Experimental and Numerical Study on the Reduction of Tsunami Flow Using Multiple Flexible Pipes

T. Tar[1], N. Kato[1], H. Suzuki[1], T. Okubayashi[1]
[1]Department of Naval Architecture and Ocean Engineering, Osaka University

thaw@naoe.eng.osaka-u.ac.jp

Background

Large-Scale Earthquake and Tsunami

- Oil and gas tanks in petroleum complexes are vulnerable to the attacks of large-scale earthquakes and tsunamis.
- During 2011 Great East Earthquake, a tsunami occurred and it attacked the oil and gas tanks located in several areas, especially Kesennuma City, Miyagi Prefecture.
- This led to severe oil spills and consequently, large scale fires occurred in the city area, threatening households and human lives.
- Such large scale disasters will be estimated in Osaka Bay, Tokyo Bay or Mikawa Bay if Tokai-Tonankai-Nankai earthquake occurs. Therefore, it is very urgent to establish the countermeasures against such large-scale disasters.

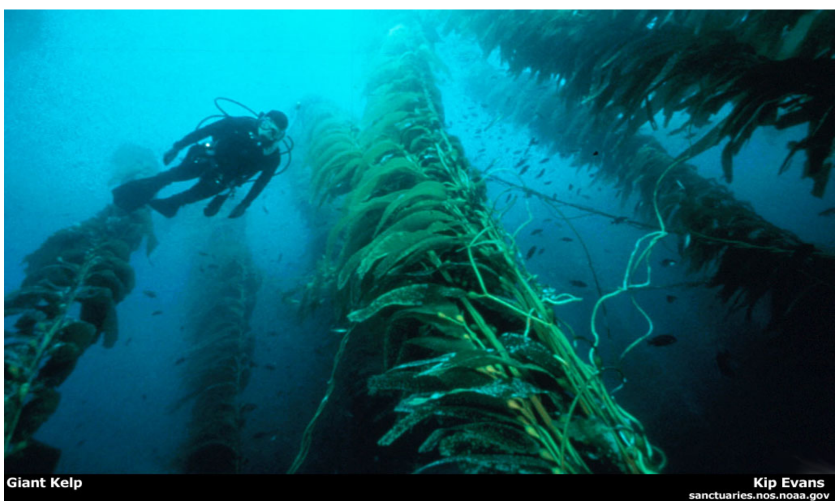
Oil tanks destroyed after tsunami, 2011 Great East Japan Earthquake

Large-scale fire in the city area, Kesennuma, Myagi Prefecture

Tsunami Damage Reduction

Oil industry in Sakai, Osaka

- **Mangrove** was found to be effective (Thailand, Indian Ocean Tsunami, 2004)
- The potential of **Giant Kelps** (Will not disturb the marine traffic)



Giant-kelps

http://www.glogster.com/juliachini/giant-kelp/g-6mb72vgddhfg1jsdef08pa0

Mangrove plants (2004, Thailand)

Sasaki, Y., Tanaka, N., Yutani, K., and Homchuen, S., (2005)

