Name of the Team: Panchayat
Name of Institute: National Institute of Technology Durgapur
Department: Mechanical Engineering
Country : India
Team Leader Email: quantummaniac609@gmail.com

1. Problem statement

Dam break for multiphase flows- A simple 2D block (with the standard one-cell depth in the z-direction), but with a small obstacle located at the bottom. The geometry consists of four blocks filled with Newtonian phases of water, oil, Galistan, and air. All phases are initially located behind membranes which are then removed simultaneously at t=0 and the fluids collapse onto each other and the obstacle. This creates a complicated mixture of the four different phases where the interaction between the phases needs to be interpolated and the contact angles are calculated.

2. CAD

Created a 2D geometry representing the dam. The domain consists of four blocks (for water, oil, Galistan, and air) and an obstacle at the bottom.

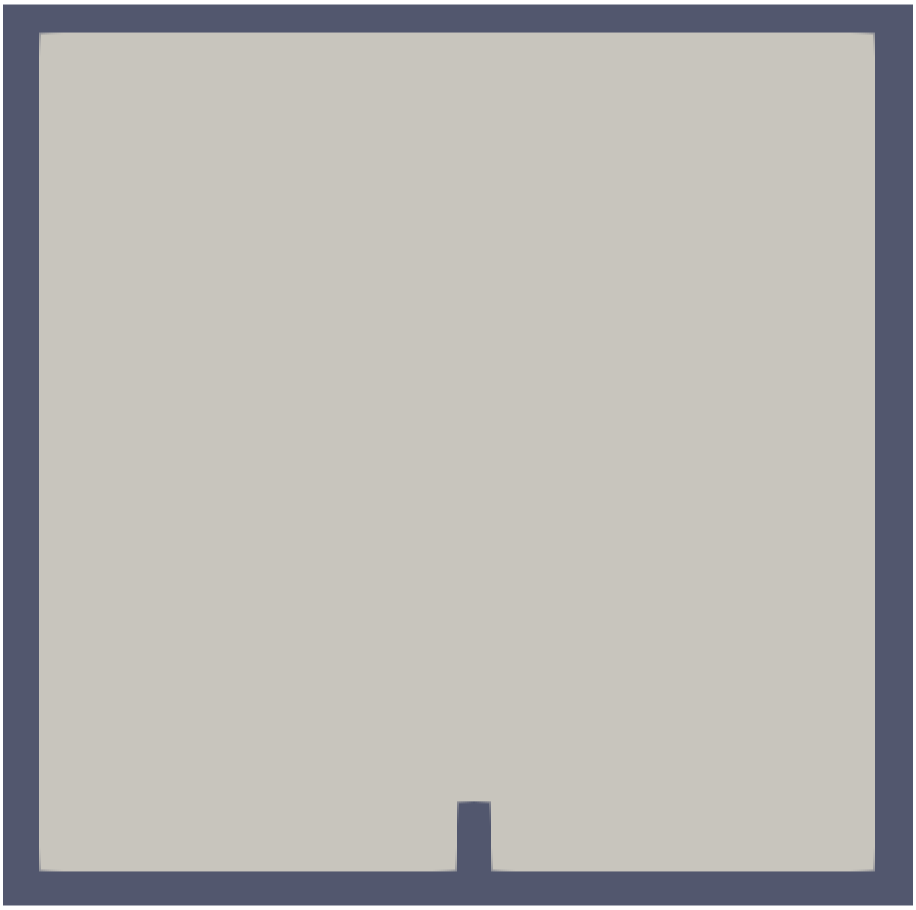

Fig 1. CAD Model

3. Meshing

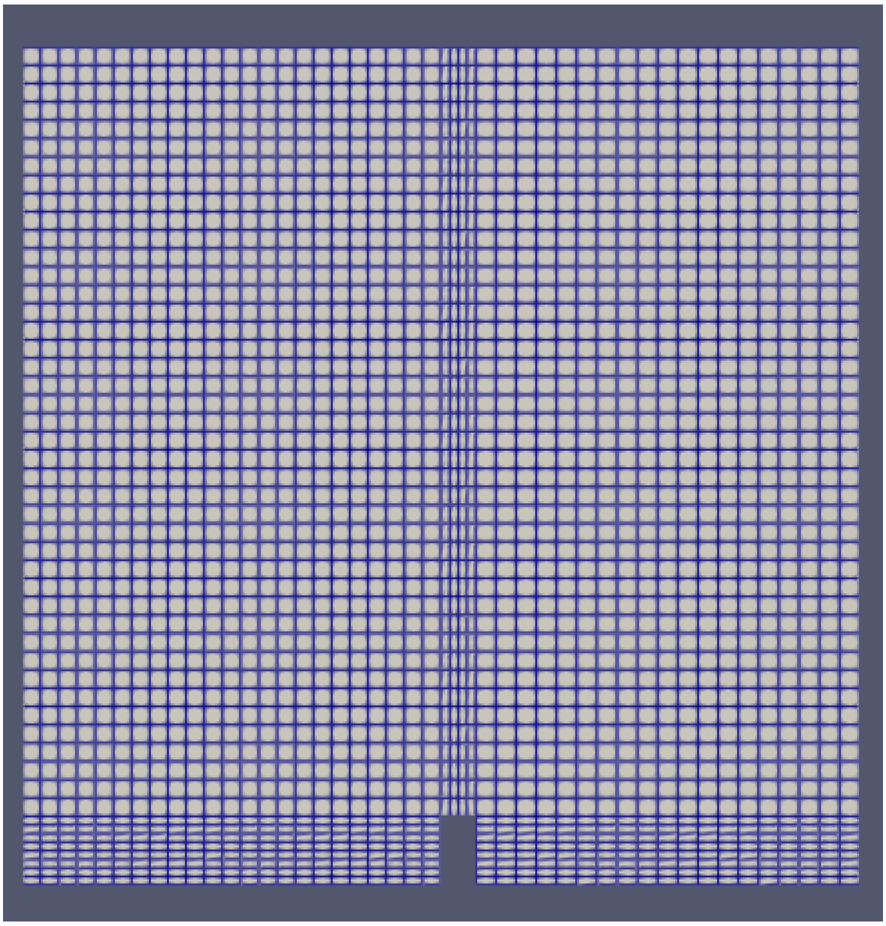

Fig 2. Structured meshing performed

Generated a computational grid for the 2D domain to discretize the problem. A good quality structured mesh is produced where the total number of cells are 2,268.Generated a computational grid for the 2D domain to discretize the problem.

4. Physics

The solver used for this problem is *interFoam* , which handles incompressible, non-isothermal immiscible fluids with phase-change. This solver is based on the volume of fluid (VOF) phase-fraction based interface capturing approach. There are five boundary patches: leftWall, rightWall, lowerWall, atmosphere, and defaultFaces. The atmosphere patch has no special attributes, and the defaultFaces patch is empty since it is not relevant for this 2D case.In the transportProperties file, fluid properties are defined in the constant directory. The material properties for each fluid. The transport model is selected with the transportModel keyword. For Newtonian fluids, the kinematic viscosity is given by the keyword nu. Gravitational acceleration is uniform across the domain.

5. Discretization Schemes

. For the simulation we have used Spatial Discretization:

6. Data Visualization

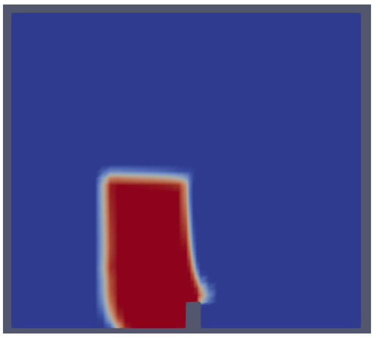

Fig3.1: at 0.05 sec

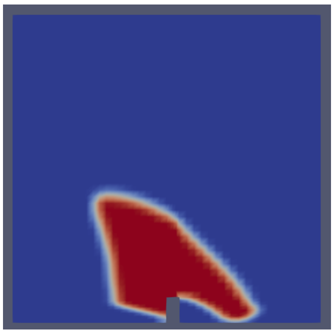

Fig3.2: at 0.15 sec

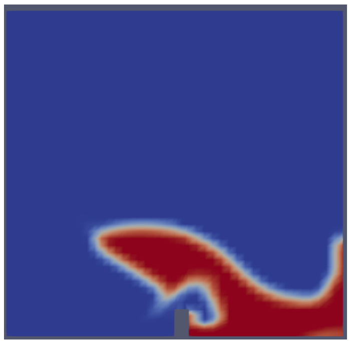

Fig3.3: at0.25 sec

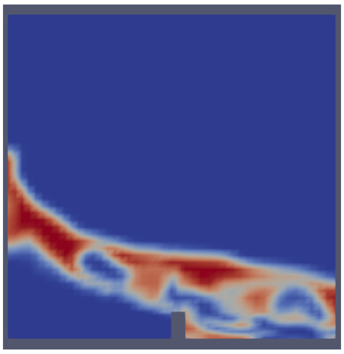

Fig3.1: at 1 sec

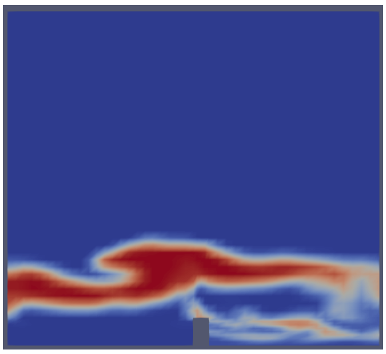

Fig3.2: at 2 sec

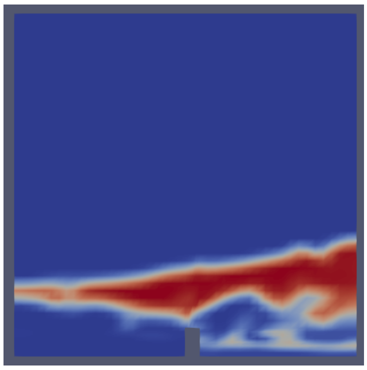

Fig3.3: at 3 sec

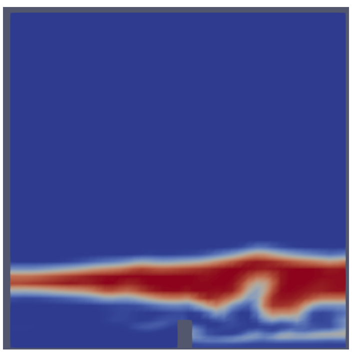

Fig3.1: at 4 sec

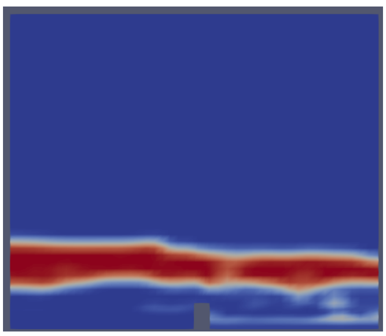

Fig3.1: at 5 sec

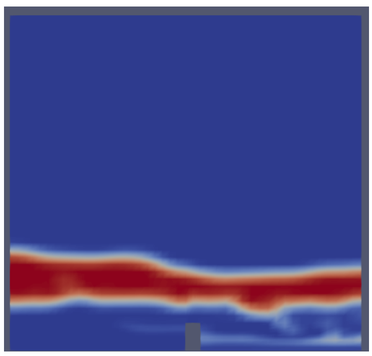

Fig3.1: at 6 sec

Fig 3. Time step analysis

Initial Conditions: Fluid is at rest or in initial movement state at t = 0

Flow Development: Gradual change in fluid motion as the flow develops over time.

Turbulent Regions: Identification of turbulent flow areas and vortex formation.

Steady State: Final stage where fluid flow reaches a steady state or continues to evolve dynamically.

7. Performance

The developed simulation for the 2D block with four Newtonian phases performed excellently in capturing the dynamics of fluid interaction and contact angle calculation.

The accurate representation of phase behavior and efficient computational performance validate the effectiveness of the simulation model for studying complex fluid mixtures.

8. Accuracy

- The simulation effectively replicated the expected physical phenomena, demonstrating high accuracy in phase interaction and contact angle determination.
- The complex fluid mixture behavior and the response to the obstacle were well-represented.