FOSSEE, IIT Bombay
OpenFOAM Research Migration Project
August, 2022

Numerical Modelling of Flow within Porous Media using OpenFOAM

Mena G Pillai[1] and Chandan Bose[2]

[1]Department of Ocean Engineering, Indian Institute of Technology Madras
[2]School of Engineering, The University of Edinburgh

August 3, 2022

Synopsis

The numerical modeling of the interaction of water waves with porous structures has been identified as one of the key challenges in coastal engineering research. Most coastal structures have a porous portion, and hence, a clear understanding of the porous media flow is highly significant in designing and investigating the stability of these porous coastal structures within the ocean environment. The simulation of multi-phase flow (water and air) through the porous media is a highly multifaceted process. In this study, a case study of the porous dam-break problem, conducted by [1] as illustrated in [2], was simulated in two-dimension. The modified Volume Averaged Navier–Stokes Equations, including the effect of the porosity, are solved. The model, governing porosity, is formulated for the intrinsic velocity inside the porous structure. The numerical method was based on a finite volume discretization on a collocated grid arrangement. The interface tracking is conducted using a volume of fluid approach (VOF). For the CFD modelling, the finite-volume based open source code OpenFOAM® (foam-extend 3.1) is used in combination with the porousWaveFoam solver [3] developed within Waves2Foam [4] toolbox. The performance of the porousWaveFoam solver is evaluated by comparing the results from the numerical simulation with those of experimental data. The results imply the use of a porous solver in investigating the fluid interaction with porous structures.

1 Introduction

Porous structures, such as porous beds and breakwaters, are widely used in the field of coastal and hydraulics engineering. The study of fluid interaction with these porous structures has been identified as one of the key challenges in coastal engineering research. A clear understanding of the porous media flow is highly significant in designing and investigating the stability of these porous coastal structures within the ocean environment. The simulation of multi-phase fluid (water and air) flow through the porous media is a highly multifaceted process. Porous media are characterized

by irregular pore shapes and sizes within a solid matrix. The porous media, consisting of a rigid skeleton and pores, is treated as one continuum which exerts forces on the fluid due to drag, friction, and acceleration. Analytical solutions become unviable for solving the fluid flow within these complicated geometries. Numerical modeling, on the other hand, proves to be an efficient tool for solving such complicated problems. The use of computational fluid dynamics (CFD) tools for solving fluid-structure interaction problems gained popularity in the last two decades. Besides, it is possible to predict and visualize the flow properties in microscopic scales with CFD solvers, which is almost impossible to measure in the laboratory environment. Full-scale simulations can be achieved by adopting CFD techniques even though computational time is still a key barrier. The key challenges which need to be tackled properly in the fluid-porous structure interaction problems are solving the fluid flow within these complex geometries, tracking the free surface, and wave generation and absorption at the boundaries of the computational domain.

In this study, the case study of the porous dam-break problem, conducted by [1] as illustrated in [2], is simulated in two dimensions. In the study conducted by [2], the wave interactions with the porous media were simulated by incorporating a combination of the FVM-VOF-DEM model. Firstly, an Eulerian finite volume method along with the VOF is developed to simulate the wave-structure interaction. In their model, the basic formulation of the NS equation developed was improved with a new formulation for the porous medium. The most accurate combination of the porous resistance terms in the NS equations was used in their study. For the CFD modelling, the finite-volume-based open source code OpenFOAM® (foam-extend 3.1) is used in combination with the porousWaveFoam solver developed by [3] within the waves2Foam toolbox [4]. The accuracy of the introduced numerical model is evaluated by comparing the surface elevation data inside and outside the porous medium obtained from the simulation with the laboratory results available in the literature. The model is based on the Volume Averaged Navier–Stokes equations, including the effect of the porosity, formulated for the intrinsic velocity inside the porous structure. The numerical method was based on a finite volume discretization on a collocated grid arrangement.

2 Governing Equations and Models

2.1 Volume Averaged Reynolds Averaged Navier-Stokes(VARANS) Equation

Flow within the porous media is described by the Reynolds-Averaged Navier-Stokes (RANS) equations. The volume averaging technique is applied to the RANS equations to incorporate the effect of porosity. For adopting RANS equations for solving the flow within the porous media, the pore size of the pores comprising the porous media should be well known, and also a very fine mesh size is required to capture the pore size which is not feasible for most of the times. Hence the volume averaging technique was adopted to the Navier-Stokes equations to a volume that was assumed to be larger than the length scale of the pores constituting the porous media [3]. The Volume Averaged Reynolds Averaged Navier-Stokes (VARANS) encompass the continuity equation and the momentum conservation equation. The fluid is assumed to be incompressible. The continuity equation is given by

$$\frac{\partial \langle \overline{u_i} \rangle}{\partial x_i} = 0, \tag{1}$$

where $\langle\overline{u_i}\rangle$ is the ensemble velocity averaged over the entire control volume and is referred to as the filter velocity or the Darcy velocity.

The momentum conservation equation can be expressed as follows:

$$(1+C_m)\frac{\partial}{\partial t}\frac{\langle\overline{u_i}\rangle}{n}+\frac{1}{n}\frac{\partial}{\partial x_j}\frac{\langle\overline{u_i}\rangle\langle\overline{u_j}\rangle}{n}=-\frac{1}{\rho}\frac{\partial\langle\bar{P}\rangle^f}{\partial x_j}+\frac{1}{n}\frac{\partial}{\partial x_j}\nu(\frac{\partial\langle\overline{u_i}\rangle}{\partial x_j}+\frac{\partial\langle\overline{u_j}\rangle}{\partial x_i})+g_j x_j\frac{\partial\rho}{\partial x_j}+F_i, \quad (2)$$