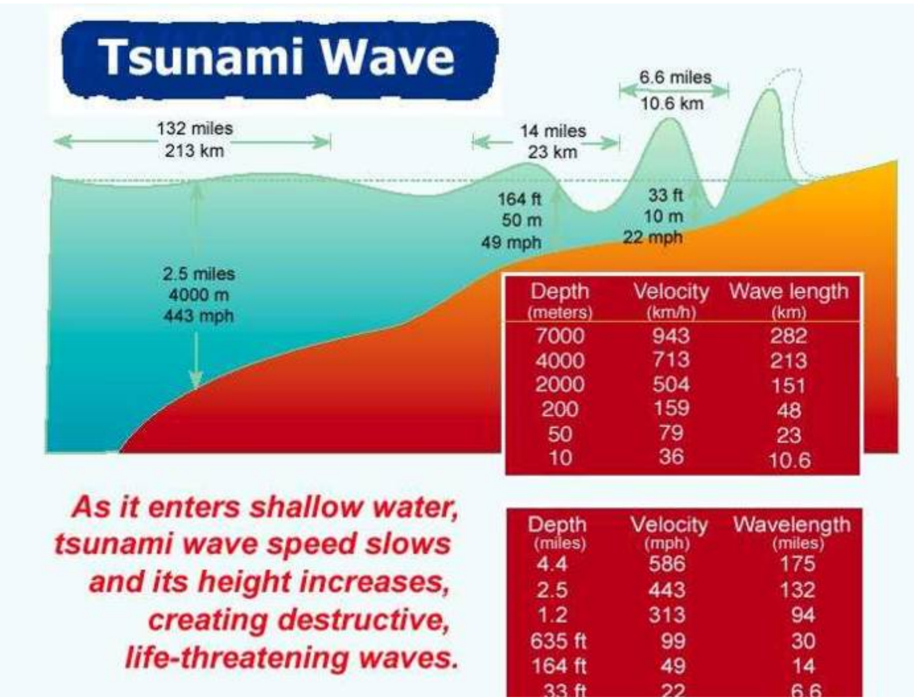
Flexible Pipes

- Wound up in the calm condition.
- Make them upright by injecting compressed air from air container or pump via a remotely operated valve.
- Such an arrangement can be used for tsunami damage protection of oil storage tanks together with other arrangements.

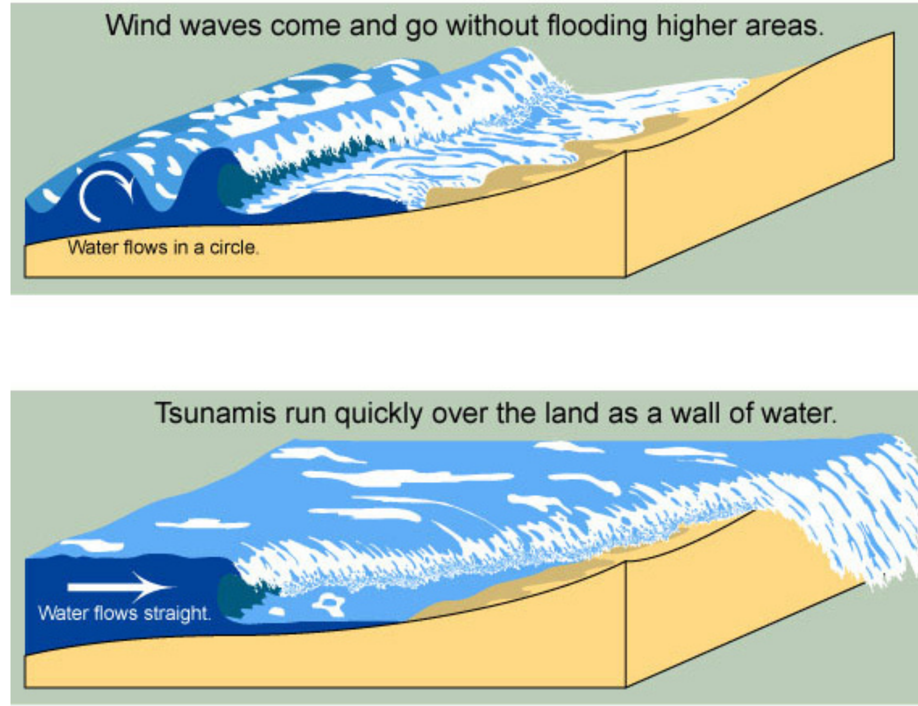
Objective

- To evaluate the amount of tsunami damage that can be reduced by the use of flexible pipes.
- To predict the phenomena of tsunami attack on the oil tanks with or without the flexible pipes arrangement.
- To establish a numerical model that can predict the tsunami damage acting on the oil and gas tanks which is reduced by the flexible pipes.

