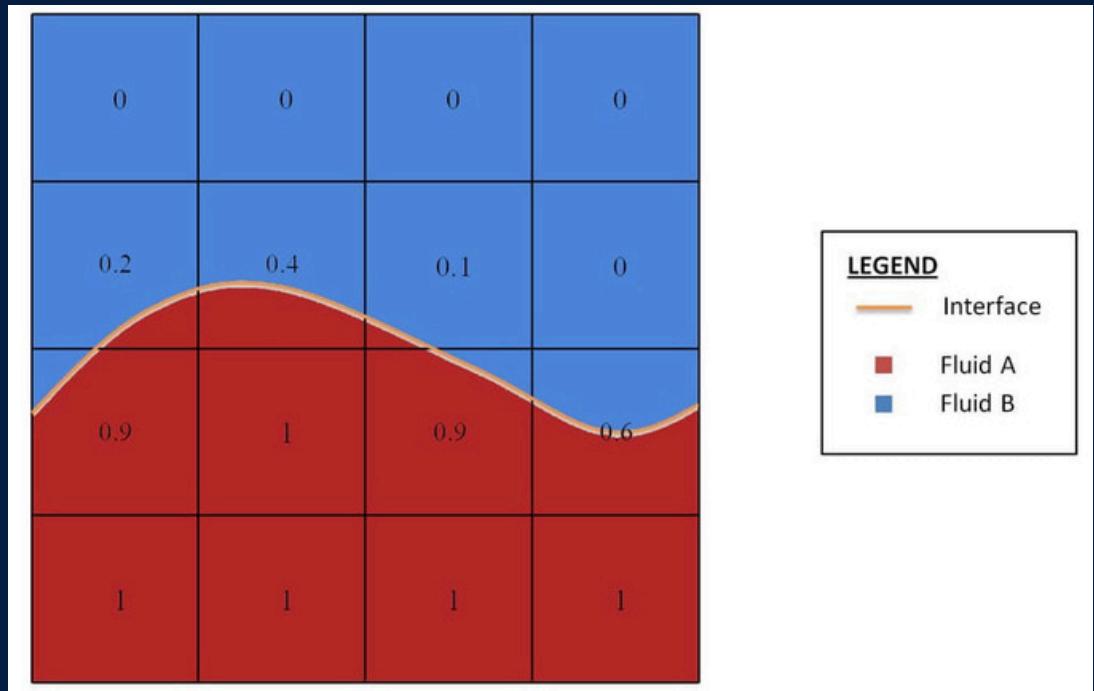
3. Eulerian-Lagrangian Model

Models the continuous phase using Eulerian framework and the dispersed phase using Lagrangian framework, tracking individual particles or droplets.

There are other ways too to solve multiphase flow

- **Definition:** The Volume of Flow Method (VOF) is a numerical technique used in CFD to track and locate the free surface (or interface) between two immiscible fluids.
- **Principles:** VOF is based on the concept of a fractional volume of fluid (vof) within each computational cell.
- **Difference:** Unlike other methods like Finite Volume Method (FVM) or Finite Element Method (FEM), VOF focuses specifically on capturing the dynamics of the interface between fluids.
- The VOF method utilizes the transport equation to handle the distribution and movement of the fluid phases. Here's a detailed explanation of how the transport equation is related to the VOF method:
- The VOF method uses a fractional volume function, F, to represent the volume fraction of one of the fluids within each computational cell. The value of F ranges from 0 to 1, where:

- $F=0$: The cell is entirely filled with the second fluid.
- $F=1$: The cell is entirely filled with the first fluid.
- $0 < F < 1$: The cell contains the interface between the two fluids.



1. Continuity Equation

For incompressible flows, the continuity equation ensures mass conservation:

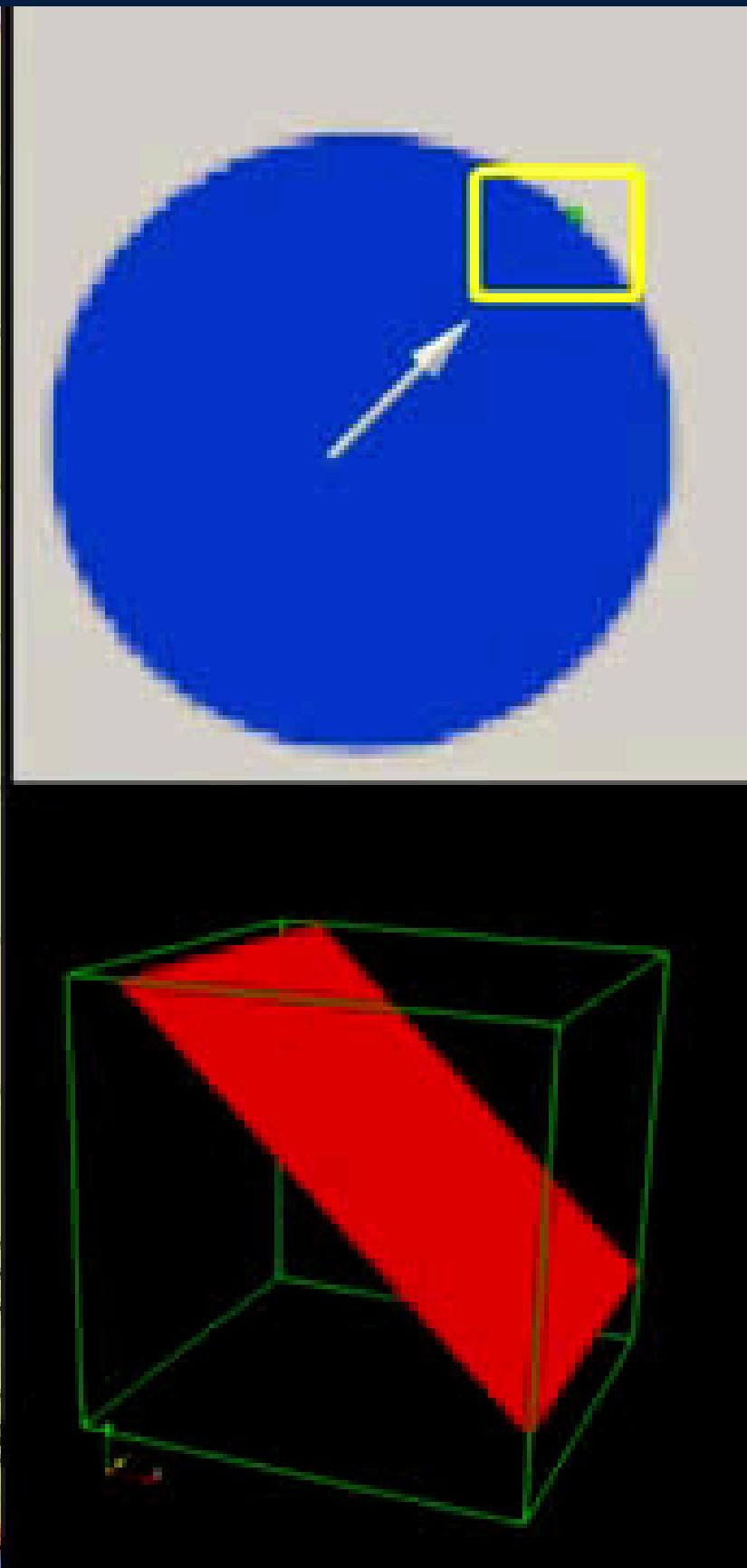
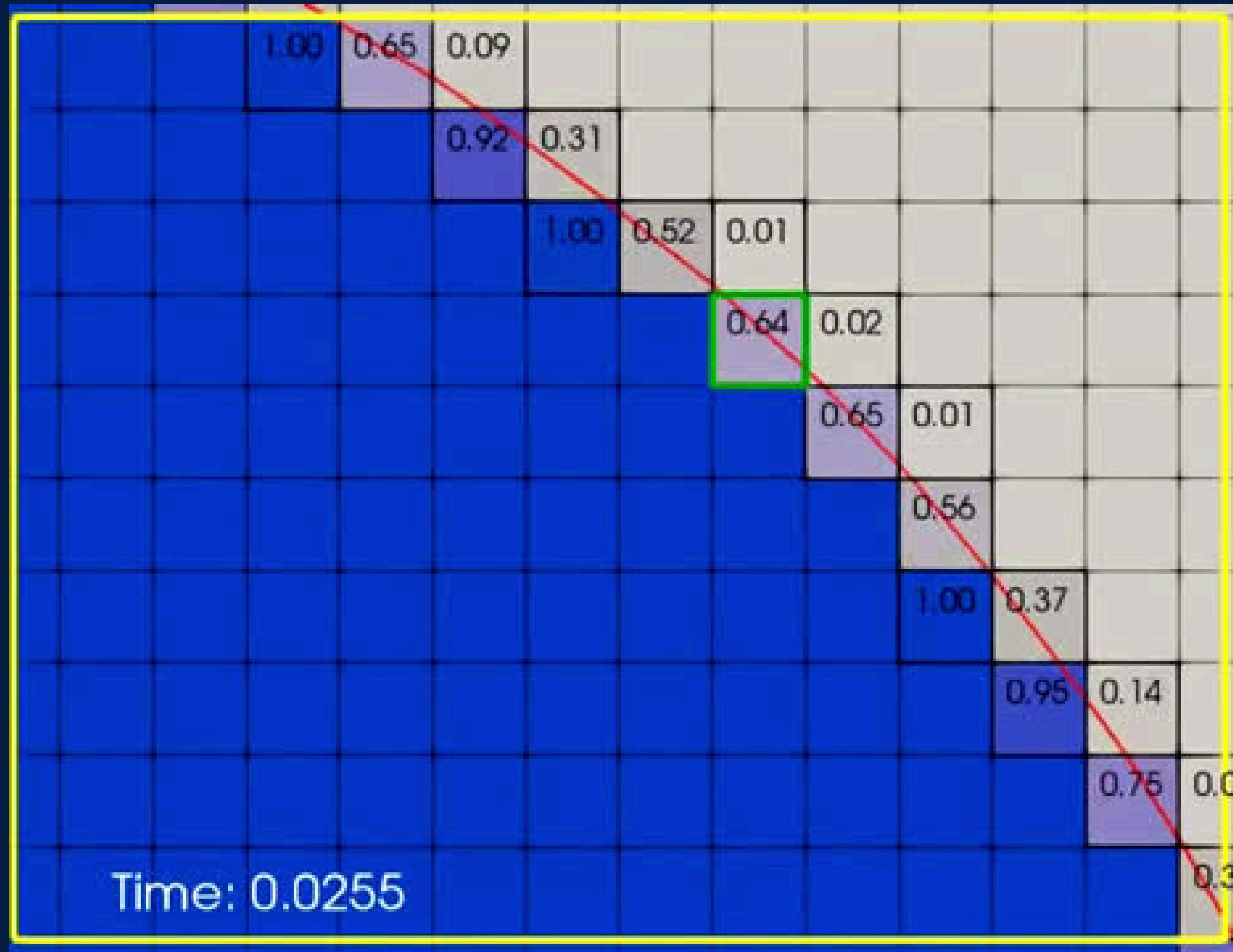
$$\nabla \cdot \mathbf{u} = 0$$

where \mathbf{u} is the velocity vector.

2. Navier-Stokes Equations

The Navier-Stokes equations describe the momentum conservation and are given by:

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot (\mu \nabla \mathbf{u}) + \mathbf{F}$$



Transport Equation

$$\frac{\partial \phi}{\partial t} = \underbrace{\nabla \cdot u \phi}_{\text{unsteady term}} - \underbrace{\nabla \cdot \Gamma \nabla \phi}_{\text{Convection term}} + \underbrace{S}_{\text{Diffusion term}} + \underbrace{S}_{\text{Source term}}$$

Convective term gives us idea that how the scalar is changing by the flow of the fluid.

Diffusion term tell us ,how the scalar diffuses in the fluid.

The source term, represents the amount of the scalar quantity created or destroyed inside the control volume.

Unsteady term represents the change of scalar w.r.t time.

Transport Equation in VOF Method

$$\frac{\partial F}{\partial t} + \nabla \cdot (\mathbf{u}F) = 0$$

This equation describes the advection of the volume fraction F with the fluid flow. Here's a breakdown of the components:

1. Local Rate of Change ($\frac{\partial F}{\partial t}$):

- This term represents the change in the volume fraction F over time at a specific point in space.

