

# ME 412- CFD STAGE 1

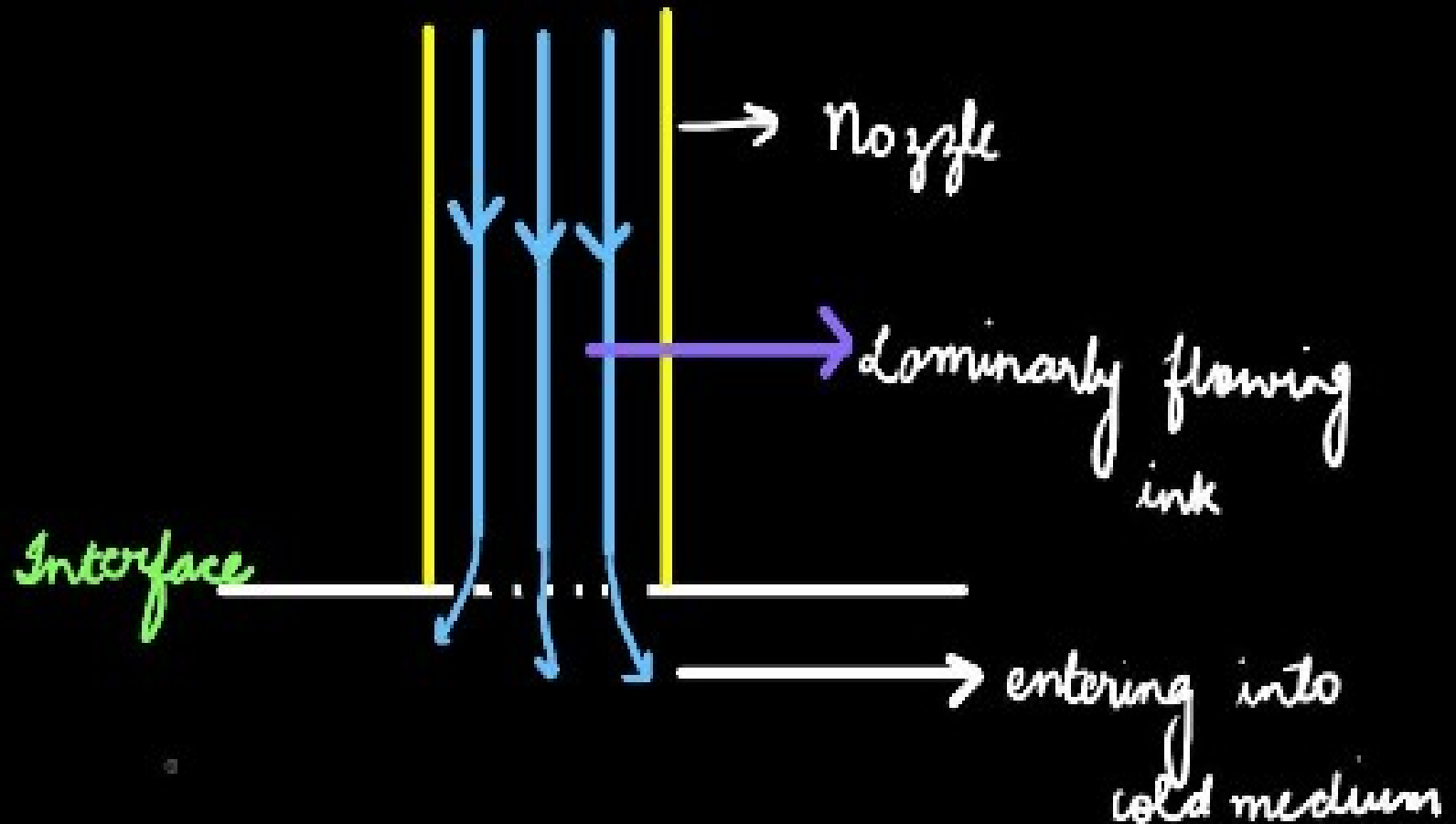
**KRITI CHATURVEDI**

**190020063**

# Objectives

- Simulate and study how a fluid jet acts when impinged into another medium.
- To Understand the fluid behaviour at the interphase
- Simulate a case very close to the actual case required
- Validate the solver for that case using a known case

# ACTUAL PROBLEM



# SOLVER and PHYSICS

- InterFoam Solver
- Incompressible
- Transient
- laminar and turbulent
- multi phase
- Immiscible
- VOF
- isothermal