where C_m is the added mass coefficient which takes into account the momentary interaction between the solid grains and the water, n is the porosity, g is the acceleration due to gravity, ν is the kinematic viscosity, $\langle\overline{P}\rangle^f$ is the intrinsic pore pressure. The viscous term ν in the equation transfers the shear force and hence is significant near the interface of the porous media and the outer flow region [1]. The term F_i indicates the resistance forces (linear, nonlinear, and inertial forces) on the surface of the porous media solids. The linear and nonlinear forces are modeled using the extended Darcy-Forcheimmer equation as

$$F_i=a\rho\langle\overline{u_i}\rangle+b\rho\sqrt{\langle\overline{u_i}\rangle\langle\overline{u_j}\rangle}\langle\overline{u_i}\rangle, \quad (3)$$

where a and b are resistance coefficients. The resistance coefficients formulated by [5], where the effect of oscillatory flows was considered in terms of the KC number were used in the study and are given by:

$$a=\alpha\frac{(1-n)^2}{n^3}\frac{\mu}{\rho d_{50}^2}, \quad (4)$$

$$b=\beta\left(1+\frac{7.5}{KC}\right)\frac{1-n}{n^3}\frac{1}{d_{50}}, \quad (5)$$

where d_{50} is the mean grain size and KC = u_mT / (nd_{50}), where u_m is the maximum oscillating velocity and T is the period of the oscillation, α, and β are empirical coefficients that depend on the mean grain size, porosity, grain shape, and other factors. The added mass coefficient Cm, used for incorporating the inertial term given in the extended Darcy–Forchheimer equation into the momentum conservation equation given by equation (2) was formulated by [5] and is given as follows:

$$C_m=\gamma_p\frac{(1-n)}{n}, \quad (6)$$

where γ_p is an empirical coefficient, which takes the value 0.34.

2.2 Free Surface Modelling

The free surface is captured using the Volume of Fluid (VOF) method. A fixed grid is used where the fraction of the fluids in the cells around the interface is computed based on an advection equation. In the VOF method, an indicator phase function is demarcated in each computational cell which takes a value between 0 and 1 according to the volumetric fraction of the fluid inside the cell. In the scope of water-air interfaces, the value of the indicator phase function takes a value of 1, when the computational cell is full of water, and 0 if the cell is filled with air. An algebraic solution algorithm called Multidimensional Universal Limiter with Explicit Solution (MULES) applies the VOF method in OpenFOAM and it limits the fluxes of the discretized divergence term to ensure the indicator phase value remains between 0 and 1 during the entire calculation process. [3]

had revised the original VOF formulation in OpenFOAM for capturing the free surface interface in porous media and is given by:

$$\frac{\partial \alpha}{\partial t} + \frac{\partial}{\partial x_i}\left(\frac{1}{n}\langle \overline{u_i}\rangle \alpha\right) + \frac{\partial}{\partial x_i}\left(\frac{1}{n}\langle \overline{u_i}^r\rangle \alpha(1-\alpha)\right) = 0, \tag{7}$$

where α is 1 for water and 0 for air. $\langle \overline{u_i}^r\rangle = \langle \overline{u_i}^f\rangle - \langle \overline{u_i}^a\rangle$ is a relative velocity between the fluid and the air as described in [6] The last term in the equation indicates the compression term and addresses the compression of the interface between fluid and air. This term can be neglected when the cell is filled with water (α= 1) and or (α = 0), and is only active in the interface region between air and water. The porosity term is included to ensure that a given volume is filled/emptied faster when a solid grain assumes a part of the volume.

3 Simulation Procedure

The numerical simulation was conducted using the porousWaveFoam solver within the OpenFOAM framework. The flow was assumed to be laminar in all the cases considered in the study. The porous flow parameters used for simulating the fluid flow through the porous dam are given in the table The solution steps of the porousWaveFoam solver can be summarized as follows:

Table 1: Porous Flow Parameters

Porous Flow Parameter	Crushed Rock	Glass Beads
Linear friction Parameter(α)	50	90
Nonlinear friction Parameter(β)	0.45	0.15
Mean Grain Size (d_{50})(m)	0.0159	0.003
Porosity(n)	0.49	0.039

- Geometry, mesh, control options, and initial conditions, as well as boundary conditions, are read
- Solve the volume fraction equation
- In the predictor step the pressure value(p) is predicted and based on this value, the momentum equation is solved to obtain the velocity field (U).
- Face fluxes are calculated and are adjusted
- The pressure-Poisson equation is discretized based on the updated velocity field
- The produced linear system of pressure (p) is solved.
- The velocity field is then updated based on the new pressure field.
- The same procedure is repeated for incremented time steps.

3.1 Geometry and Mesh

The two-dimensional case study of a dam break with a porous media comprising crushed rocks and glass beads conducted by [1] as illustrated in [2] was simulated in this study. The water tank used in the experiments is 89.2 cm long, 44cm wide, and 58 cm high. A porous dam that is 29 cm long, 44 cm wide, and 58 cm high was placed at the center of the tank. A water column of an initial height of 25cm was placed on the left-hand side of the tank up to a width of 28cm. The water column is separated from the porous media by a sluice gate of a thickness of 2cm. A 2.5cm water layer is provided across the entire bottom of the tank. Two porous materials were used in the experiments. One is the crushed rocks with a mean grain size of 1.59 cm and porosity of 0.49. Another porous material is the uniform glass beads with a diameter of 0.3 cm and a porosity of 0.39. The experimental setup used by [1] for the porous dam breaking case is shown in figure 1 The computational mesh used in the study is a structured one with uniform spatial grid distribution