2. Convective Transport ($\nabla \cdot (\mathbf{u}F)$):

- This term represents the transport of the volume fraction F due to the fluid motion. The velocity field \mathbf{u} advects the fluid interface through the computational domain.

The physical properties (e.g., density ρ and viscosity μ) of the fluid mixture are computed as weighted averages based on the volume fraction F :

- Density:

$$\rho = F\rho_1 + (1 - F)\rho_2$$

- Where ρ_1 and ρ_2 are the densities of the two fluids.

- Viscosity:

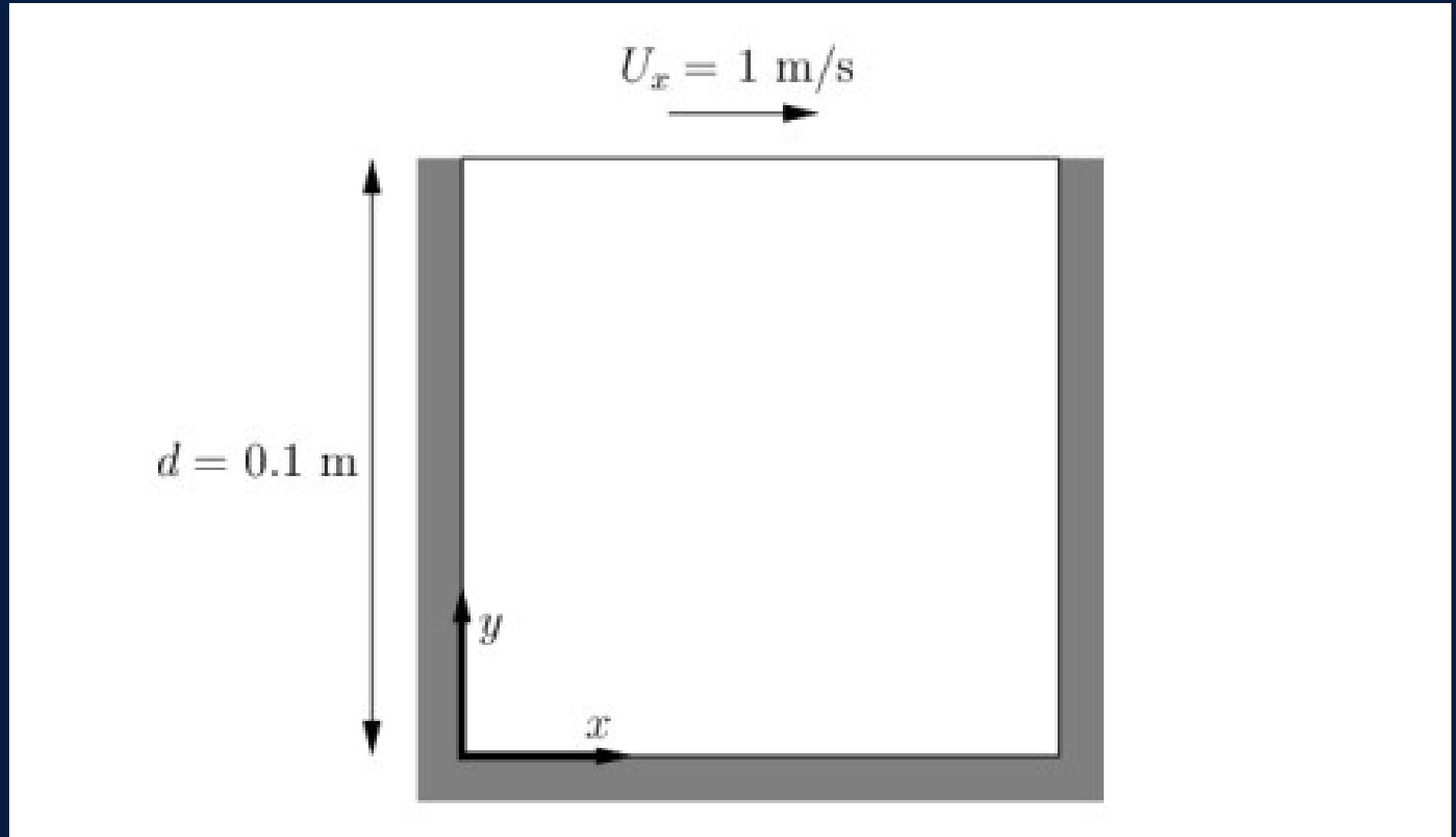
$$\mu = F\mu_1 + (1 - F)\mu_2$$

- Where μ_1 and μ_2 are the viscosities of the two fluids.

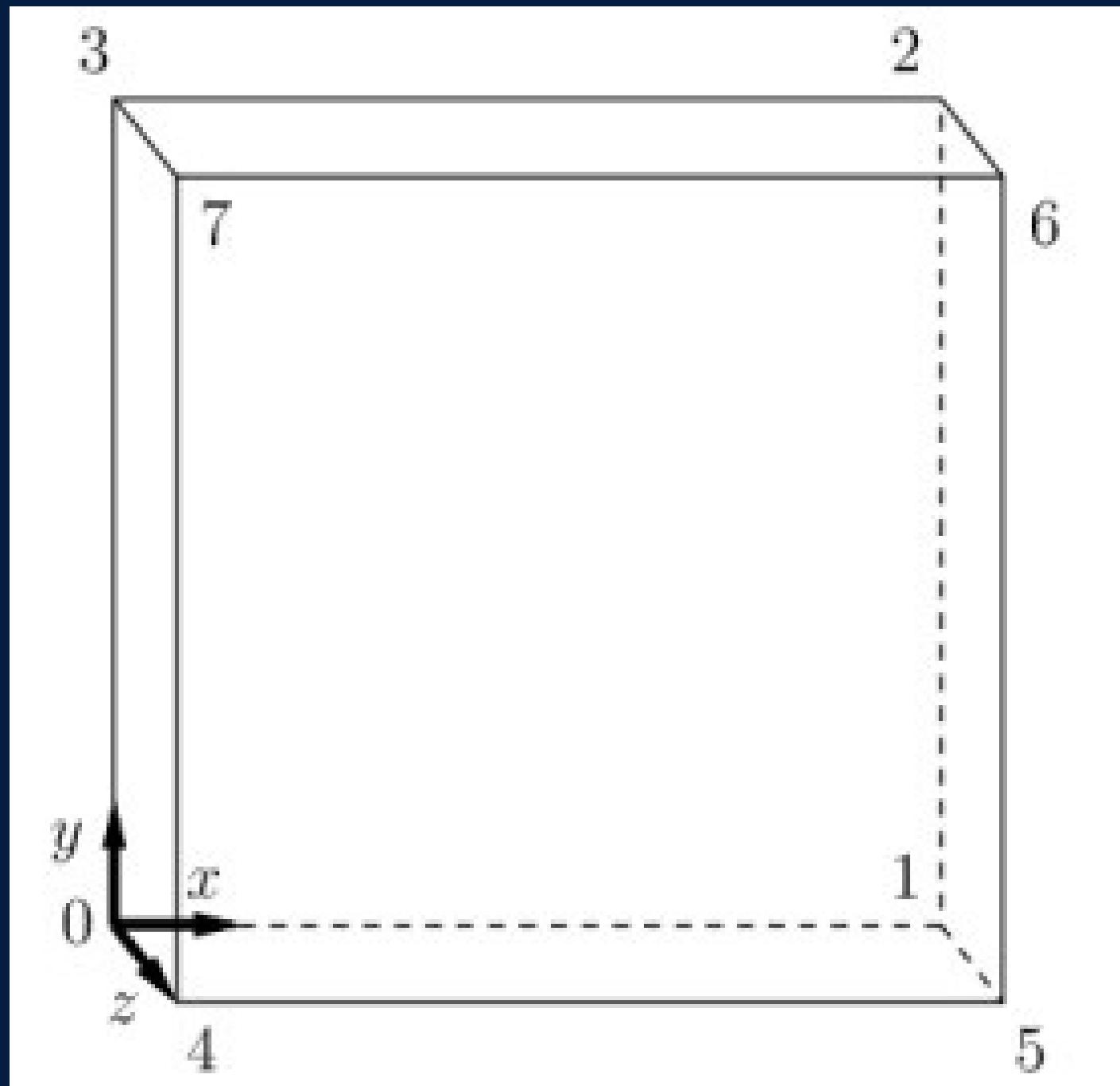
Laminar Flow

- Problem -1 (Lid-Driven Cavity)

the flow will be assumed laminar and will be solved on a uniform mesh using the `icoFoam` solver for laminar, isothermal, incompressible flow.



• Computational Domain

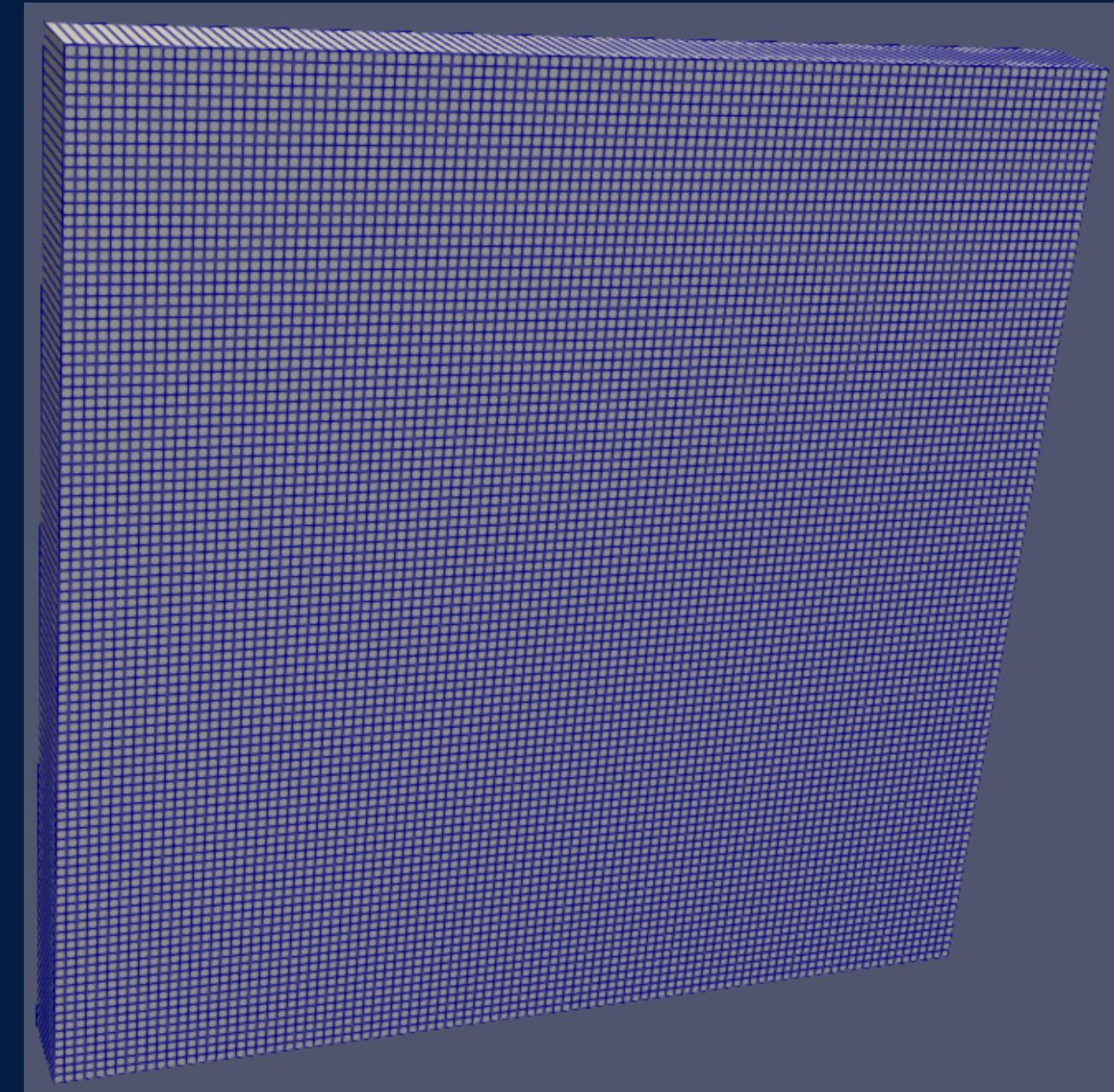


- movingWall -(wall)
(3 7 6 2)
- fixedWalls -(Wall)
(0 4 7 3)
(2 6 5 1)
(1 5 4 0)
- frontAndBack-(Empty)
(0 3 2 1)
(4 5 6 7)

- Type of mesh

The computational mesh is a structured, uniform, rectangular grid.