# EQUATIONS

***Continuity equation:***

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

***Momentum equations:***

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \rho \mathbf{v}[2S] + F$$

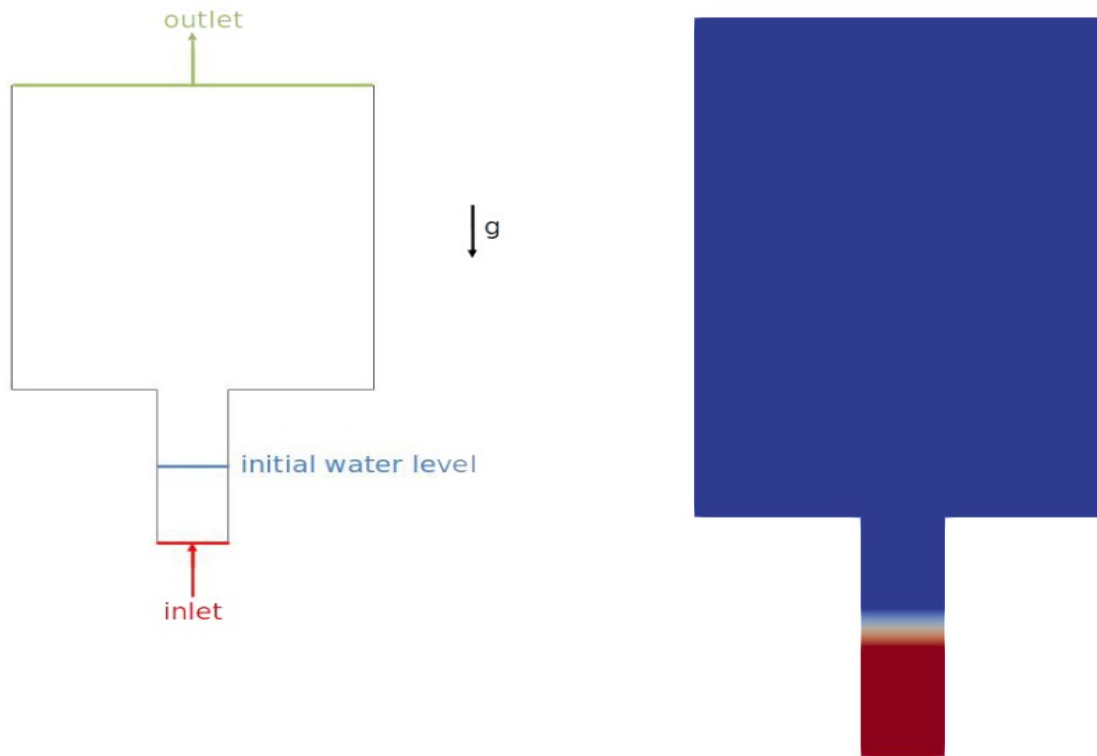
***Volume of Fluid:***

$$\rho = \alpha \rho_l + (1 - \alpha) \rho_g$$

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{u}) + \nabla \cdot (\alpha(1 - \alpha) \mathbf{u}_r) = 0$$

# NEAREST CASE

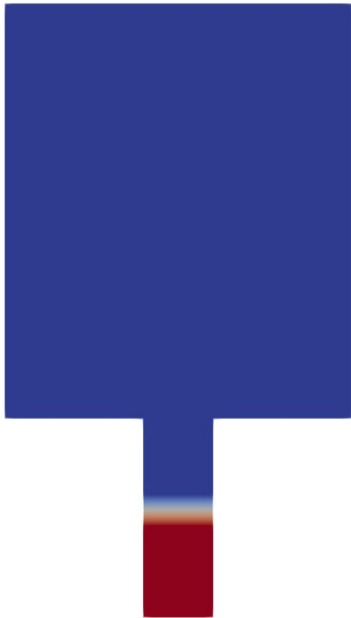
- Water Jet Simulation



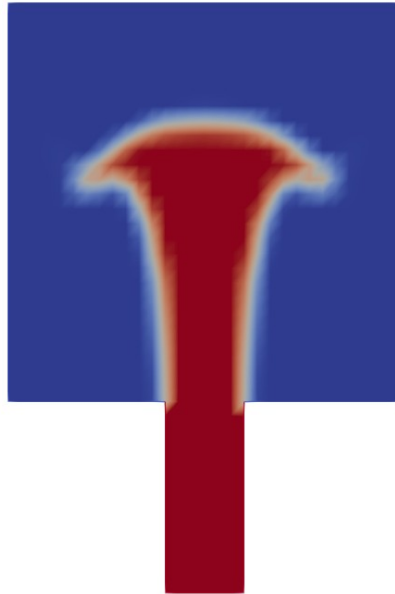
# Problem Description

## **Managable Domain :**

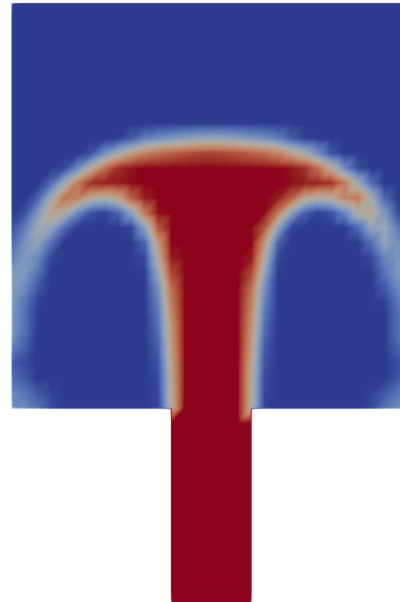
- Inlet
- Outlet
- Pipe
- Tank
- Coarse mesh was created using BlockMesh then SnappyHexMesh was used to refine it.
- Different regions were specified in the setFields Dict
- Different parts were made in paraview and saved as .stl file.
- Simulation will be carried out in Open Foam using interFoam Solver