Representing the Tsunami Wave



Tsunami Wave Characteristics
(source: Maine Geology Surveys)



Ordinary ocean waves vs Tsunami Wave
(Source: Seismic Safety Commission, California)

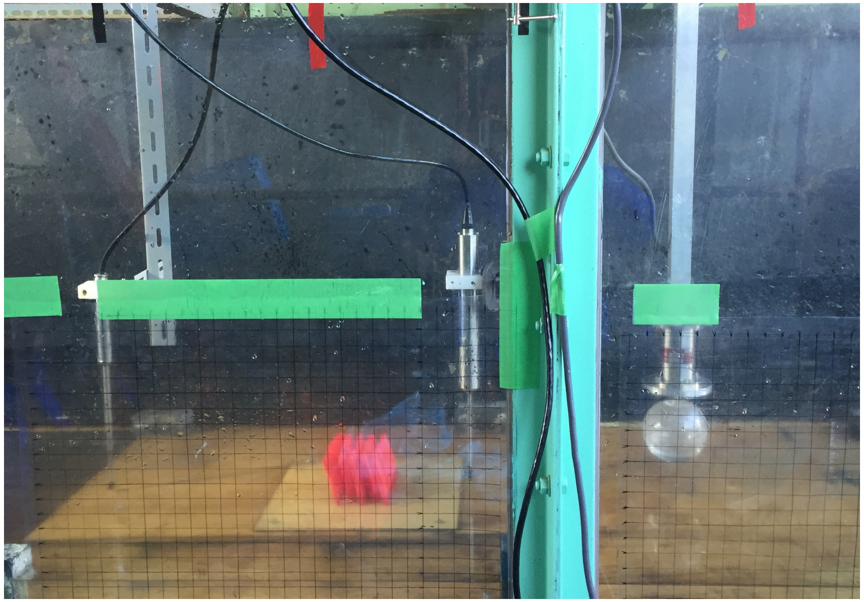
Tsunami is a shallow water wave with very long wave periods. Therefore, it can assumed as a uniform flow. However, due to its very long periods, it is very difficult to generate a scaled tsunami wave in the experiments.

The common methods to generate tsunami-like waves are :

1. Plunger type wave generators
2. Dam break method

In this study, tsunami-like wave is generated by using dam break method in the tsunami basin.

Approach

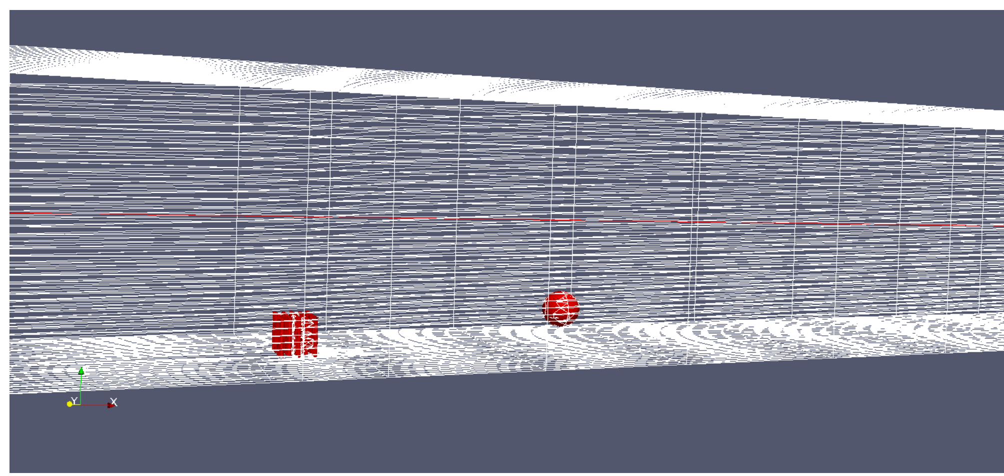


Scale Model Experiments

- Carried out in the tsunami basin.
- Tsunami was generated by dam break method.
- The flow velocity in front of and behind the pipes as well as hydrodynamic force acting on the sphere that represents oil tanks is measured.

Numerical Analysis

- Computational Fluid Dynamics (CFD)
- Numerical model representing the scale-model experiments is created and compared with the experiment results.
- Rigid pipes case is considered instead of flexible pipes.