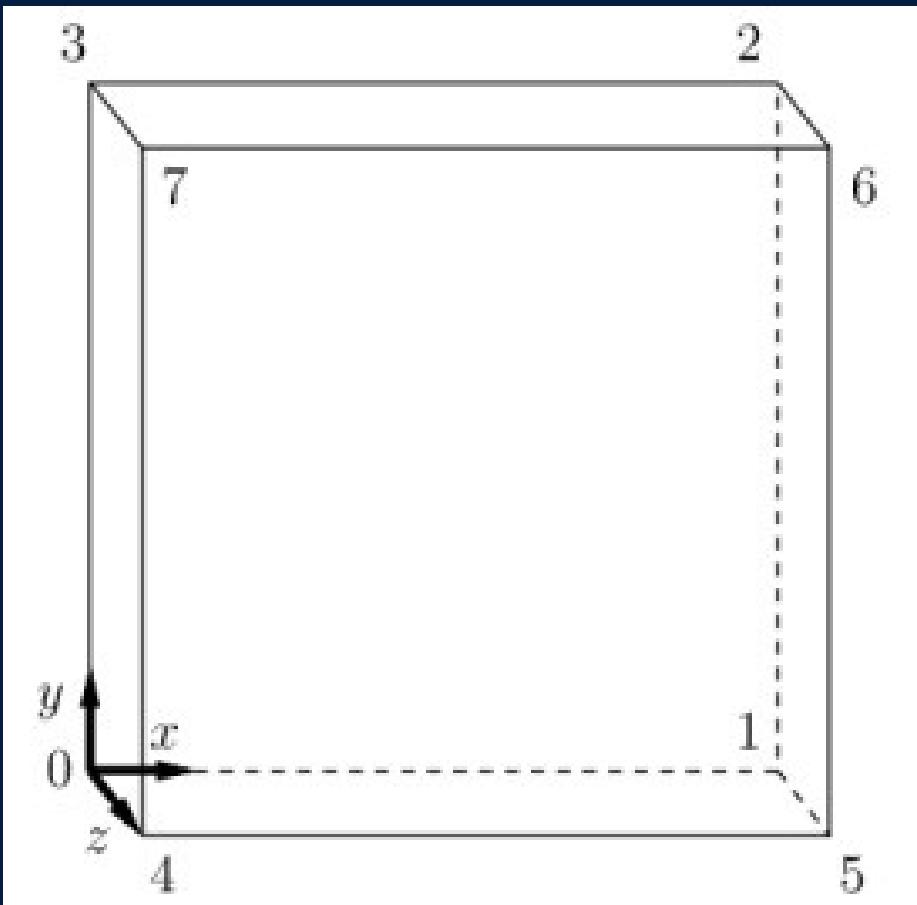
The grid is discretized into a finite number of cells. The domain is divided into 100 cells along the x-axis and 100 cells along the y-axis with only 1 cell in z- direction.



100 x 100

```
blocks
(
    hex (0 1 2 3 4 5 6 7) (100 100 1) simpleGrading (1 1 1)
);
```

• Boundary and initial Conditions



- movingWall -(wall)
(3 7 6 2)
- fixedWalls -(Wall)
(0 4 7 3)
(2 6 5 1)
(1 5 4 0)
- frontAndBack-(Empty)
(0 3 2 1)
(4 5 6 7)

Pressure

- movingWall -(wall)
Zero Gradient
- fixedWalls -(Wall)
Zero Gradient
- frontAndBack-(Empty)
empty

INFERENCE- there is no pressure difference along thier length

Velocity

- movingWall -(wall)
fixedValue (1 0 0)

INFERENCE- the velocity field have value of 1 in x direction

- fixedWalls -(Wall)
no Slip

INFERENCE- The velocity of the fluid in contact with the wall is zero.

- frontAndBack-(Empty)
empty

internalField uniform (0 0 0);

INFERENCE- Initially, the velocity inside the domain is zero.

● Result

1. Velocity Field:

- Formation of a primary vortex due to moving lid and secondary vortices near stationary corners.

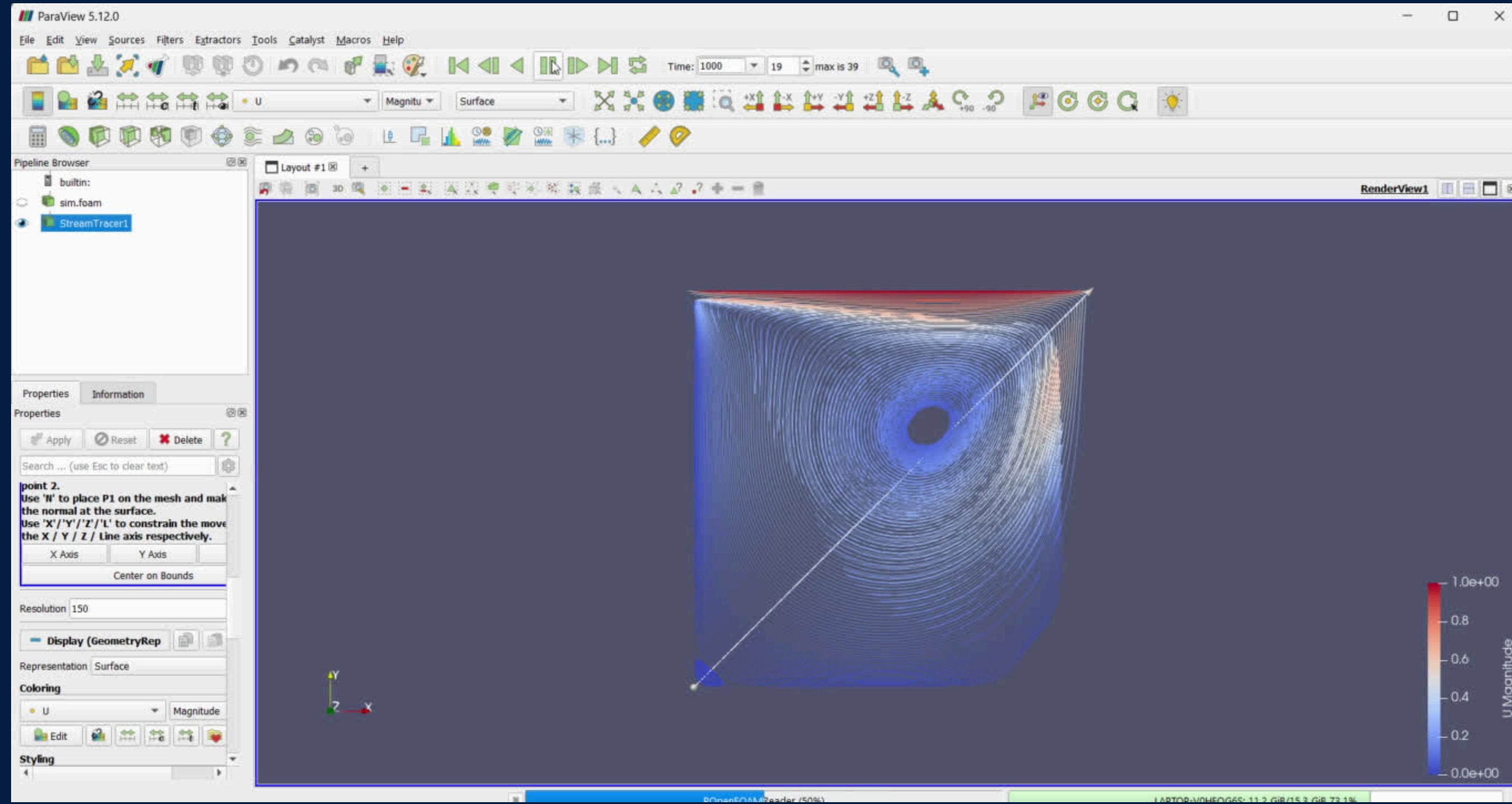
2. Pressure Field :

- Higher pressure at the lower right corner and lower pressure at the upper left corner.

3. Streamlines :

- Visualization of flow paths within the cavity . Clear indication of vortex structure.

● Result



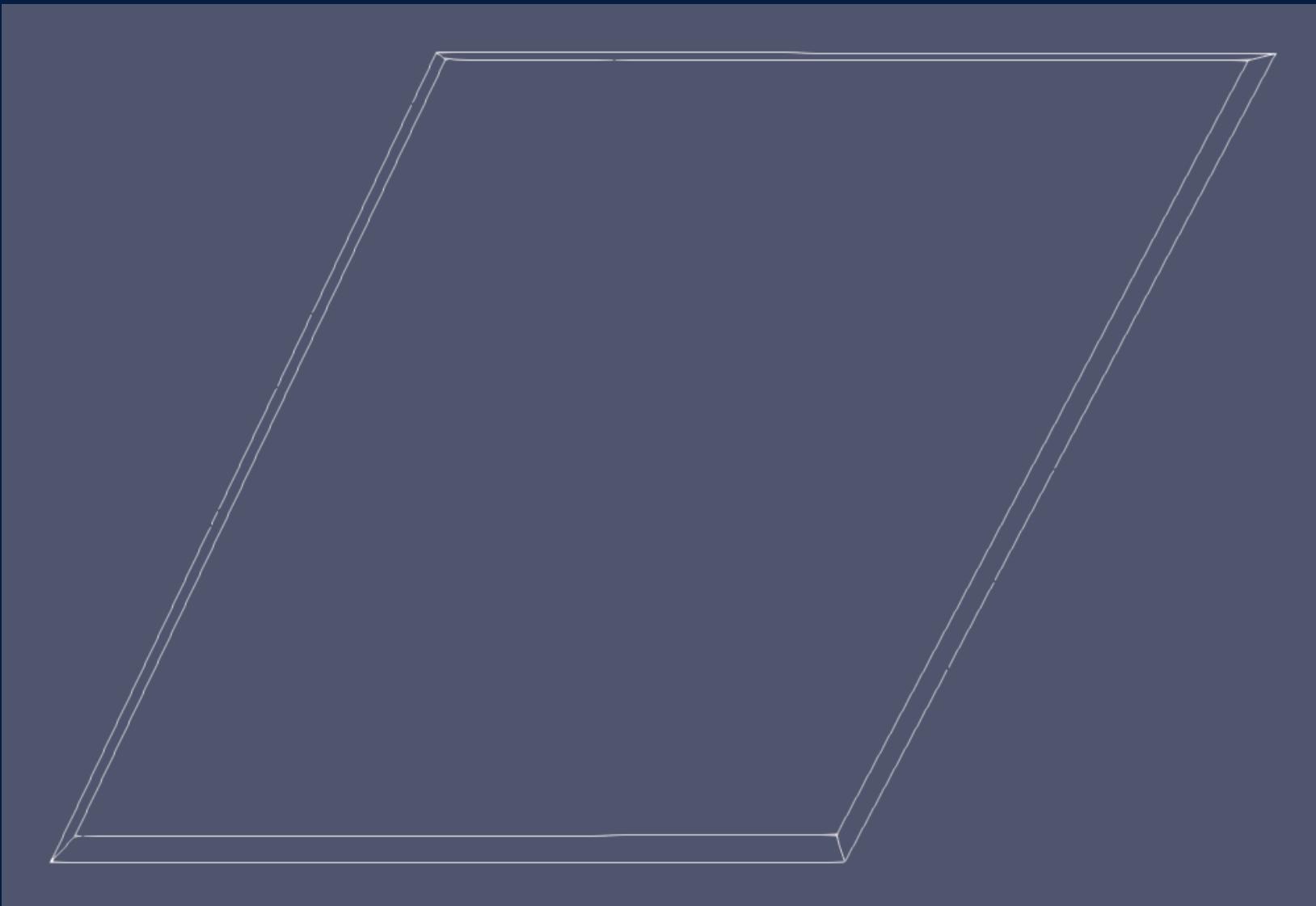
Laminar Flow

- Problem -2 (Skewed Lid-Driven Skew Cavity)

the flow will be assumed laminar and will be solved on a uniform mesh using the `icoFoam` solver for laminar, isothermal, incompressible flow.