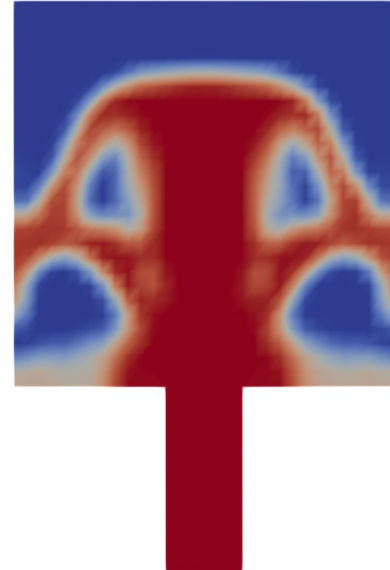
• T=0s



T=0.5s



T=0.8s



T=1.6s



# Validation Case

## Dam Break problem (Tutorial Case)

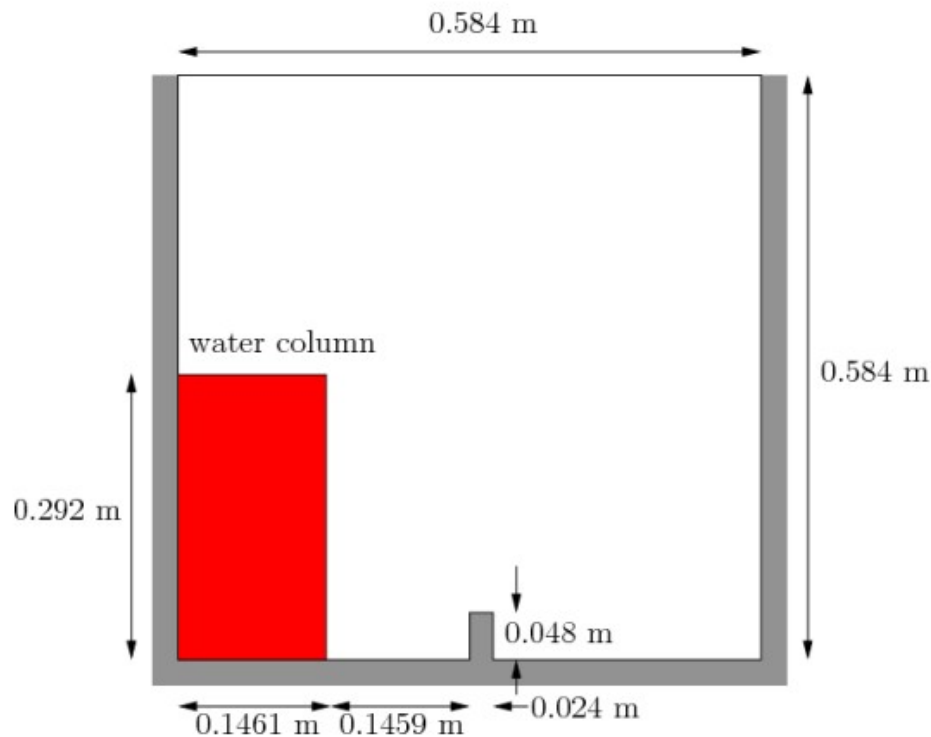


Figure 2.21: Geometry of the dam break.

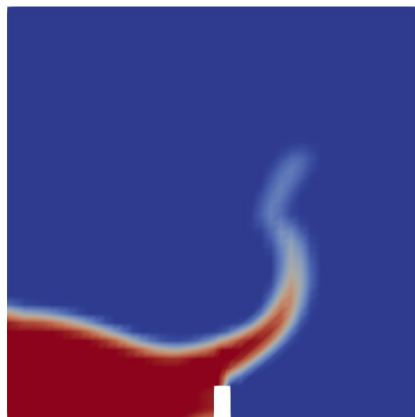
- Fluid Properties

water properties			
Kinematic viscosity	$\text{m}^2\text{s}^{-1}$	nu	$1.0 \times 10^{-6}$
Density	$\text{kgm}^{-3}$	rho	$1.0 \times 10^3$
air properties			
Kinematic viscosity	$\text{m}^2\text{s}^{-1}$	nu	$1.48 \times 10^{-5}$
Density	$\text{kgm}^{-3}$	rho	1.0
Properties of both phases			
Surface tension	$\text{Nm}^{-1}$	sigma	0.07