Tsunami Basin Experiment

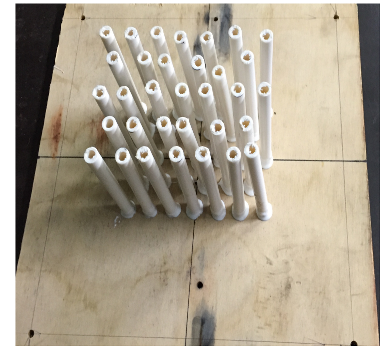
Tsunami Basin
L x W x D : 100m x 0.7m x 0.9m
Flow Velocity: around 1.2 m/sec

Full – scale Pipes

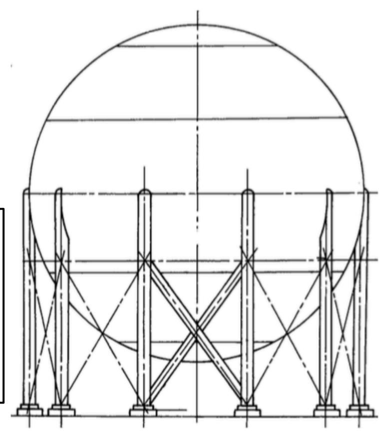
Length	15 m
Diameter	1.5 m
Bending Stiffness (EI)	201.7 MPa. m^4
Tsunami Velocity	12.5 m/s

$\lambda = 150$

Model – scale Pipes

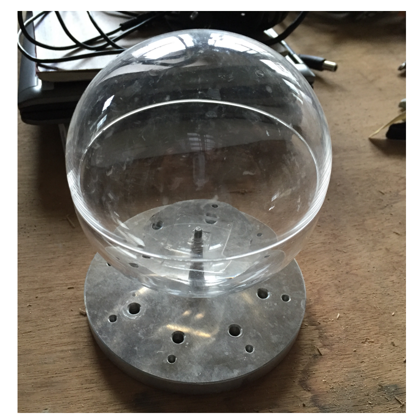


Length	10 cm
Diameter	1.0 cm
Bending Stiffness (EI)	59.76 Pa. m^4
Inflow Velocity	1s m/s