• Computational Domain

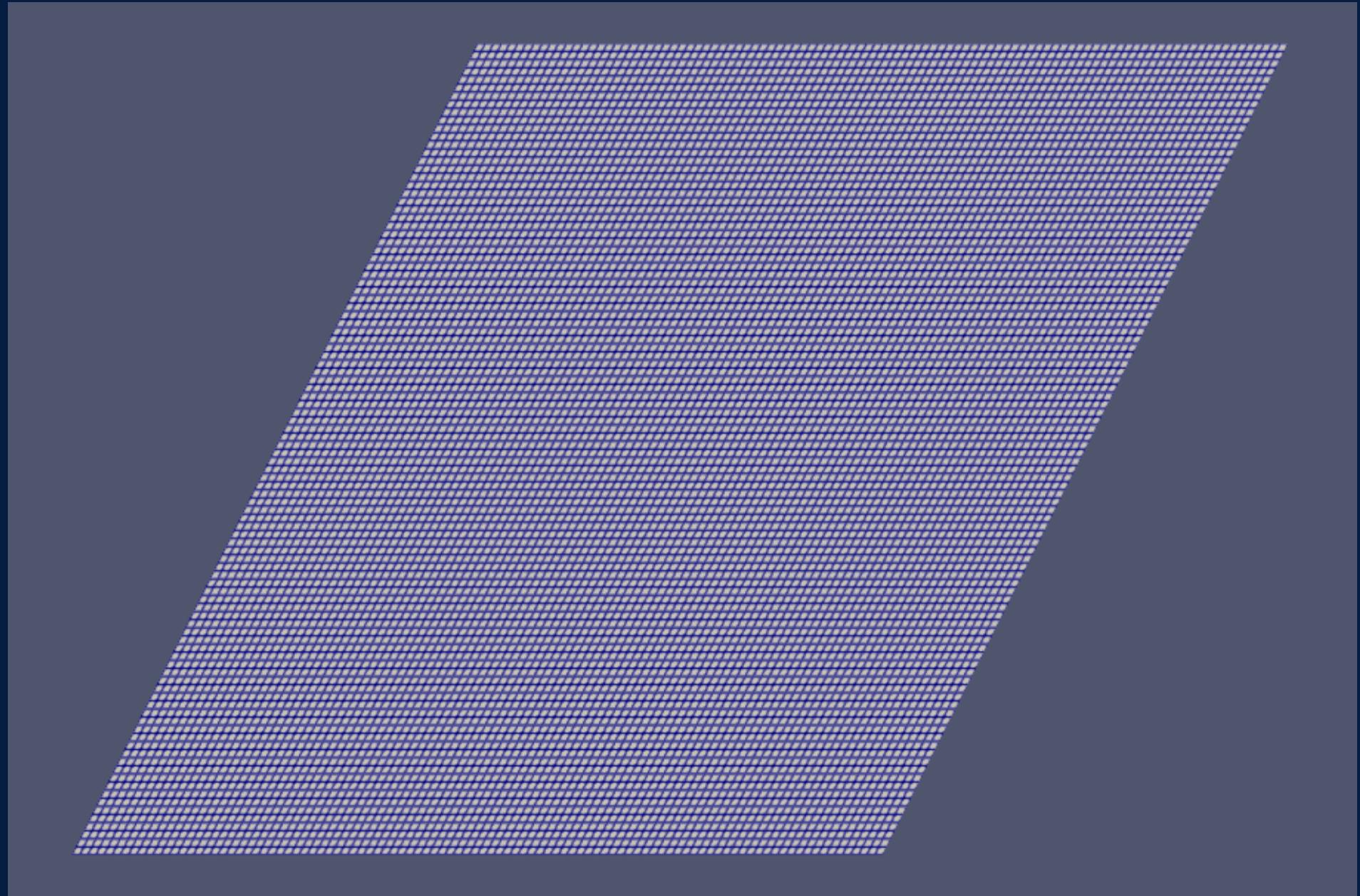


- movingWall -(wall)
(3 7 6 2)
- fixedWalls -(Wall)
(0 4 7 3)
(2 6 5 1)
(1 5 4 0)
- frontAndBack-(Empty)
(0 3 2 1)
(4 5 6 7)

- **Type of mesh**

The computational mesh is a structured, uniform, skewed grid at 60 degrees.

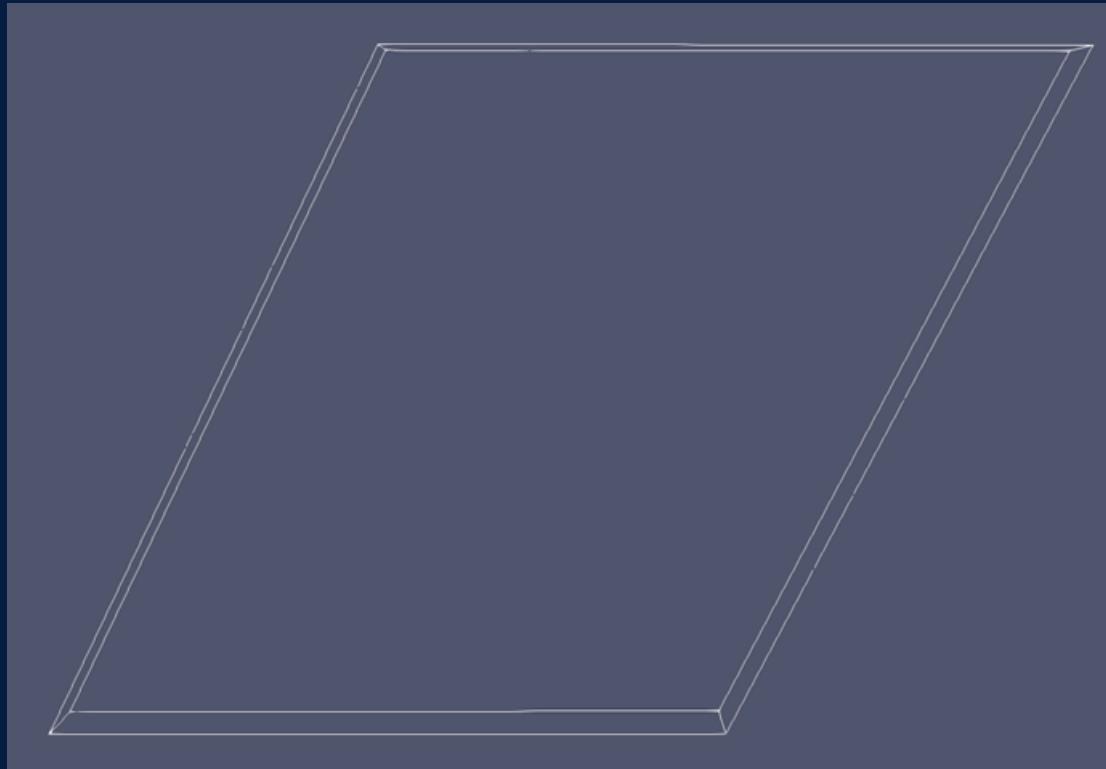
The grid is discretized into a finite number of cells. The domain is divided into 100 cells along the x-axis and 100 cells along the y-axis with only 1 cell in z- direction.



100 x 100

```
blocks
(
    hex (0 1 2 3 4 5 6 7) (100 100 1) simpleGrading (1 1 1)
);
```

- Boundary and initial Conditions



- movingWall -(wall)
(3 7 6 2)
- fixedWalls -(Wall)
(0 4 7 3)
(2 6 5 1)
(1 5 4 0)
- frontAndBack-(Empty)
(0 3 2 1)
(4 5 6 7)

Pressure

- movingWall -(wall)
Zero Gradient
- fixedWalls -(Wall)
Zero Gradient
- frontAndBack-(Empty)
empty

INFERENCE- there is no pressure difference along thier length

Velocity

- movingWall -(wall)
fixedValue (1 0 0)

INFERENCE- the velocity field have value of 1 in x direction

- fixedWalls -(Wall)
no Slip

INFERENCE- The velocity of the fluid in contact with the wall is zero.

- frontAndBack-(Empty)
empty

internalField uniform (0 0 0);

INFERENCE- Initially, the velocity inside the domain is zero.

● Result

1. Velocity Field:

- Skewed primary vortex, possible secondary vortices near corners, and altered velocity vectors and contours.

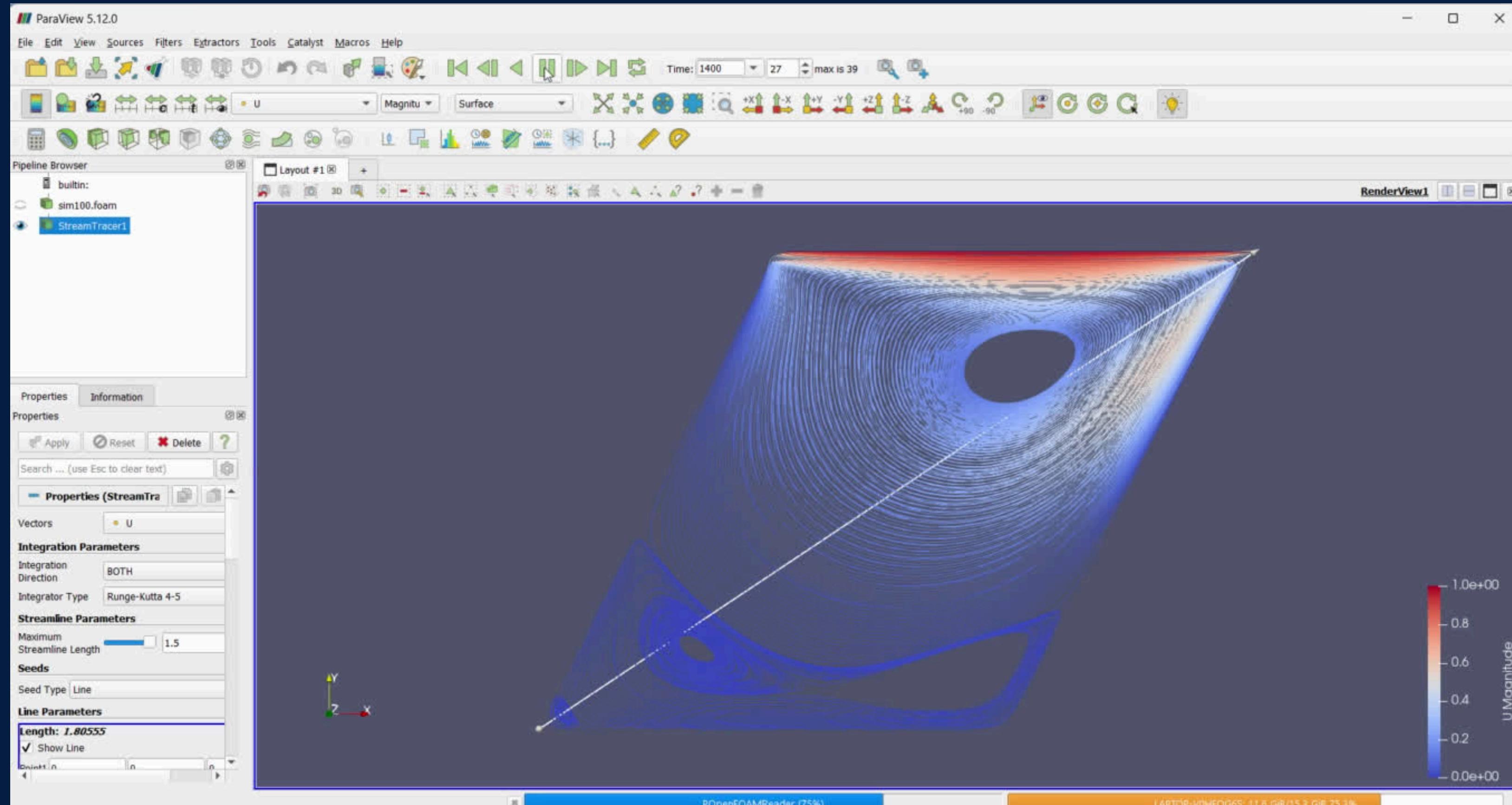
2. Pressure Field :

- Adjusted high-pressure regions and shifted zones influenced by 60-degree skewness.

3. Streamlines :

- Skewed flow paths reflecting the inclination. Clear depiction of skewed vortex structures.