Table 2.3: Fluid properties for the *damBreak* tutorial



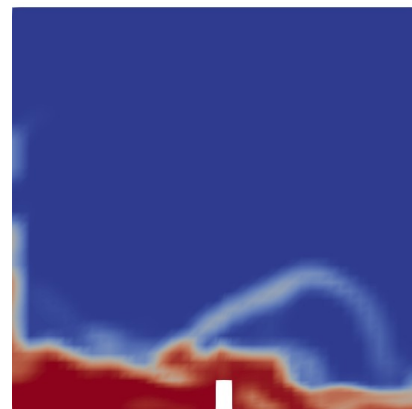
•  $t=0s$



$t=0.25s$



$t=0.5s$



$t=1s$

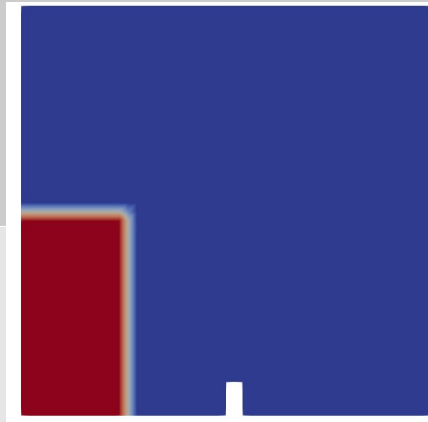
# COMPARING

TEST CASE



$t=0$

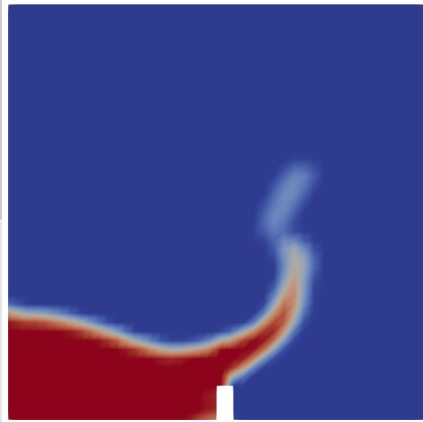
DAM BREAK



$t=0$

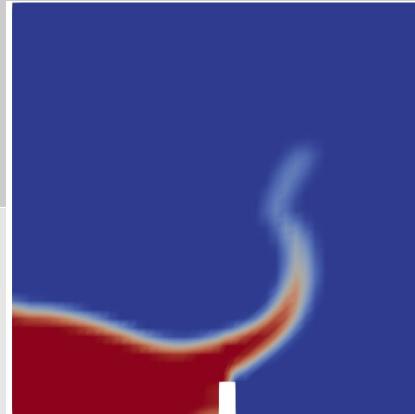
# COMPARING

TEST CASE



$T=0.25s$

DAM BREAK



$T=0.25s$

# COMPARING

TEST CASE

DAM BREAK

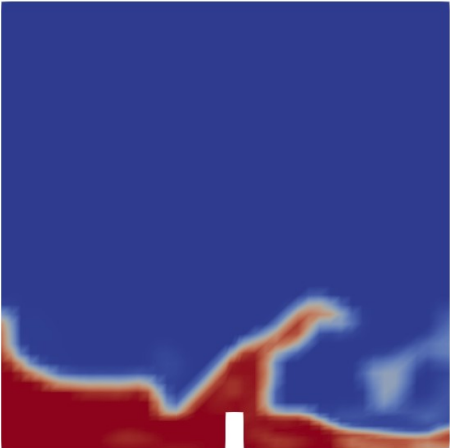
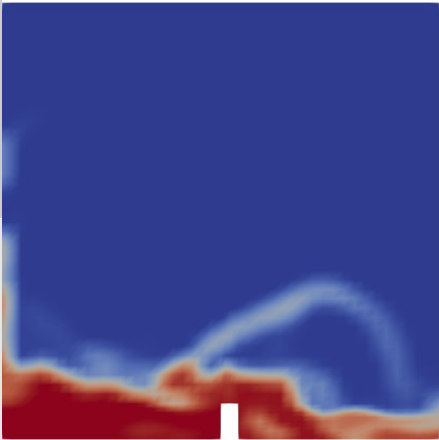


$T=0.5s$



$T=0.5s$

# COMPARING

TEST CASE		DAM BREAK	
	T=1s		T=1s

# ME 412- CFD STAGE 2

**KRITI CHATURVEDI**

**190020063**



# Additional Capabilities

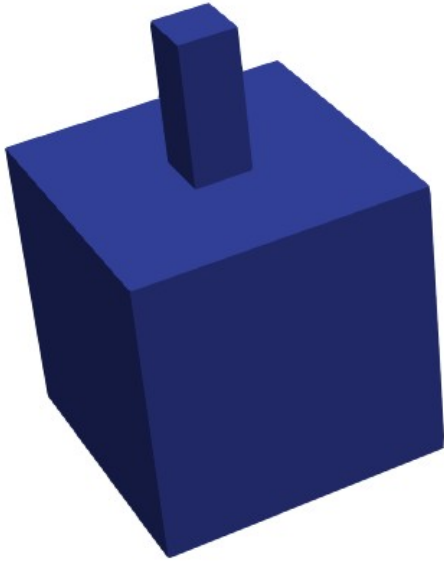
- Geometry Changed
- Fluid properties changed
- Ignored gravity
- Different initial water level
- Refined mesh

# GEOMETRY

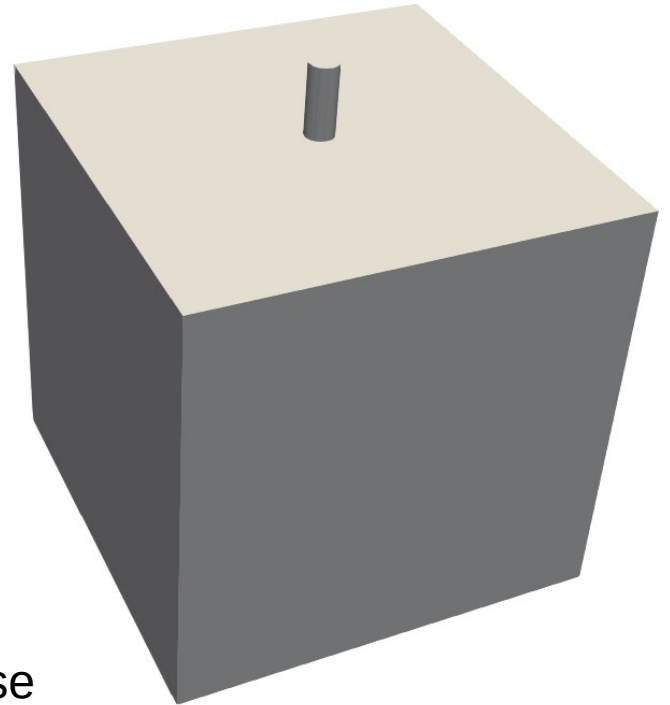
- Geometry was made in FreeCAD
- Meshed using snappyHexMesh
- Had to enable snap feature and increased the refinement level from (0,0) to (1,1)