球形貯槽の主な諸元
球殻の外径： 15665 mm
球殻底部の地上高：2478 mm
呼称容量： 2000 m^3

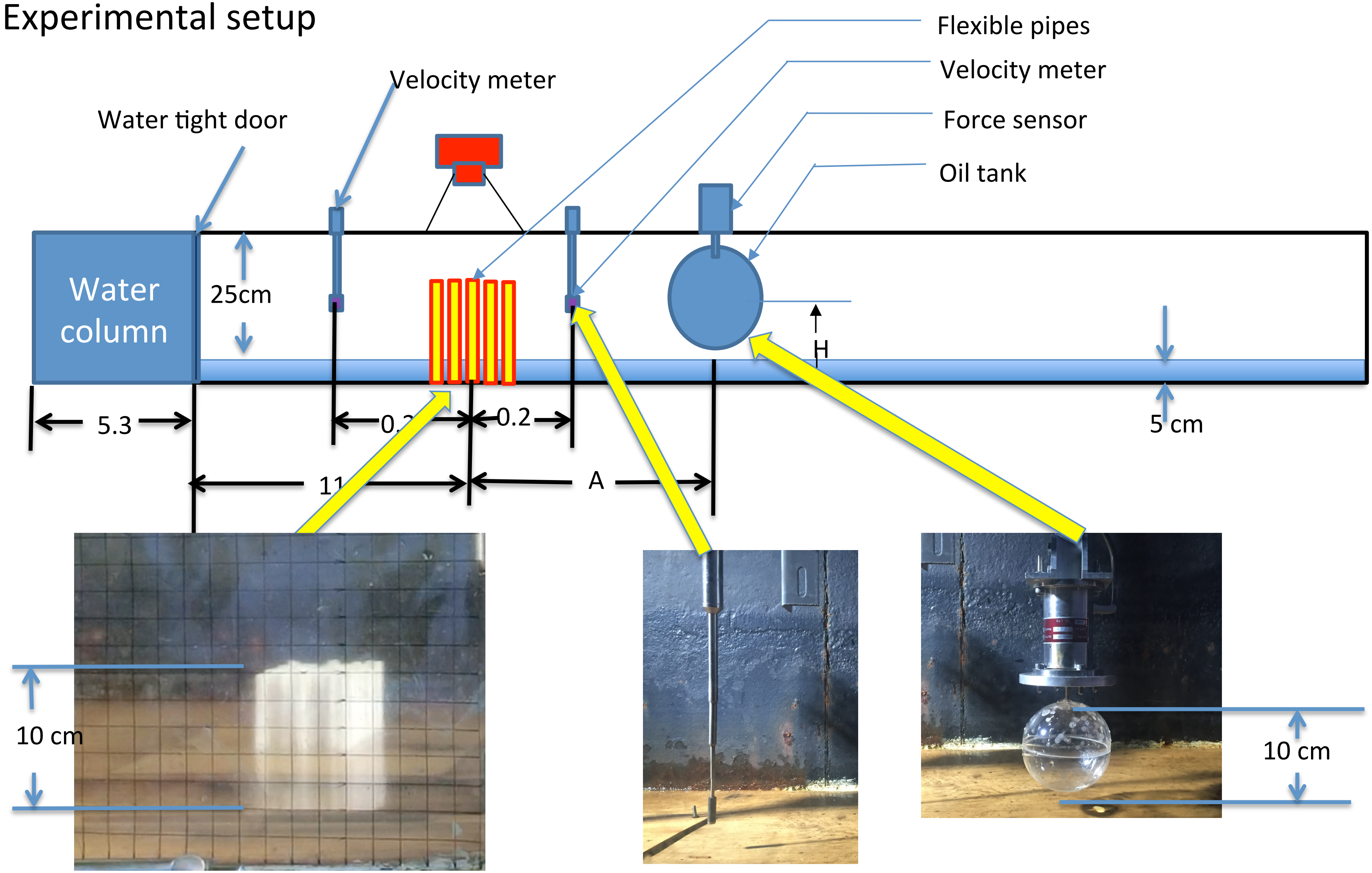
Full-cale spherical tank (15m dia.)