● Result



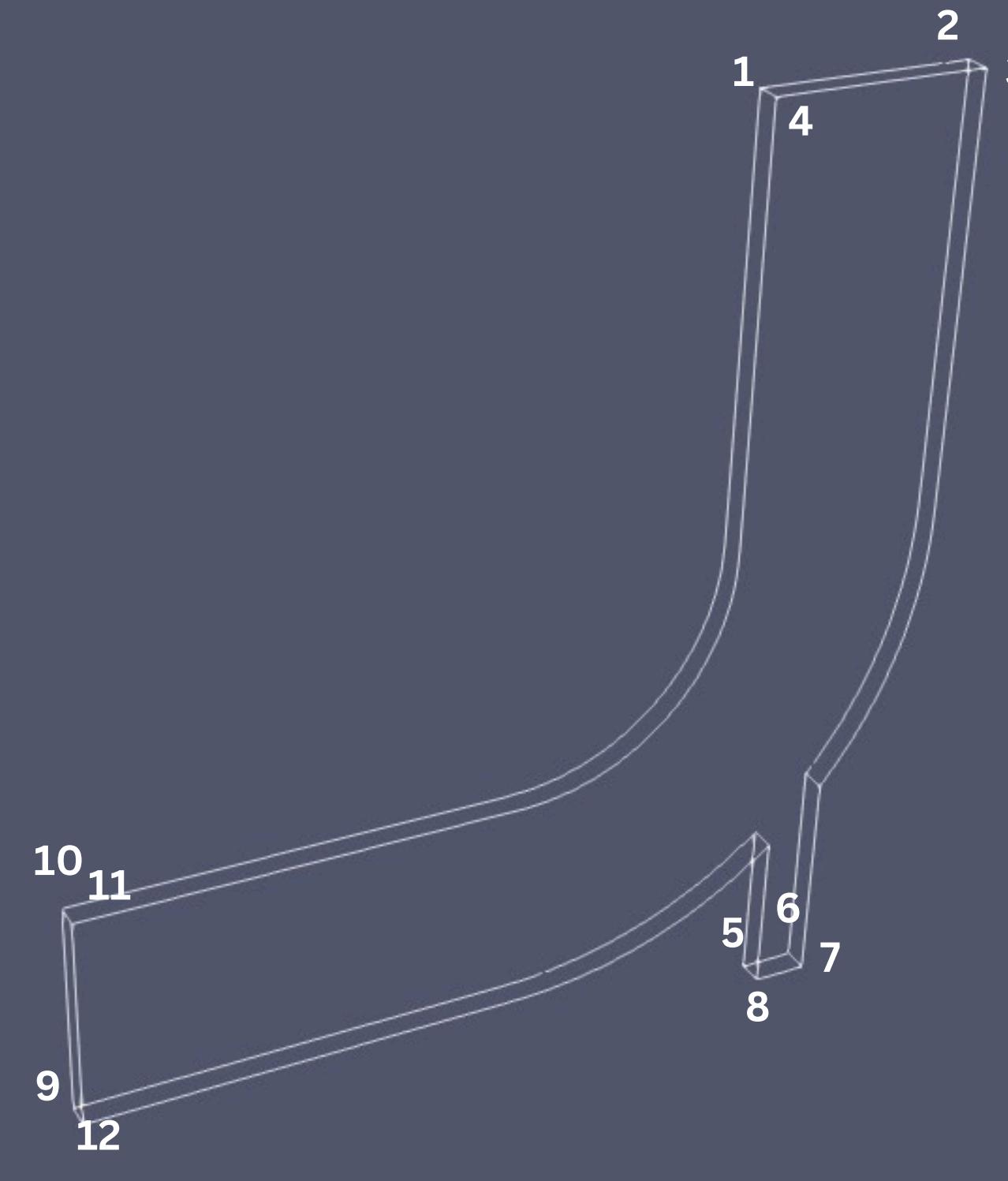
Laminar Flow

- **Problem -3 (elbow)**

The elbow tutorial in OpenFOAM provides a comprehensive example of setting up, running, and post-processing a steady-state incompressible turbulent flow simulation in a curved pipe.



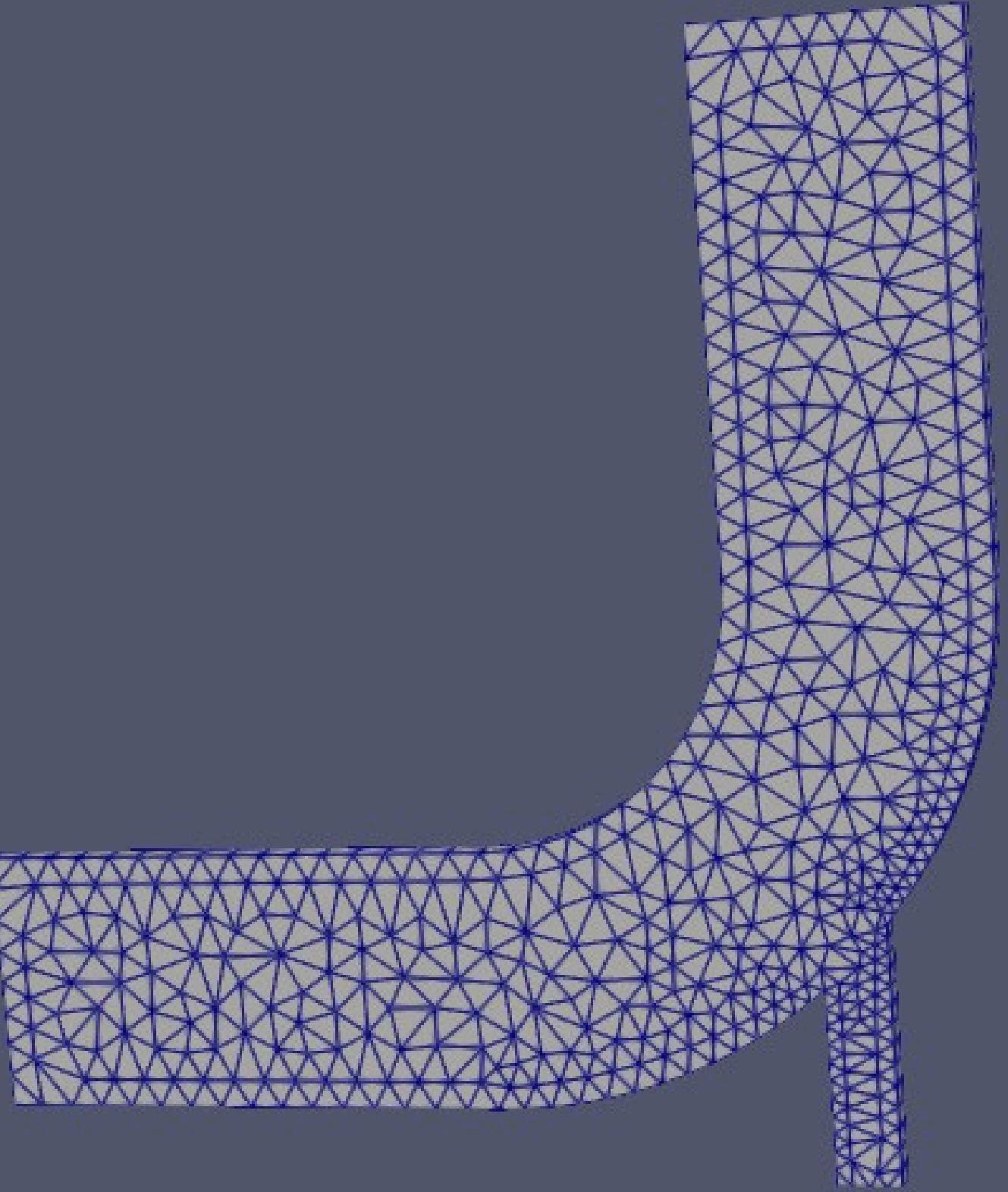
• Computational Domain



- Inlet 1
(9 10 11 12)
- Inlet 2
(5 6 7 8)
- Outlet
(1 2 3 4)
- Fixed wall Upper
(10 1 4 11)
- Fixed wall Lower
(9 2 3 12)

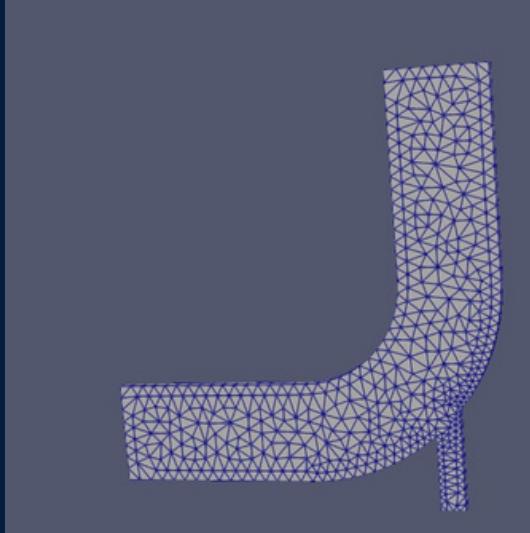
- Type of mesh

1. The computational mesh is an unstructured, non-uniform, triangular mesh.
2. With inlets having nFaces 8 and 4, outlet with 8 and walls having 34 and 100.
3. Mesh is generated using Fluent with .msh extension and then converted in openFoam format with “fluentToFoam” command.



Boundary and initial Conditions

Pressure



- Inlet 1
(9 10 11 12)

- Inlet 2
(5 6 7 8)

- Outlet
(12 3 4)

- Fixed wall Upper
(10 1 4 11)

- Fixed wall Lower
(9 2 3 12)

Velocity

- Inlet 1
fixed Value - uniform(1 0 0)

INFERENCE- the velocity filed at inlet 2 with 1m/s in x direction

- Inlet 2
fixed Value - uniform(0 3 0)

INFERENCE- the velocity filed at inlet 1 with 3m/s in y direction

- Fixed wall Upper
no slip

- Fixed wall Lower
no slip

- frontAndBack-(Empty)
empty

- outlet
Zero Gradient

INFERENCE- there is no pressure difference along thier length

- frontAndBack-(Empty)
empty

- outlet
Fixed Value

INFERENCE- the pressure at the outlet of the computational domain is set to a constant value, which is crucial for driving the flow through the elbow.

● Result

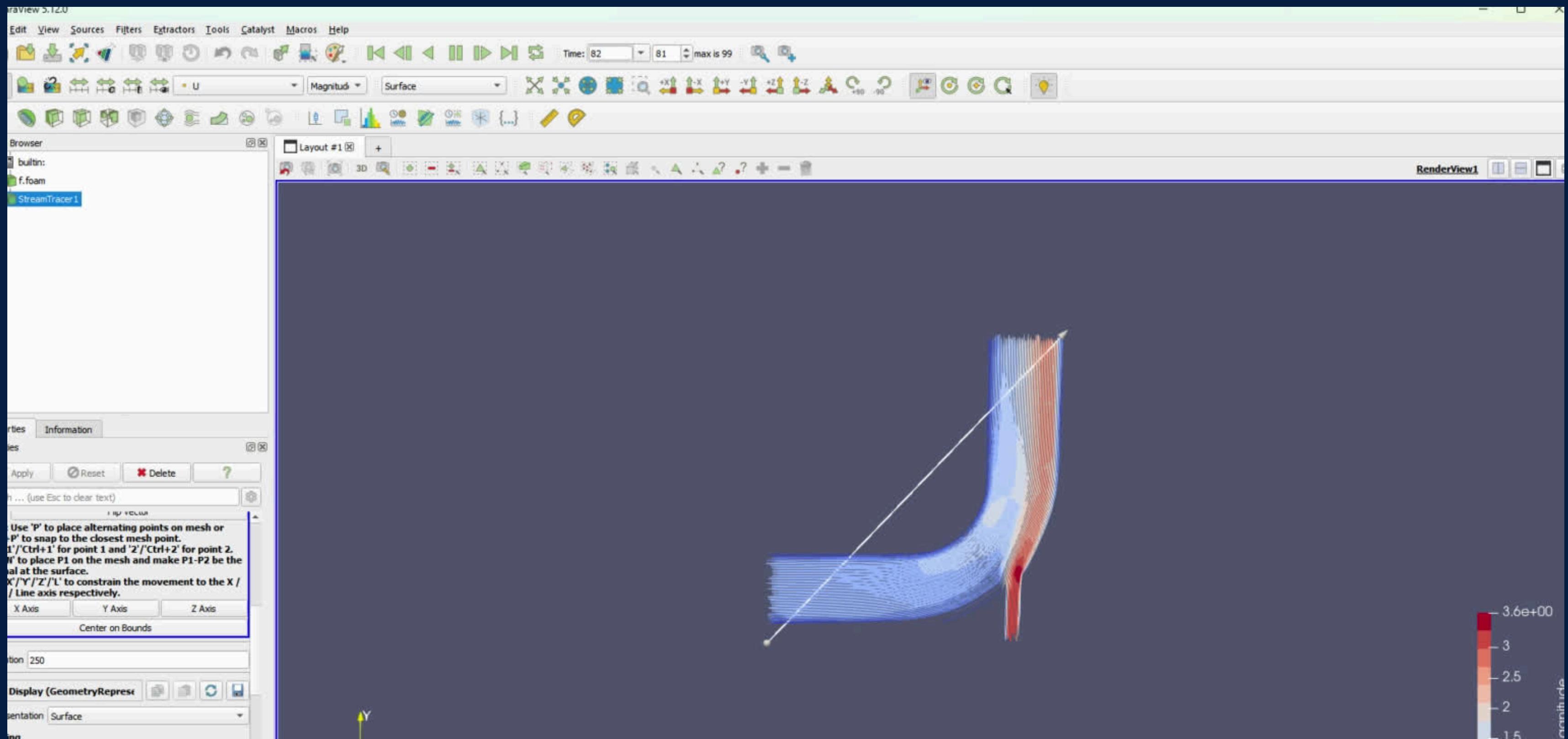
1. Velocity Fields :

- Inlets show smooth and uniform velocity due to zeroGradient, ensuring consistent flow entry.