Test Case	Project Case
1)Cuboidal pipe 2)l=b=0.2m, h=0.5m 3)Cubical Tank (1mx1mx1m)	1)Cylindrical pipe D=10m H=50m 2)Cubical Tank (300mx300mx300m)

# GEOMETRY COMPARISON

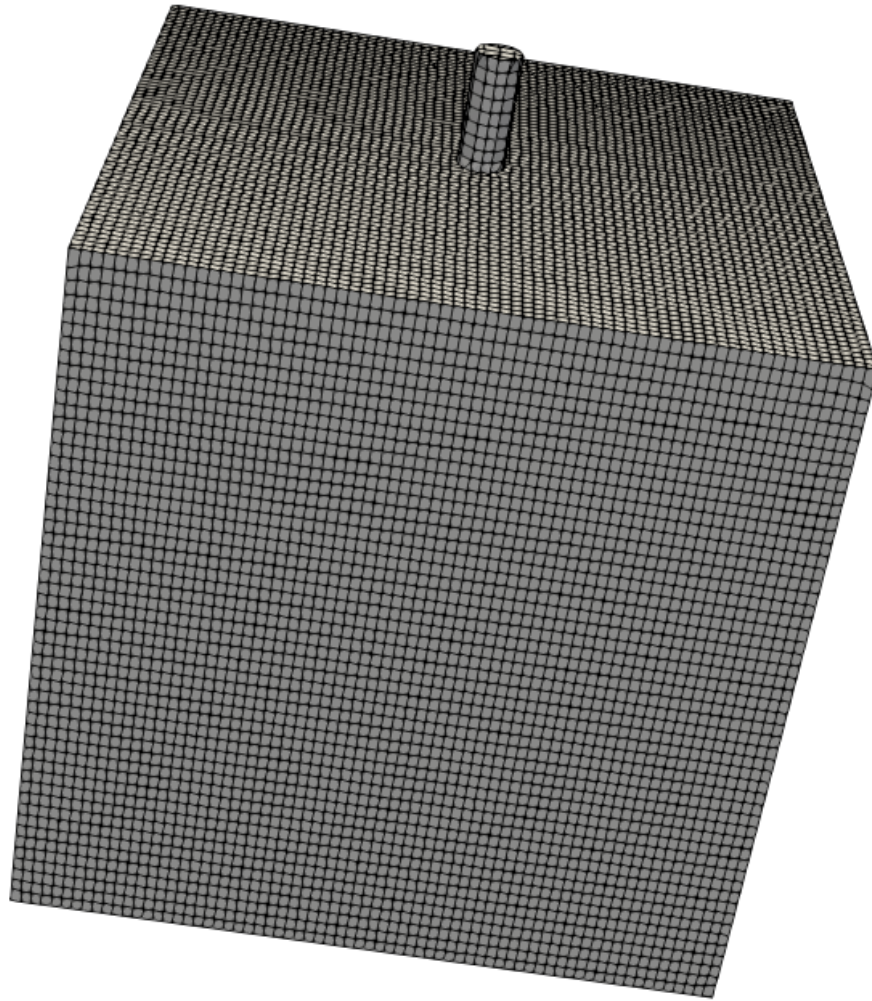


Test Case



Project Case

# MESH

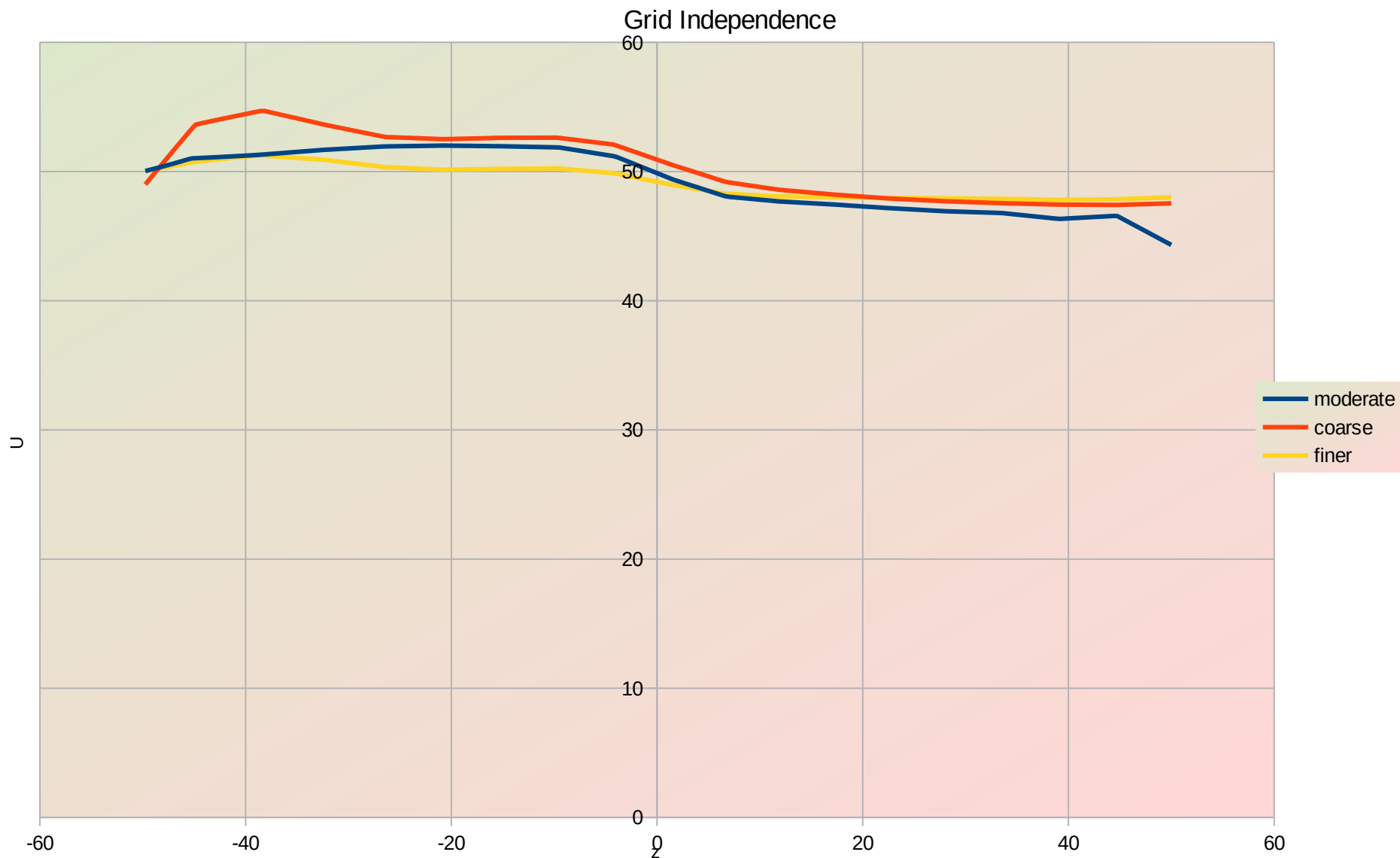


# Grid Independence

- Three grids were made by changing the values of number of blocks in the blockMeshDict file.

Grid	Specification	• No. of Cells
Grid 1 (coarse)	(25, 25, 40)	• 100240
Grid 2 (moderate)	(30, 30, 40)	• 140508
Grid 3 (fine)	(35, 35, 40)	• 181094

# U v/s Z



# Grid Independence

- U was plotted at 5 seconds
- Error analysis
- Average Absolute Error was calculated for each of the pair of mesh
- It was observed that the error decreases as we move towards finer mesh
- Fine mesh is used for further analysis as it has least error

Coarse-Fine	Coarse-Moderate	Moderate-Fine
1.4463	1.1964	1.0173

# Defining Case

- Acceleration due to gravity ( $g$ ) is taken to be 0 to ignore gravity