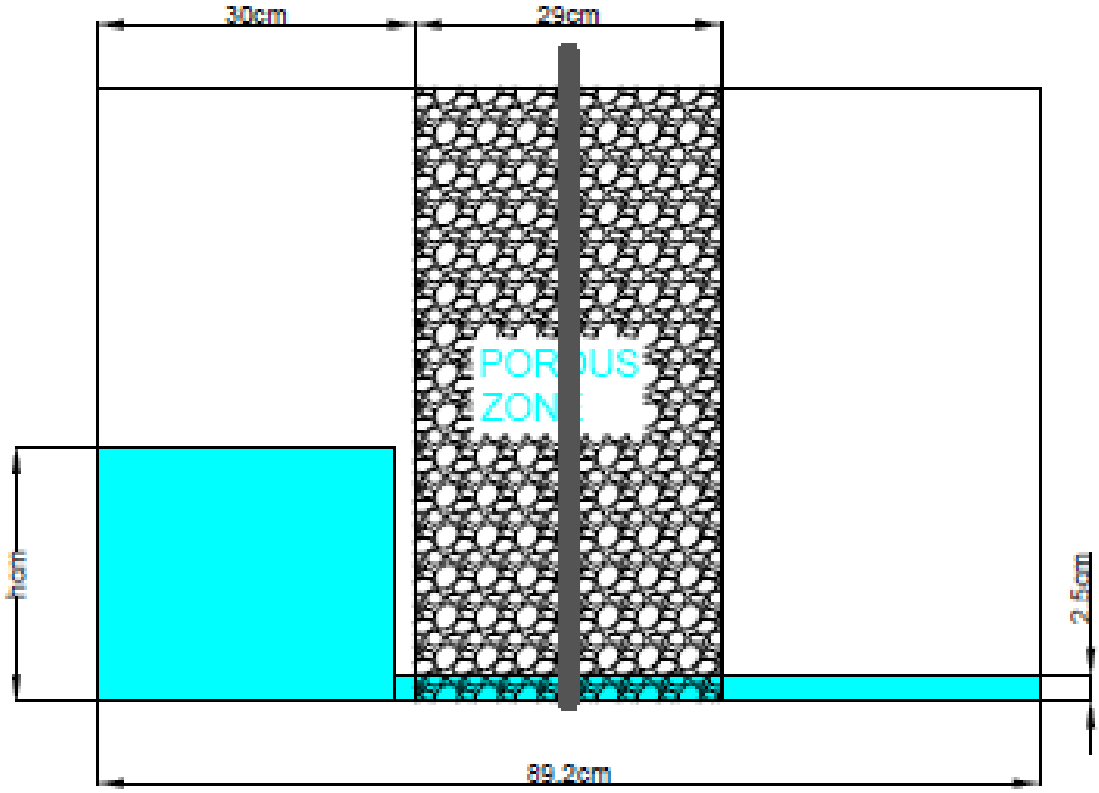

Figure 1: Experimental Setup of Porous Dam Breaking Case

and is illustrated in figure 2. The uniform grid size of $\Delta x = \Delta y$ = 0.5cm was chosen based on the grid Independence study and was adopted in the simulations.

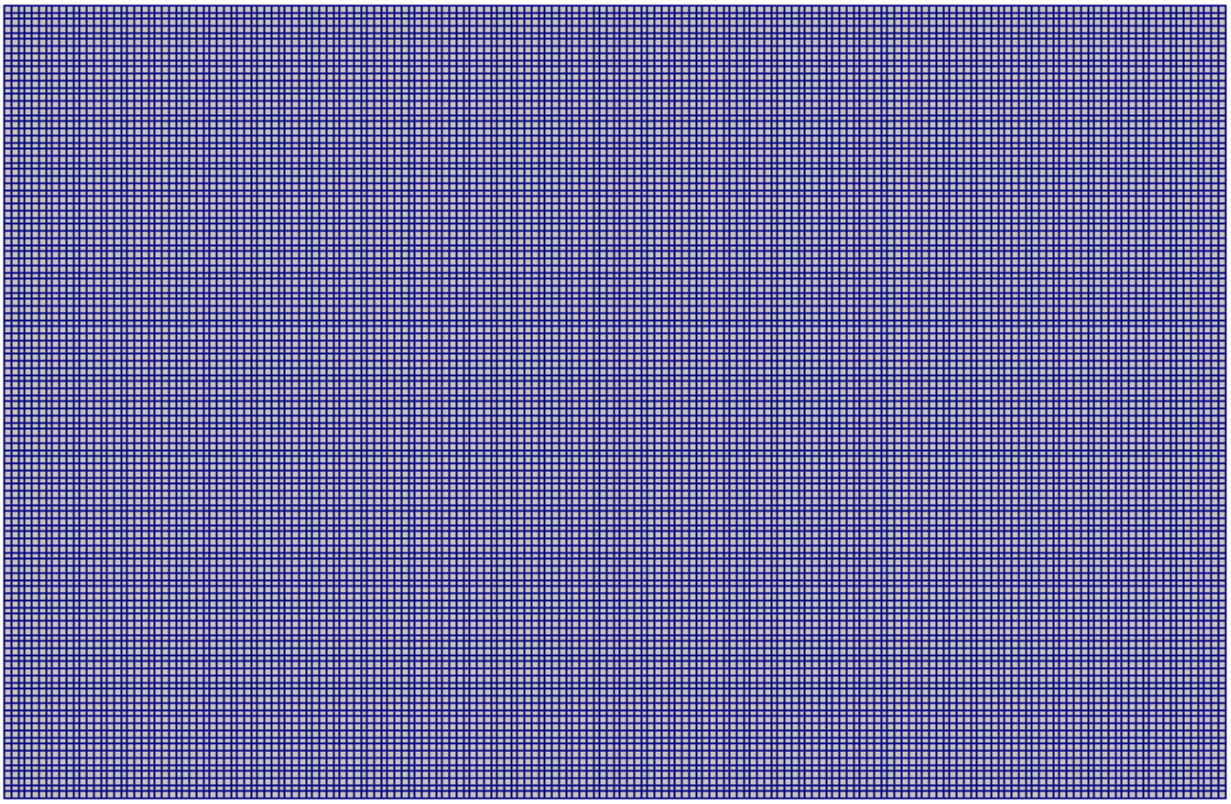

Figure 2: Computational Mesh used for the Numerical Simulation

3.2 Initial and Boundary Conditions

In the numerical simulation, to simulate the behaviour of the fluid interaction with the porous media, appropriate boundary conditions were applied. The tank has five boundaries which are the left wall, right wall, lower wall, atmosphere, and frontAndBack. As all the cases considered in the study were assumed to be two-dimensional, both front and back boundaries were considered "empty". At the upper boundary, an atmospheric boundary condition is used, which implies that the air and the water are allowed to flow out, whereas only air is allowed to flow in. The boundary conditions adopted in the study are given in table 2.