2. Pressure Fields :

- Outlets maintain constant pressure via fixedValue, ensuring controlled pressure.
- Pressure drop along the elbow, typical for fluid flow through curves.

● Result

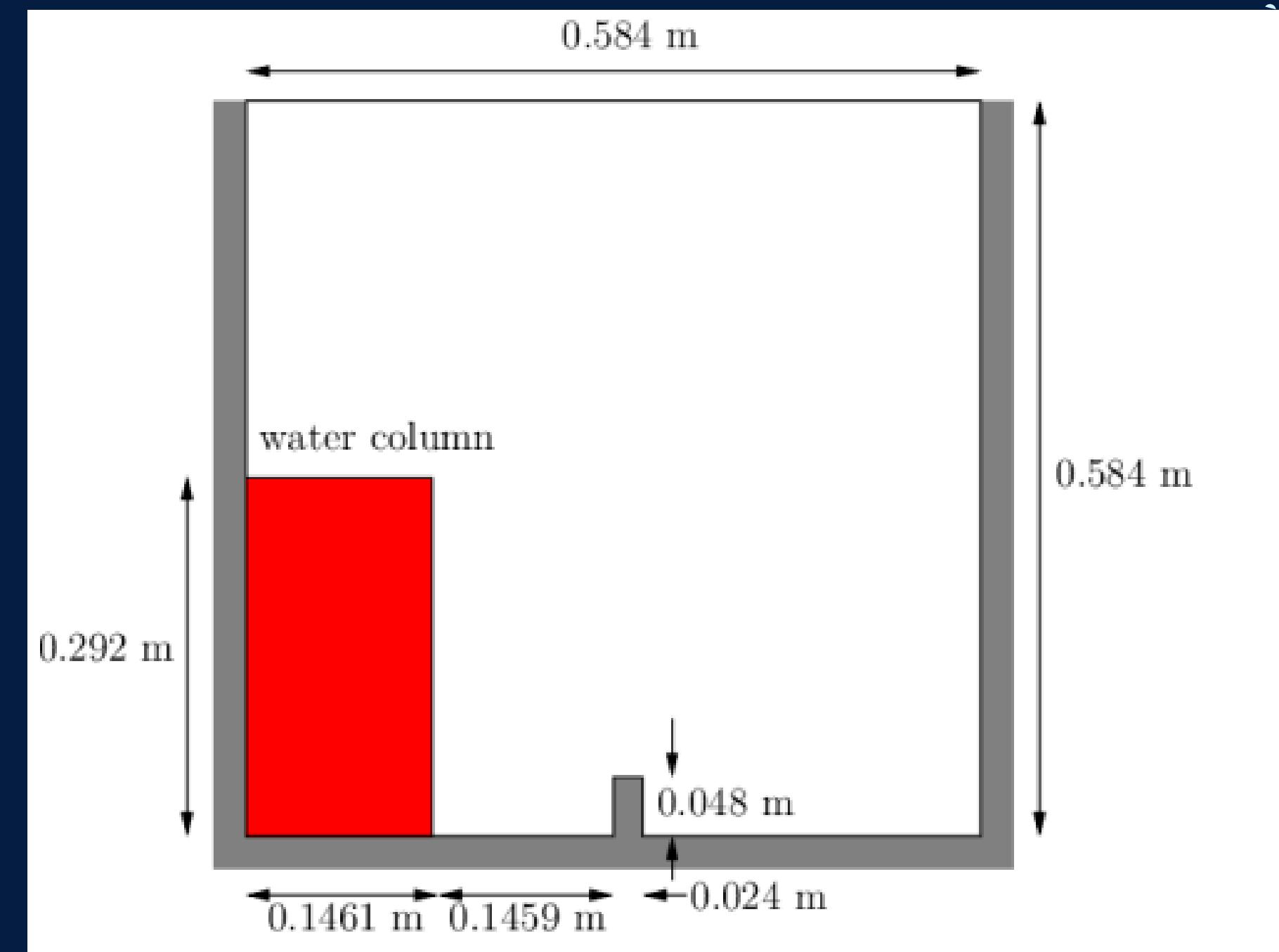


VOF METHOD

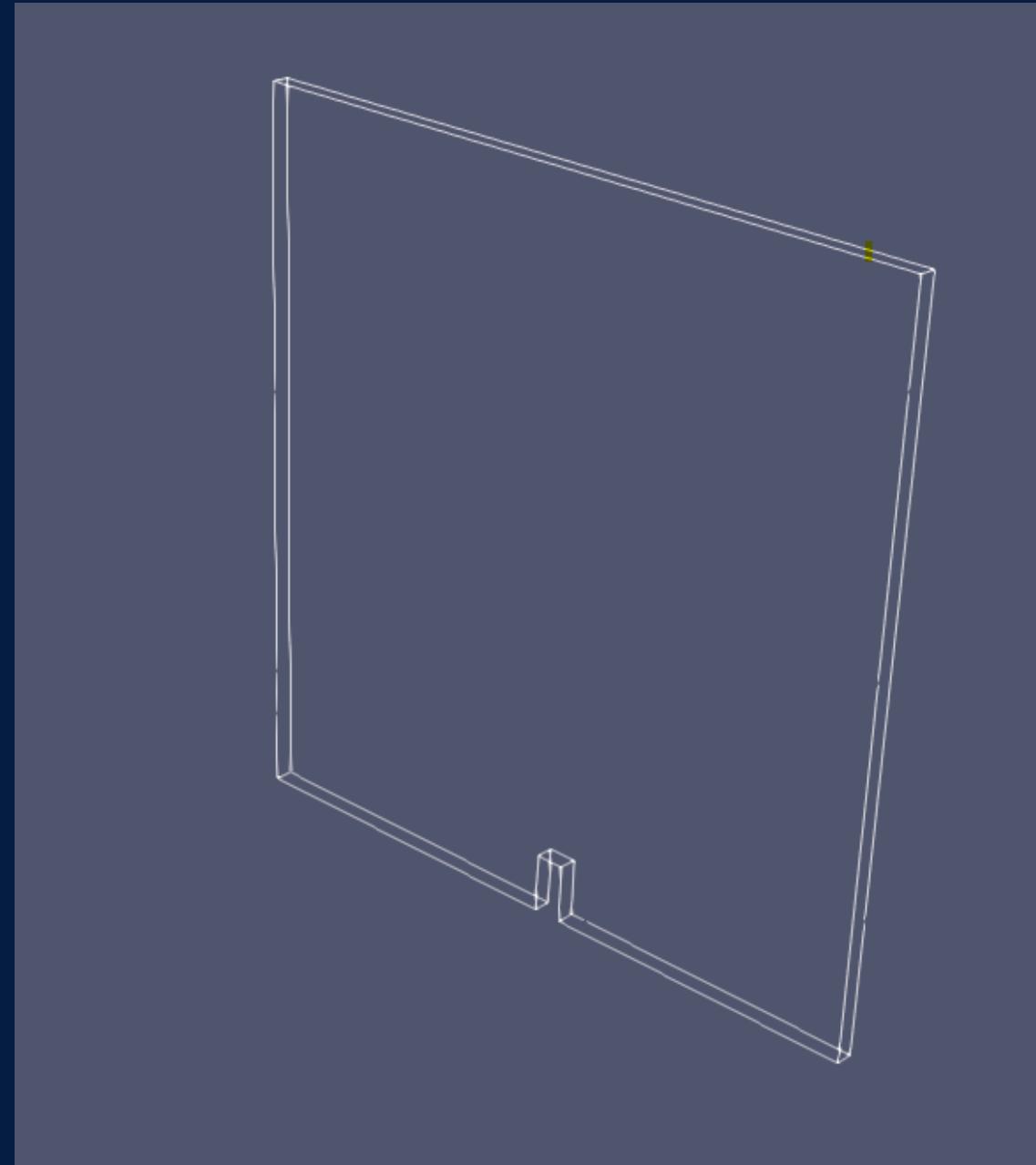
- Problem -4 (Dam-break)

The test setup consists of a column of water at rest located behind a membrane on the left side of a tank. At time $t = 0 \text{ s}$, the membrane is removed and the column of water collapses. During the collapse, the water impacts an obstacle at the bottom of the tank and creates a complicated flow structure, including several captured pockets of air.

Our goal is to study that complicated structure.

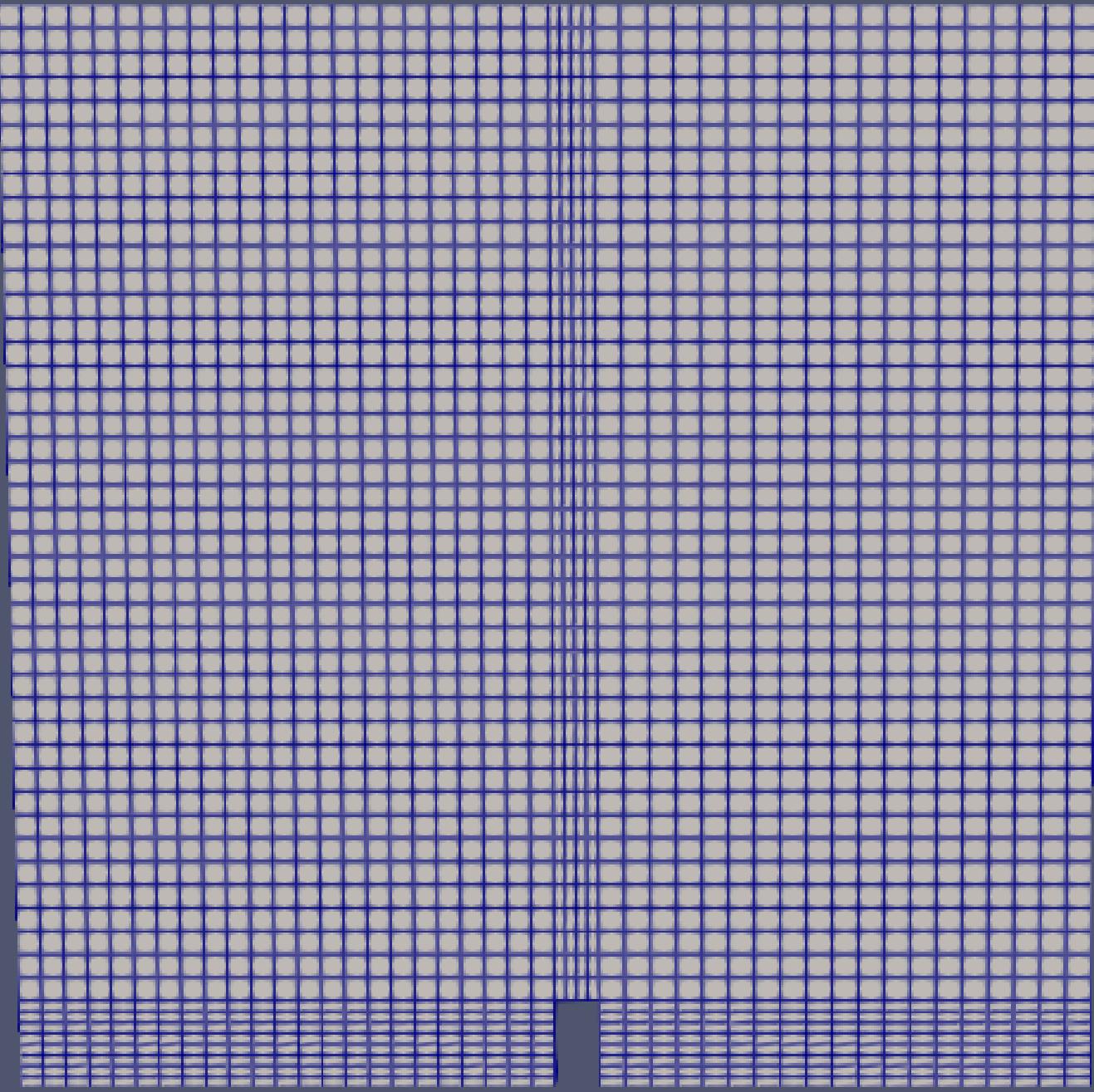


• Computational Domain



- left Wall
(0 12 16 4)
(4 16 20 8)
- right Wall
(7 19 15 3)
(11 23 19 7)
- lower Wall
(0 1 13 12)
(1 5 17 13)
(5 6 18 17)
(2 14 18 6)
(2 3 15 14)
- atmosphere
(8 20 21 9)
(9 21 22 10)
(10 22 23 11)