- Transport Properties
- Initial and Boundary Conditions
- Defining the Domain



# Transport Properties

- Transport Model – Newtonian
- Surface Tension Ink/Fluid Matrix – 0.03

Property	• Ink	Fluid
Kinematic Viscosity (m <sup>2</sup> /s)	1e-06	4e-6
Density (kg/m <sup>3</sup> )	1000	1000

# Initial and Boundary Conditions

- **U (m/s)**

Inlet:

type	fixedValue;
value	uniform (0 0 50)

Outlet:

type	pressureInletOutletVelocity
value	uniform (0 0 0)

Volume:

type	noSlip
------	--------

- **P\_rgh**

- Inlet:

• type	fixedFluxPressure
• value	uniform 0

- Outlet:

• type	totalPressure
• p0	uniform 0

- Volume:

• type	fixedFluxPressure;
• value	uniform 0;

- **alpha**

Inlet:

type	fixedValue
value	uniform 1

Outlet:

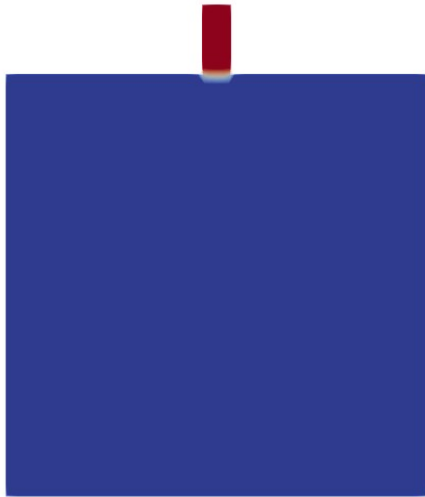
type	inletOutlet
inletValue	uniform 0
value	uniform 0

Volume:

type	zeroGradient
------	--------------

# Contour Plots

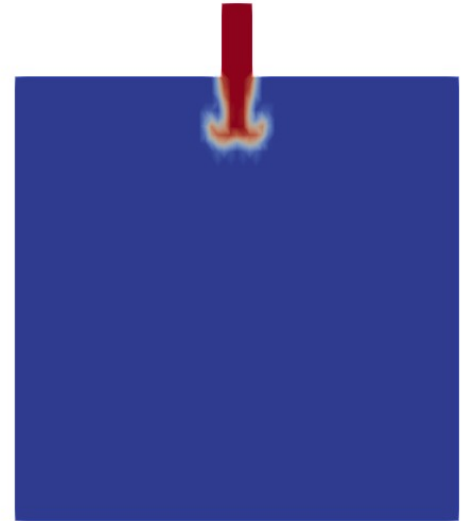
Plot of  $\alpha(\text{water})$ , inlet velocity=50m/s