Table 2: Boundary Conditions

Boundary	alpha	Pressure	Velocity
leftWall	zeroGradient	zeroGradient	fixedValue
rightWall	zeroGradient	zeroGradient	fixedValue
lowerWall	zeroGradient	zeroGradient	fixedValue
atmosphere	inletOutlet	totalPressure	pressureInletOutletVelocity

3.3 Solver

To simulate the fluid flow within the porous media, [3] developed the porousWaveFoam solver within the waves2Foam toolbox [4] to solve VARANS equations using finite volume discretization. The solver adopts an iterative solution strategy for computing the velocity and pressure using the pressure-linked equations PIMPLE algorithm by solving the continuity equation (equation(1)), the momentum conservation equation (equation(2)) and the Poisson equation for pressure given by :

$$\nabla^2 P = f(u, \nabla P) \tag{8}$$

The PIMPLE algorithm is a combination of the PISO (pressure implicit with the splitting of operators) algorithm and the SIMPLE (semi-implicit method for pressure linked equations) algorithm. PISO uses one predictor step and two corrector steps to ensure mass conservation, providing a stable and quick calculation of velocity and pressure, while SIMPLE ensures convergence within a one-time step using a relaxation factor.PIMPLE attempts to predict velocity explicitly based on previous pressure calculations, while also ensuring convergence with the relaxation factor, resulting in quick and stable solutions. The spatial discretizations used in the solver are second-order accurate. Gauss linear, Gauss limited linear, Gauss vanLeer, and Gauss linear corrected schemes were used for the gradient, divergence, and Laplacian terms. A first-order bounded implicit Euler scheme is used for the temporal discretization with a maximum Courant number of 0.5. The pressure and velocity equations were solved using a Preconditioned Conjugate Gradient (PCG) solver and Preconditioned Bi-stab Conjugate Gradient (PBiCG) solver respectively. The tolerance of each iteration for pressure and velocity was set to be less than 10^{-7} and 10^{-6} respectively.

4 Results and Discussions

The present study investigates the flow through the porous media equations using the finite volume-based open-source software OpenFOAM. The performance of the porousWaveFoam solver was investigated based on the comparisons of the numerical results from the simulation with the available experimental data in the literature. The model is found to be accurate in predicting the fluid flow through the porous structures and in evaluating the free surface profiles

4.1 Grid Convergence Study

A grid-independence study was conducted to ensure convergence of the solution. The grid has been generated using the blockMesh utility in the OpenFOAM software. The structured uniform hexahedral mesh was generated in the entire computational domain. A thorough grid independence study was conducted for different grid sizes. Three different grid sizes with increasing degree of fineness were considered in the entire domain and the details for the different meshes are given in table 3 The variation of the free surface profiles along the length of the computational domain for the various mesh sizes is shown in figure 3. There is no significant change in the value of free surface elevation for the different mesh sizes considered. Hence a grid size of $\Delta x = \Delta y$ = 0.5cm was adopted for all the simulations considered in this study.

Table 3: Details of the Grid Convergence Study

Grid	Δx (m)	Δy (m)	Number of Grid Points
Grid(G1)	0.01	0.01	10620
Grid (G2)	0.005	0.005	41886
Grid(G3)	0.0025	0.0025	166362

4.2 Time Independent Study

To ensure numerical stability and accuracy of the temporal terms, a time-independence study was conducted. The time step was chosen in such a way that the maximum Courant number was kept below 0.5 for all the simulations. Three different time steps (Δt= 0.01s, 0.001s, 1×10^{-5}s) were considered in the study and the corresponding free surface profiles were obtained and are shown in figure 4. For better computational efficiency and time saving a time step of 0.001s was adopted in all the simulations conducted.

4.3 Model Verification Studies

The efficiency and accuracy of the porousWaveFoam solver in solving coastal problems were assessed by comparing the numerical results with laboratory experimental results of the flow passing through a porous dam conducted by [1] as illustrated in [2]. Two different porous materials like crushed rock and small glass beads were used for the simulation of the problem. An initial height of 25cm for the water column was considered in the study. The surface elevation data was provided in the experiments conducted with high spatial resolution inside and outside the porous medium, making them useful for the evaluation of the solver performance in both clear fluid region and porous media.

4.3.1 Dam Break Case with Crushed Rock as Porous Media

The flow passing through the dam with crushed rocks as porous was investigated. The free surface profiles across the inside and outside the porous media during the period of fluid flow through the porous dam were plotted and compared with the experimental results and are shown in the figure **??**. Small discrepancies were noted in the surface elevation values between the experimental and numerical results even though the trend was satisfactorily captured. The flow was considered to be laminar in the study. However, owing to the large velocity of flow and large pore sizes, in actual conditions, the flow was turbulent. The discrepancies in surface elevation values are attributed to the negligence of turbulence effects of the flow.

4.3.2 Dam Break Case with Small Glass Beads as Porous Media

Simulations were conducted to study the fluid flow through a porous dam made of small spherical glass beads. A laminar flow was considered based on the assumptions made by [7] and [5]. The surface profiles obtained from the simulation at various time intervals are shown in figure **??** The comparisons of the free surface profiles between the numerical results and experimental data were done and are shown in figure**??**. A better agreement is obtained between the simulated results and the experimental data. Slight discrepancies are noted during the initial flow periods, which is

attributed to the time difference in gate opening. As reported by [1], the 2 cm gap between the porous dam and the gate caused a piling up of water resulting in the formation of an upward jet on the surface of the porous dam, which was clearly visible in the experiment and was also predicted by the numerical simulation. Since the gate was opened manually within a finite duration in the experiment, a faster advancement of the water surface near the bottom inside the porous dam was noted.