- Type of mesh



The computational mesh is structured, regular, rectangular grids. The Domain is divided into 5 blocks each having finite no. of cells.

```
blocks
(
    hex (0 1 5 4 12 13 17 16) (23 8 1) simpleGrading (1 1 1)
    hex (2 3 7 6 14 15 19 18) (19 8 1) simpleGrading (1 1 1)
    hex (4 5 9 8 16 17 21 20) (23 42 1) simpleGrading (1 1 1)
    hex (5 6 10 9 17 18 22 21) (4 42 1) simpleGrading (1 1 1)
    hex (6 7 11 10 18 19 23 22) (19 42 1) simpleGrading (1 1 1)
);
```

• Boundary and initial Conditions

Pressure

- leftWall
fixedFluxPressure - uniform 0
 - rightWall
fixedFluxPressure - uniform 0
 - lowerWall
fixedFluxPressure - uniform 0
- INFERENCE-** The flux is specified at the boundary, and pressure is adjusted to maintain this flux.
- atmosphere
totalPressure - uniform 0
 - defaultFaces
empty

Velocity

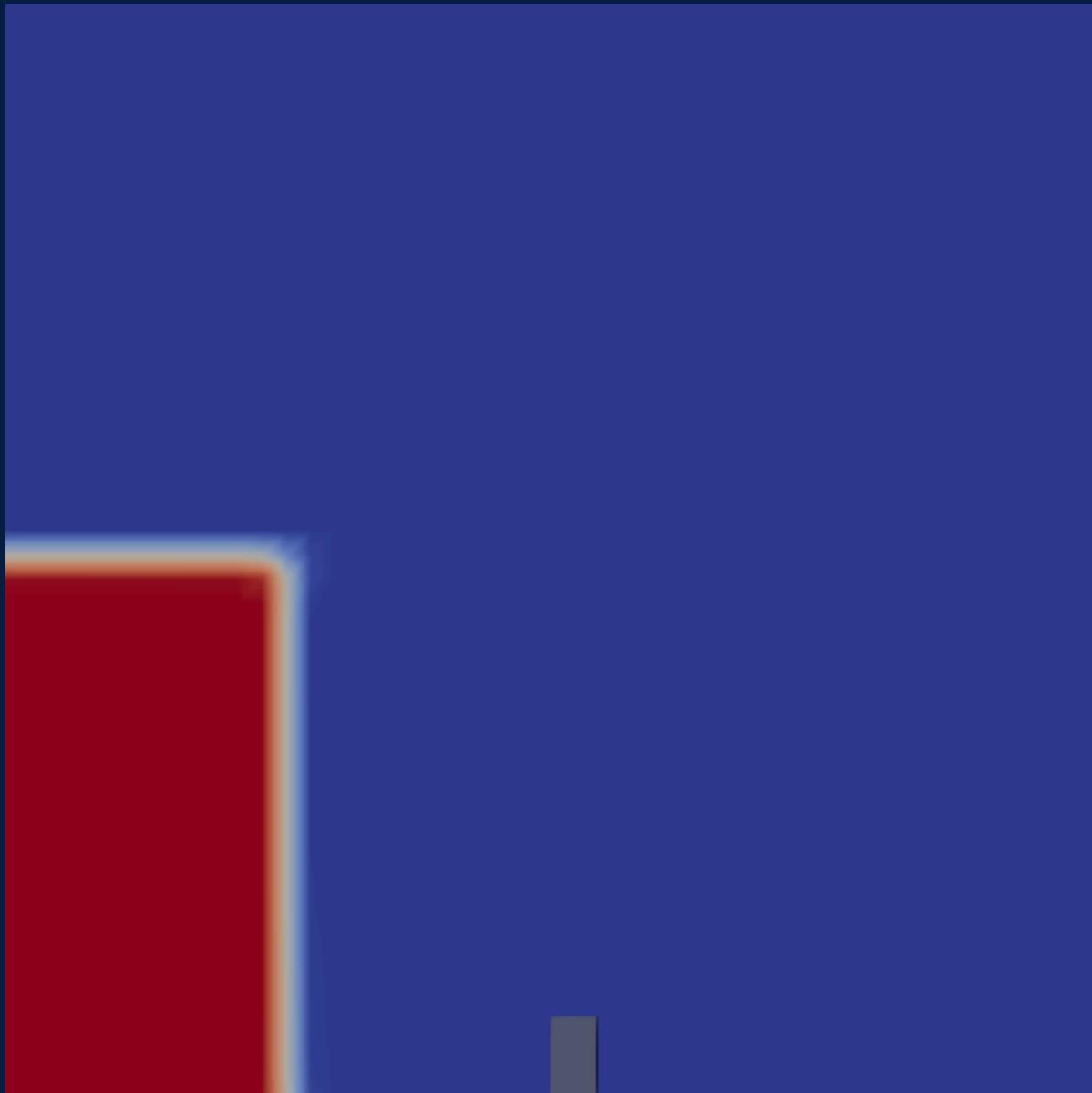
- leftWall
noSlip
 - rightWall
noslip
 - lowerWall
noSlip
- INFERENCE- velocity at wall is 0**
- atmosphere
pressureInletOutletVelocity - uniform (0 0 0)
- INFERENCE-**
- defaultFaces
empty

alpha.water

- internalField uniform 0
- leftWall
zeroGradient
 - rightWall
zeroGradient
 - lowerWall
zeroGradient
- INFERENCE-**
- atmosphere
inletOutlet
inlet - uniform 0
 - defaultFaces
empty

- Initial Set Fields :
value of volume fraction(f) or alpha, is set by taking primary fluid as water and secondary fluid as air.

$$f = 1 \text{ (water)} \quad f = 0 \text{ (air)}$$



```
defaultFieldValues
(
    volScalarFieldValue alpha.water 0
);
// domain is field with air as f = 0, region is shown by blue.

regions
(
    boxToCell
    {
        box (0 0 -1) (0.1461 0.292 1);
        fieldValues
        (
            volScalarFieldValue alpha.water 1
        );
    }
);

// columne of water is at initial condition is set in region having f =1, region
// shown by red color.
```

● Result

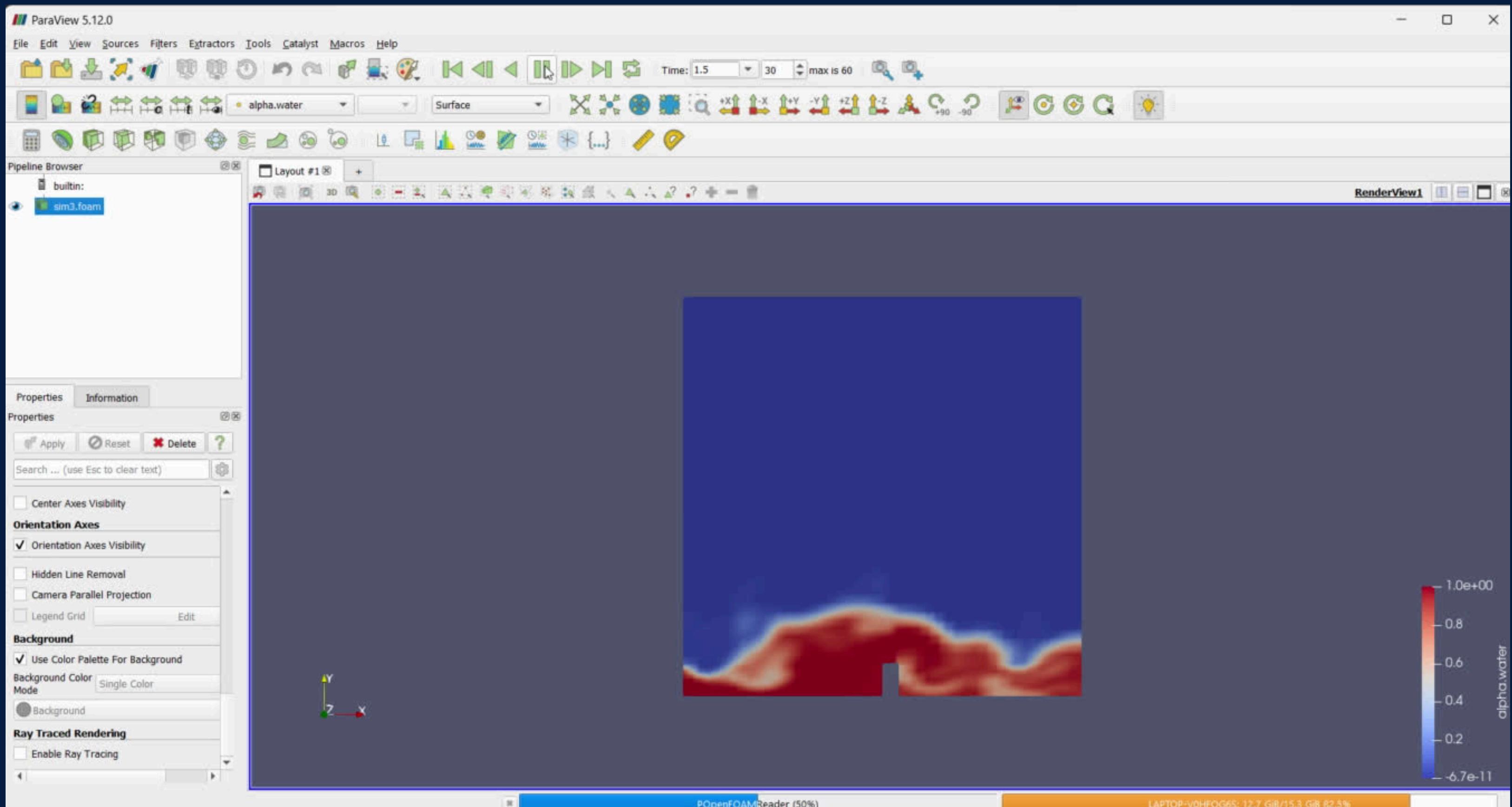
1. Early Collapse Phase:

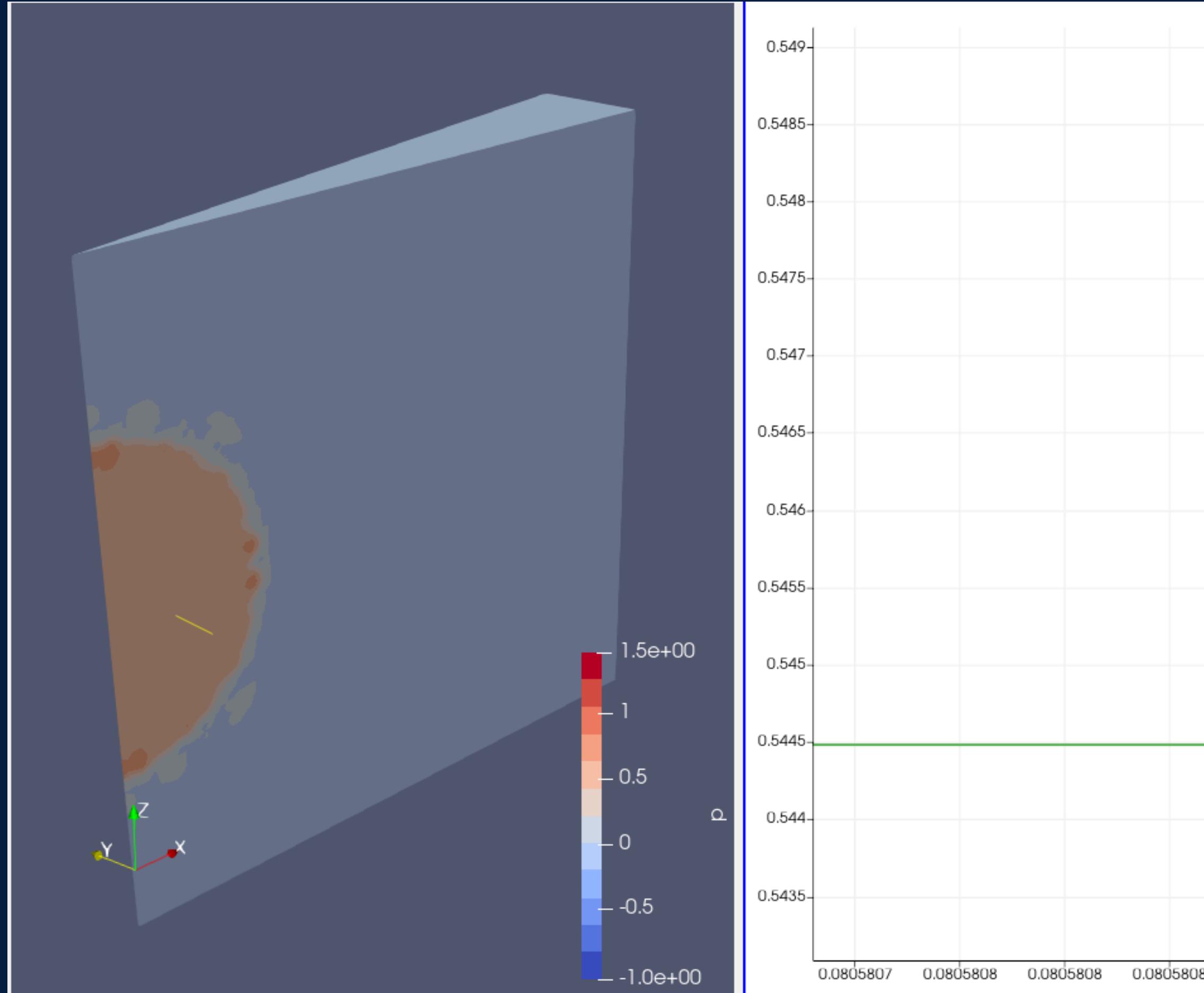
- Water spreads out, forming a complex interface.
- High-velocity jet forms, leading to horizontal spread.

2. Impact on Walls:

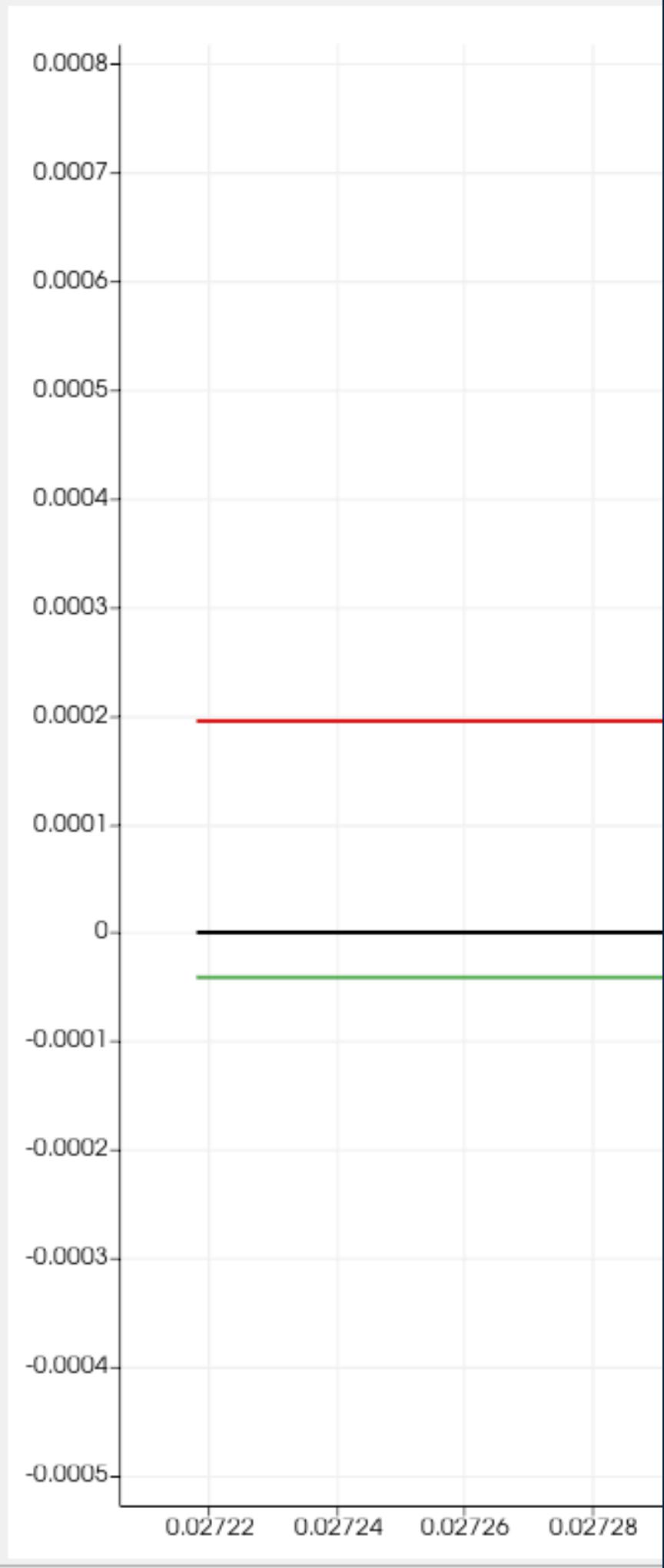
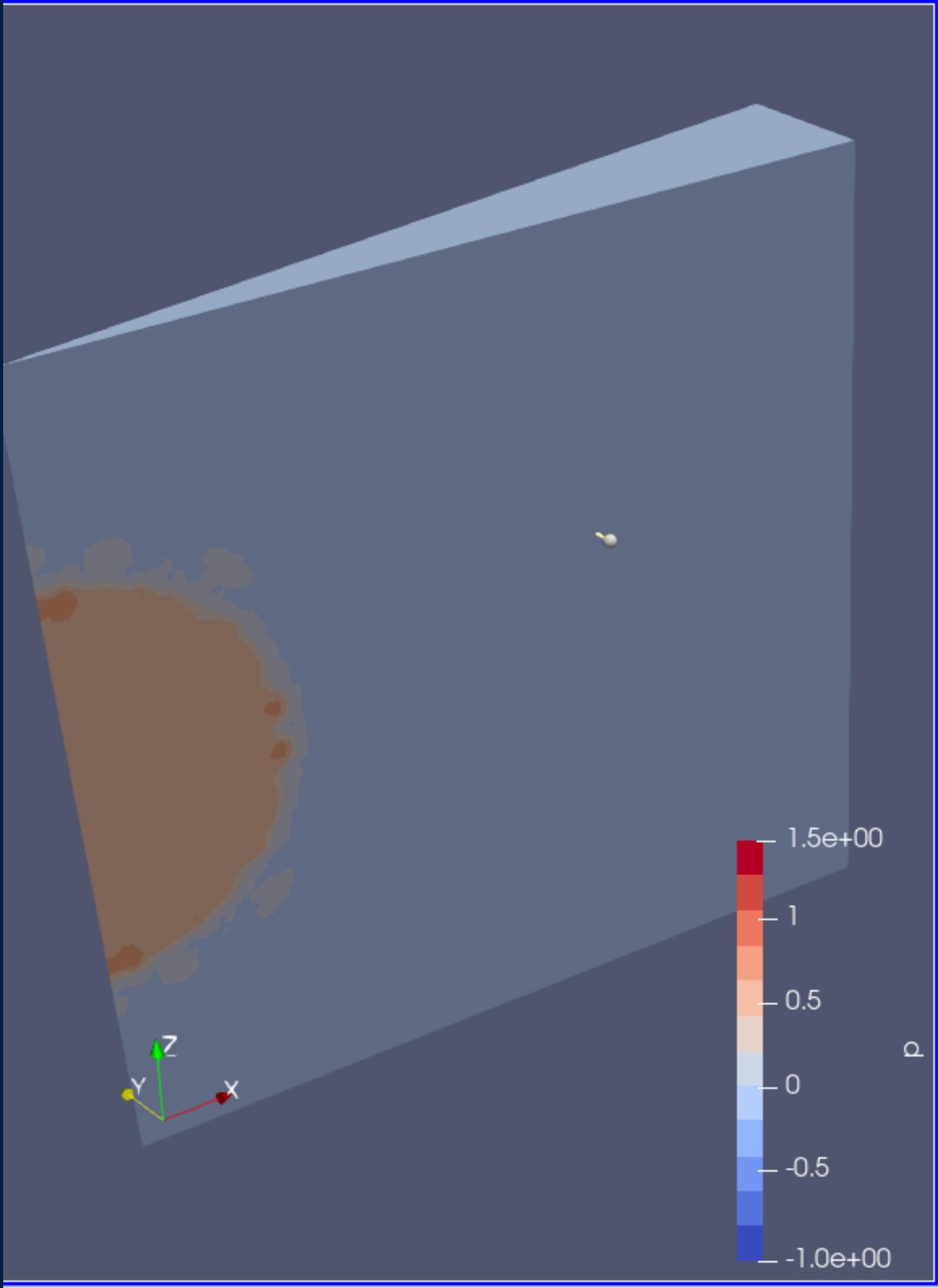
- Jet impacts the opposite wall, causing splashes and secondary flows.
- Air pockets and mixing regions form near the impact site.

• Result





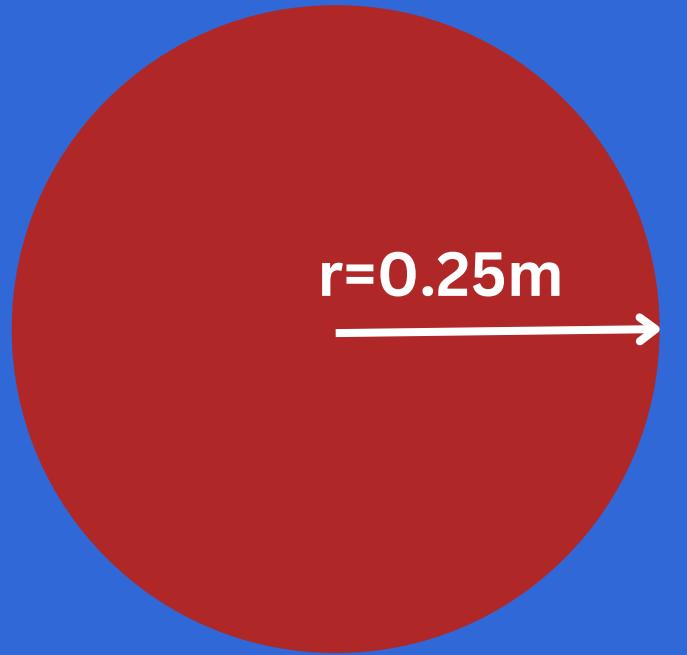
Pressure inside
water drop
=0.5445 N/m²



Pressure outside
water drop
= 0(approx)

- theoretically pressure difference =

$$\begin{aligned} & 2*\Sigma/\rho \\ & =2*(0.072)/(0.25) \\ & =0.576 \text{ N/m}^2 \end{aligned}$$



- Computational result pressure difference =

$$=0.5445 \text{ N/m}^2$$

- Error = 0.0315
- Error % = 3.15%