t=0

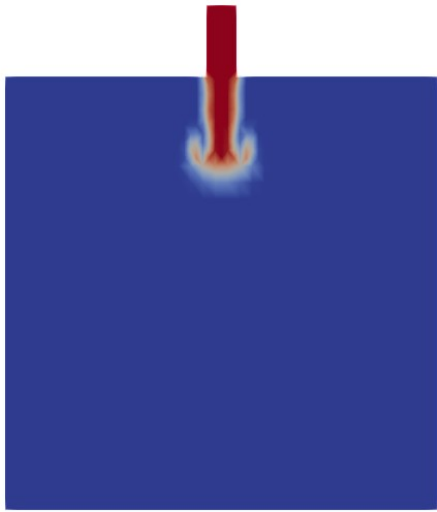


t=1s

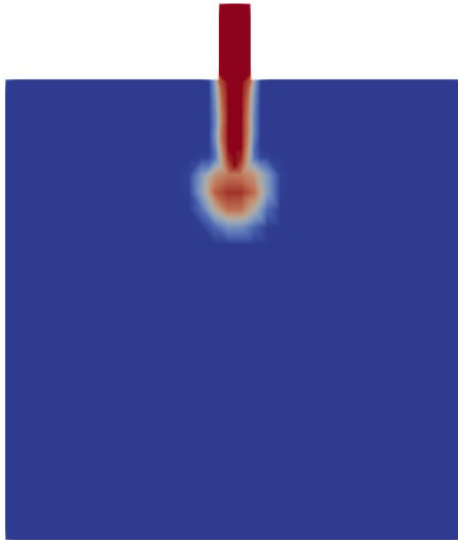


t=2s

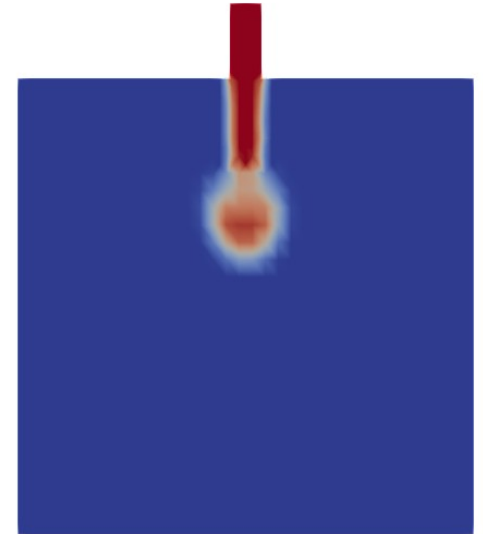
# Contour Plots



$t=3s$



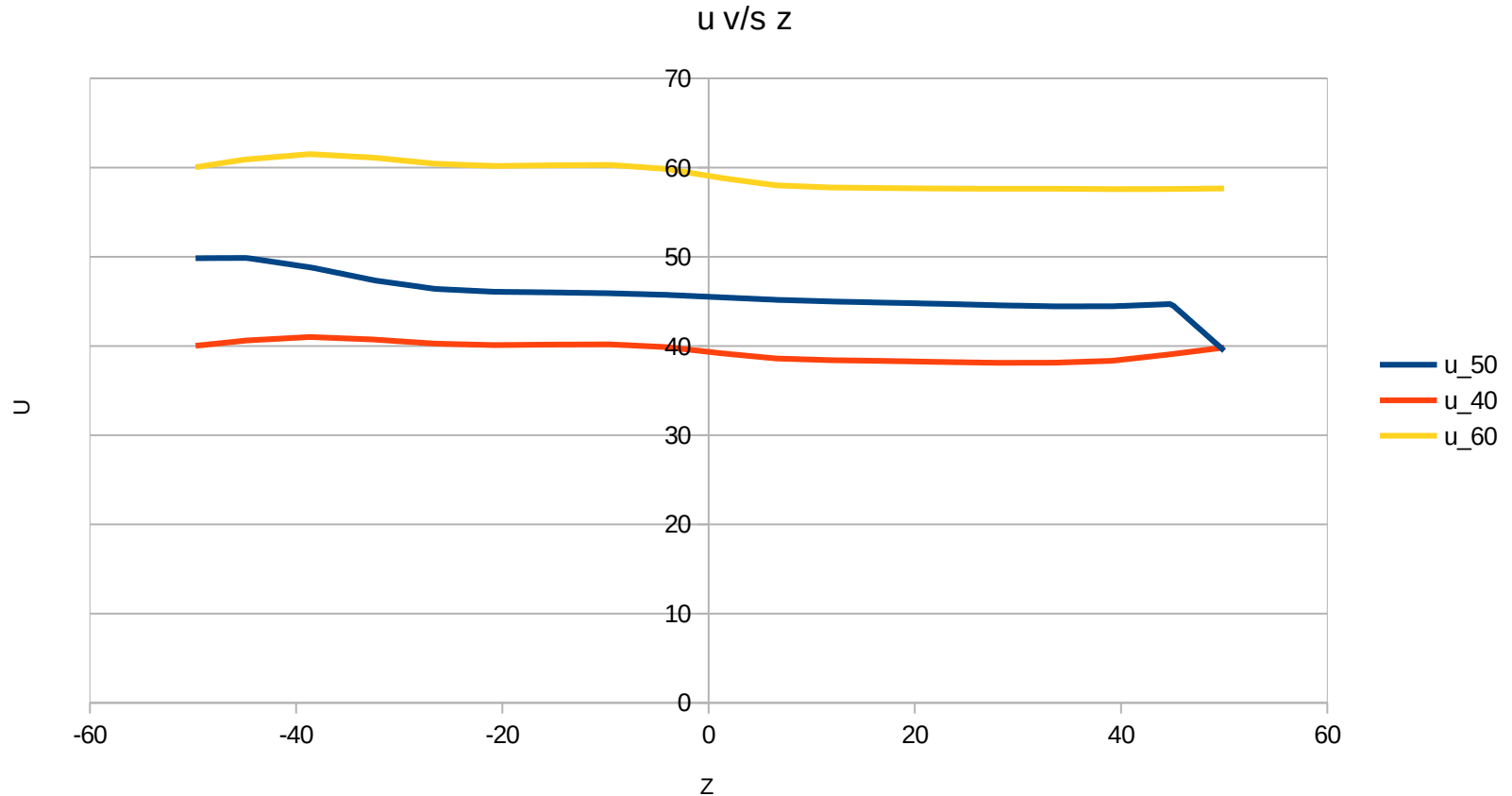
$t=4s$



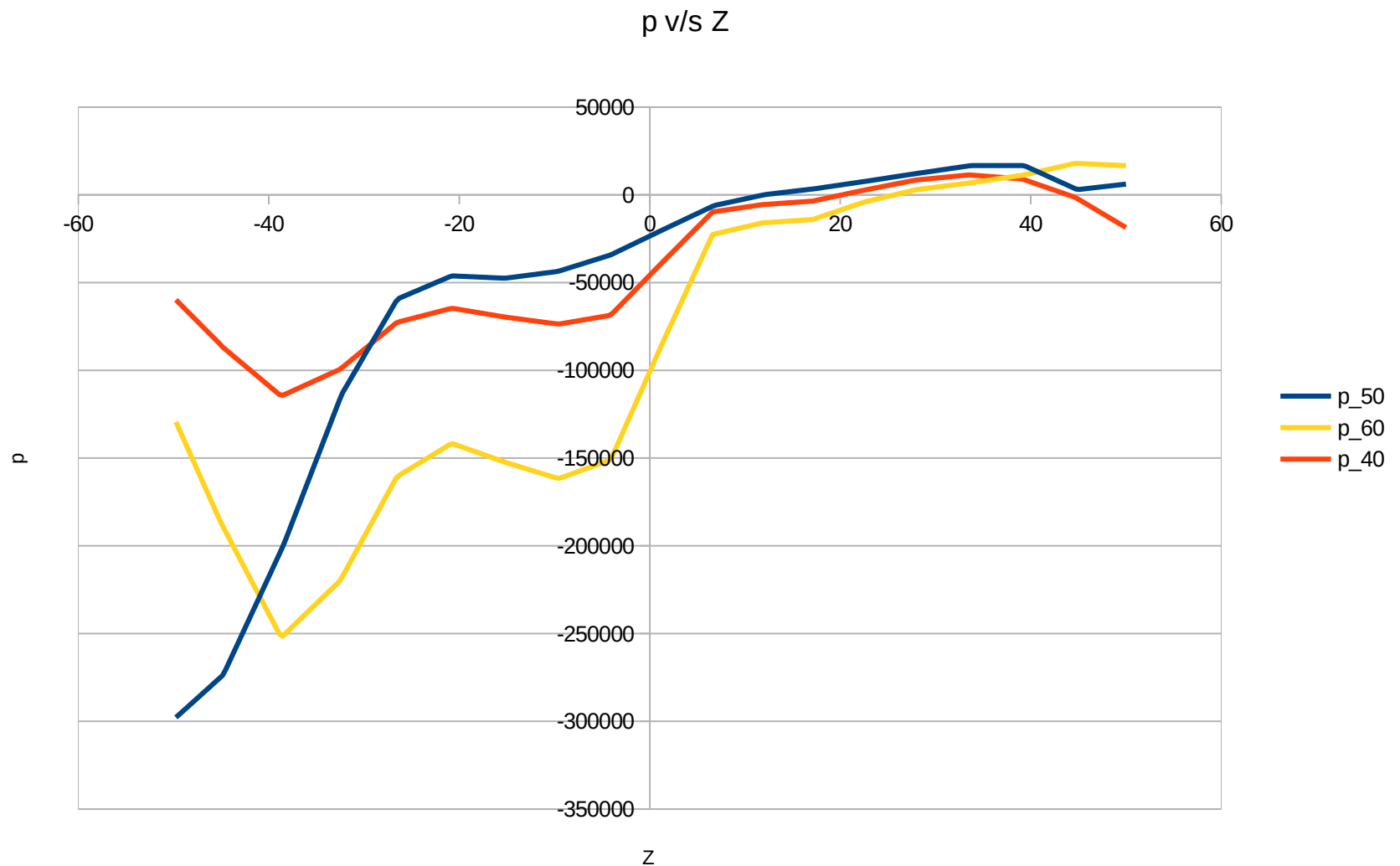
$t=5s$

# Parametric Analysis

- $U$  is chosen as a parameter.
- 3 values of  $u$  are taken 40, 50, 60 m/s



# Parametric Analysis



# References

- <https://cfd.direct/openfoam/user-guide/v6-dambreak/>
- [https://wiki.openfoam.com/Multiphase \(VOF\) Simulation Project by Jozsef Nagy](https://wiki.openfoam.com/Multiphase_(VOF)_Simulation_Project_by_Jozsef_Nagy)
- Verification of the open package OpenFOAM on dam break problems  
A.Zh. Zhainakov1 and A.Y. Kurbanaliev