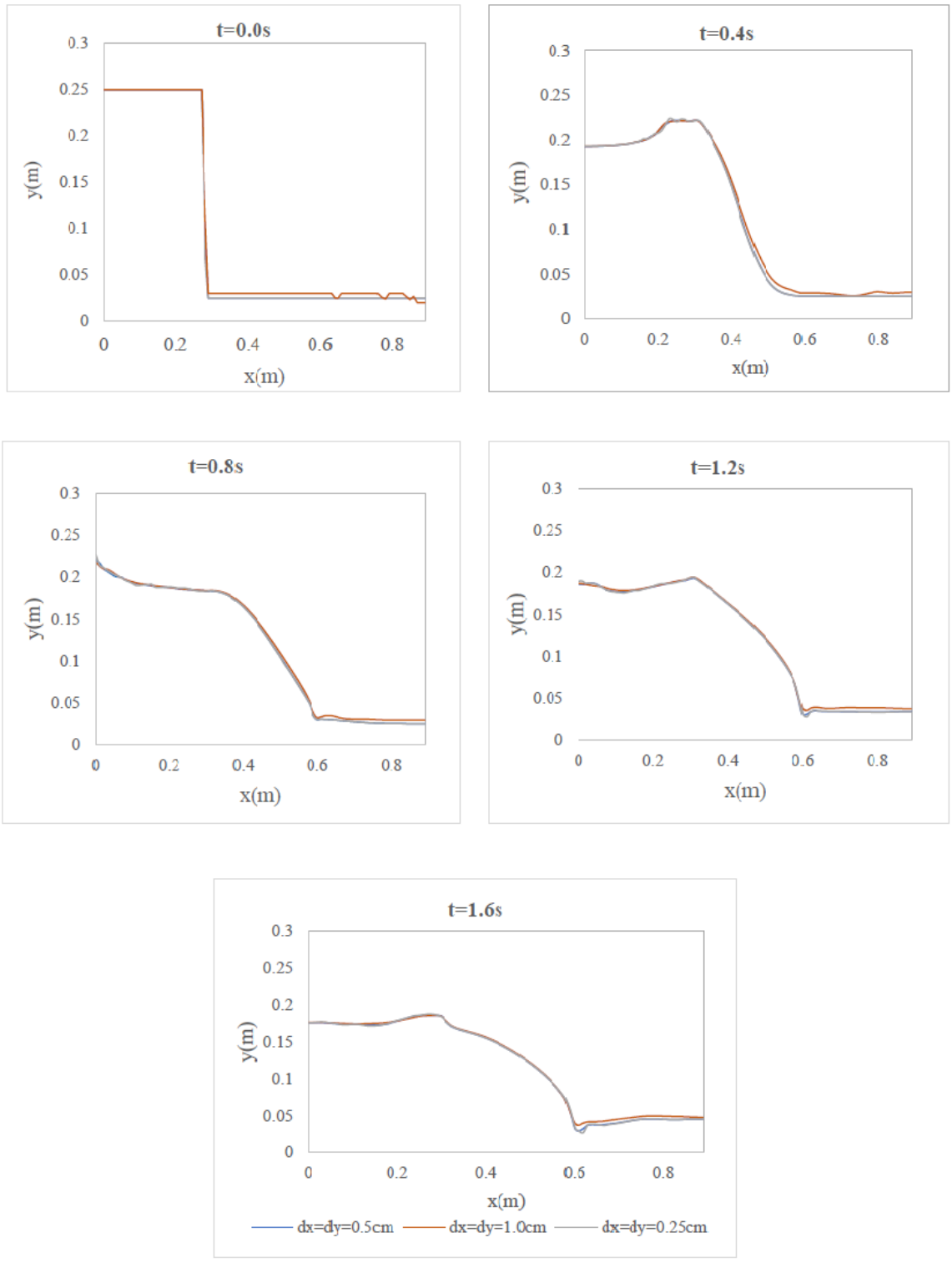

Figure 3: Comparison of Free Surface Profiles for Different Mesh Sizes

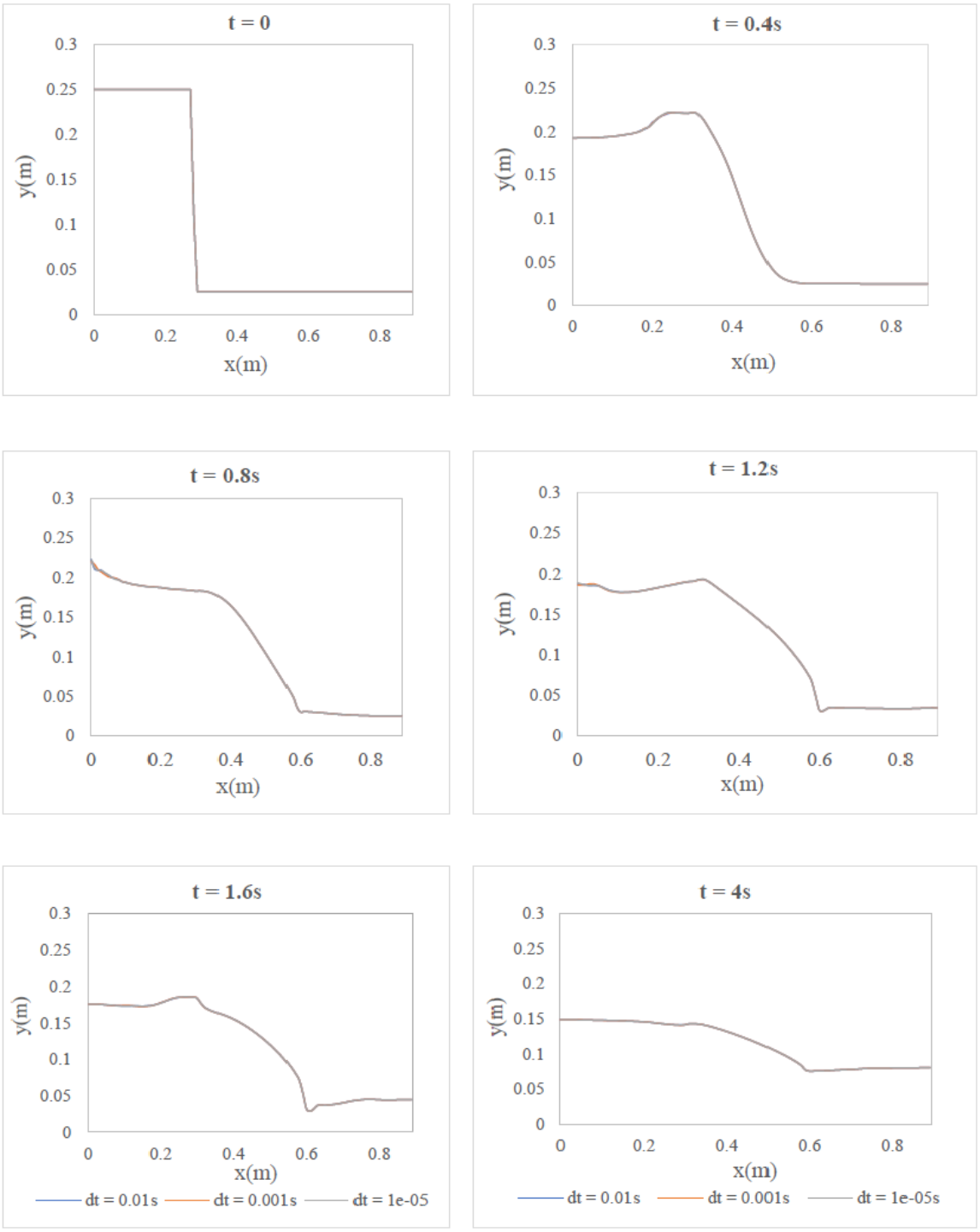

Figure 4: Comparison of Free Surface Profiles for Different Time Steps

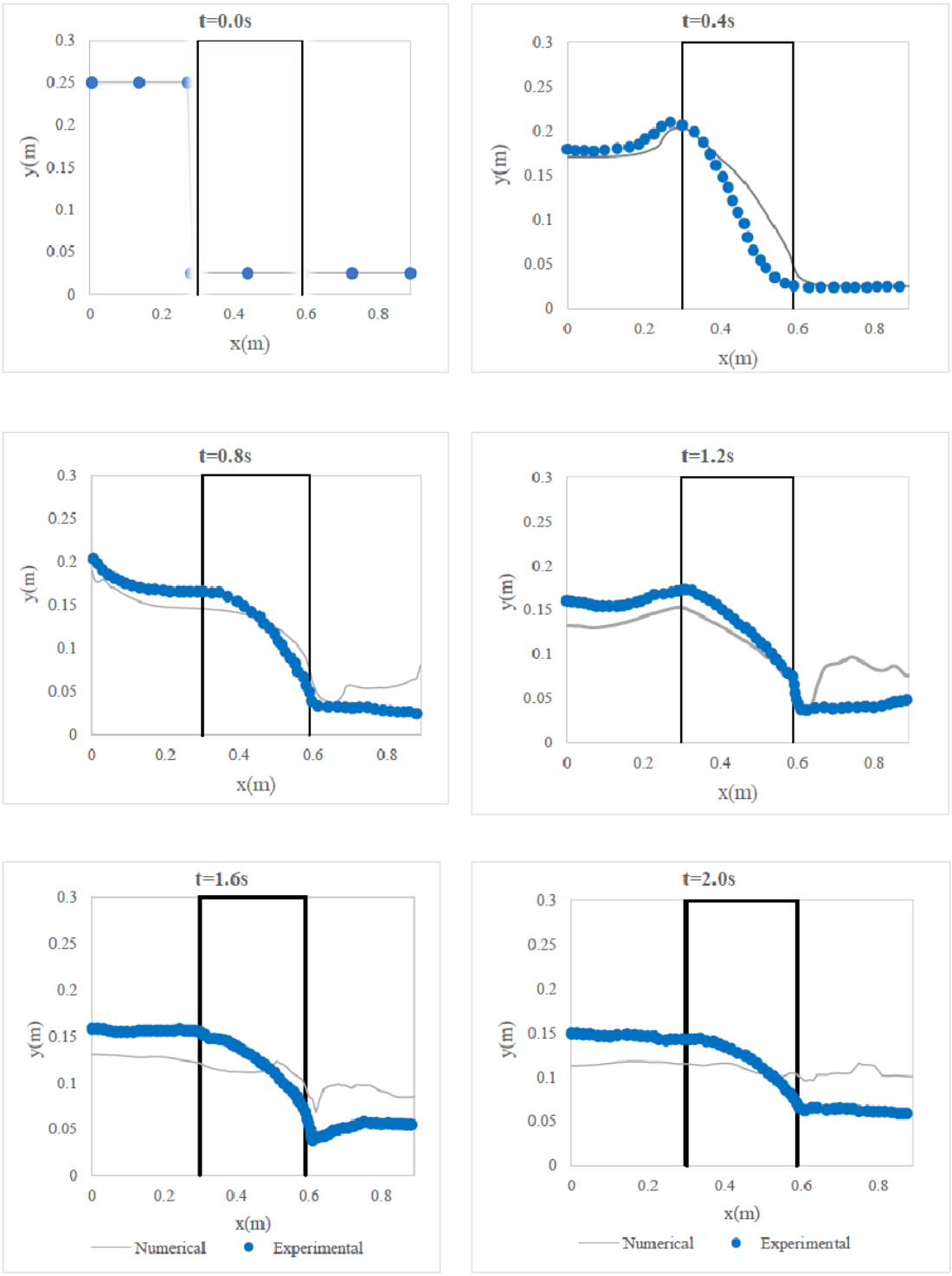

Figure 5: Free Surface Profiles for Flow Passing through Porous Dam with Crushed Rock as Porous Media

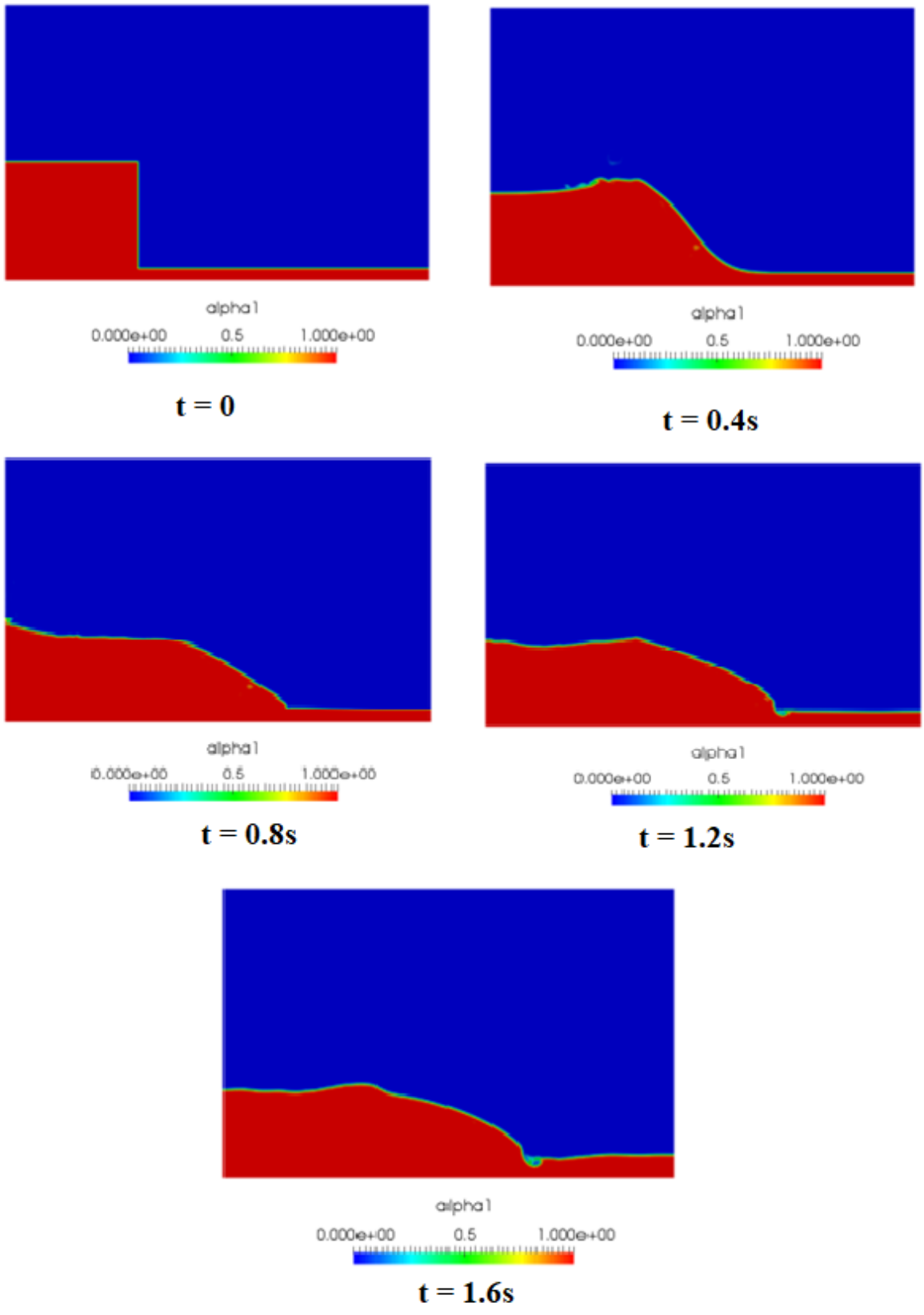

Figure 6: Surface profiles of Fluid Flow through Porous Media at Various Time Intervals

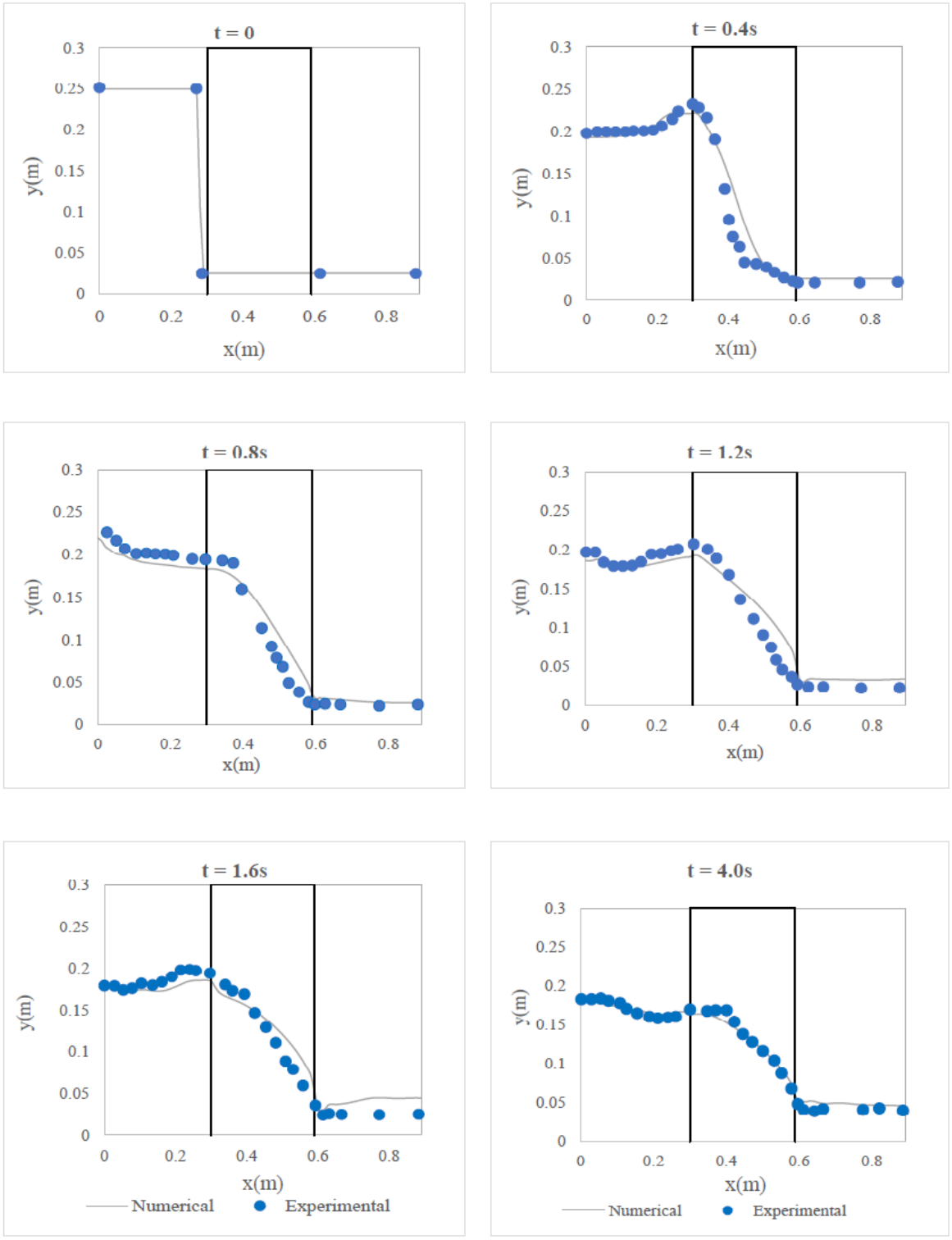

Figure 7: Free Surface Profiles for Flow Passing through Porous Dam with Small Spherical Glass Beads as Porous Media

5 Conclusions

The present study investigates the flow through the porous media using the finite volume-based open-source software OpenFOAM. The performance of the porousWaveFoam solver was analyzed. based on the comparisons of the numerical results from the simulation with the available experimental data in the literature. The model is found to be accurate in predicting the fluid flow through the porous structures and in evaluating the free surface profiles

6 Acknowledgement

I would like to express my deepest gratitude to my mentor Dr Chandan Bose for the constant support, encouragement, and guidance throughout the completion of my work. His experience and guidance made it possible to complete the job. He always spared time for our long discussions without hesitation and never refrained from sharing his deep understanding of OpenFOAM with me. His guidance always made my path clear and confident. I place on record my sincere thanks to Prof.Janani Srree Murallidharan and Prof. Manaswitha Bose for the valuable suggestions rendered during this internship. I would like to extend my gratitude to Mr Divyesh Variya, Assistant Project Manager, and Ms Payel Mukherjee, Project Manager CFD-OpenFOAM Team FOSSEE for the support given to me during the internship period. I would like to record my sincere thanks to my fellow intern Mr Tinto Thomas for the fruitful discussions during the internship period. I would like to extend my thanks to my friends at IITM Mr Vijith P P, and Mr Shamjad Aboobacker for clarifying my doubts about CFD during the screening task and the course of the internship. I would like to thank my family for their support during the work. Last but not least I would like to record my gratitude to the entire CFD-OpenFOAM Team FOSSEE for giving me this wonderful opportunity.

References

[1] P. L.-F. Liu, P. Lin, K.-A. Chang, and T. Sakakiyama, "Numerical modeling of wave interaction with porous structures," *Journal of Waterway, Port, Coastal and Ocean Engineering*, vol. 125, no. 6, pp. 322–330, 1999.

[2] E. Jafari, M. M. Namin, and P. Badiei, "Numerical simulation of wave interaction with porous structures," *Applied Ocean Research*, vol. 108, p. 102522, 2021.

[3] B. Jensen, N. G. Jacobsen, and E. D. Christensen, "Investigations on the porous media equations and resistance coefficients for coastal structures," *Coastal Engineering*, vol. 84, pp. 56–72, 2014.

[4] N. G. Jacobsen, D. R. Fuhrman, and J. Fredsøe, "A wave generation toolbox for the open-source cfd library: Openfoam®," *International Journal for numerical methods in fluids*, vol. 70, no. 9, pp. 1073–1088, 2012.

[5] M. Van Gent, "Porous flow through rubble-mound material," *Journal of waterway, port, coastal, and ocean engineering*, vol. 121, no. 3, pp. 176–181, 1995.

[6] E. Berberović, N. P. van Hinsberg, S. Jakirlić, I. V. Roisman, and C. Tropea, "Drop impact onto a liquid layer of finite thickness: Dynamics of the cavity evolution," *Physical Review E*, vol. 79, no. 3, pp. 036 306–1 –036 306–15, 2009.

[7] H. Burcharth and O. Andersen, "On the one-dimensional steady and unsteady porous flow equations," *Coastal engineering*, vol. 24, no. 3-4, pp. 233–257, 1995.

DISCLAIMER: This project reproduces the results from an existing work, which has been acknowledged in the report. Any query related to the original work should not be directed to the contributor of this project.