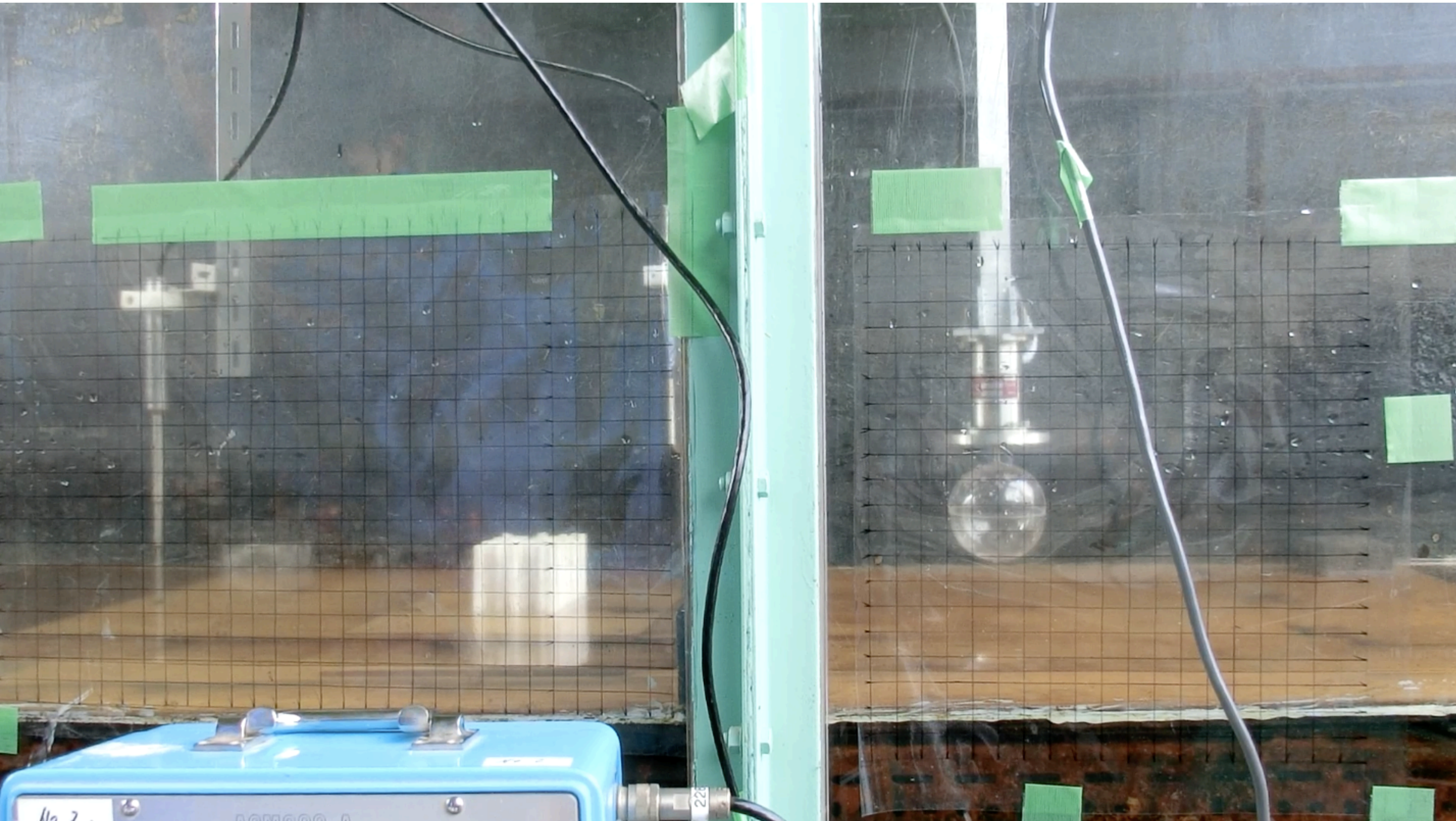
Scaled Model Sphere (10cm dia)

Tsunami Basin Experiment

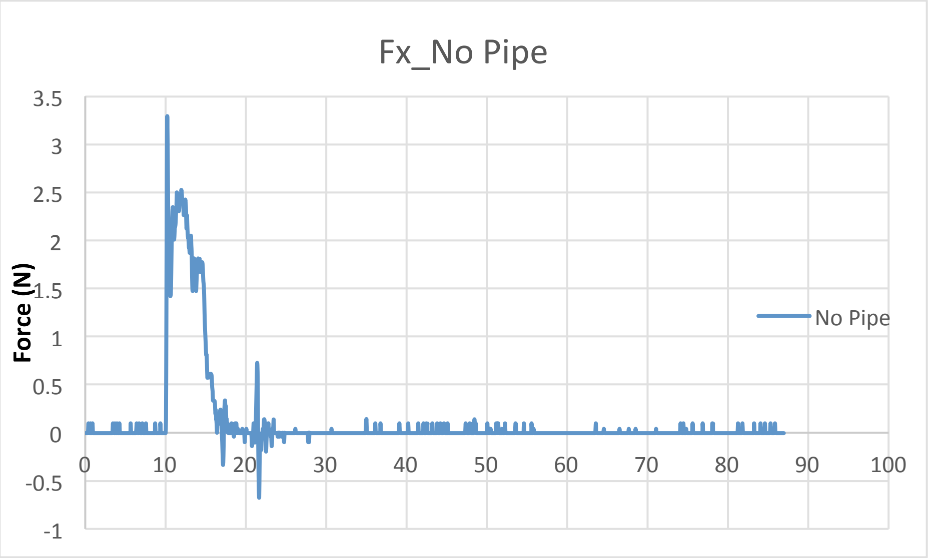
Experimental setup

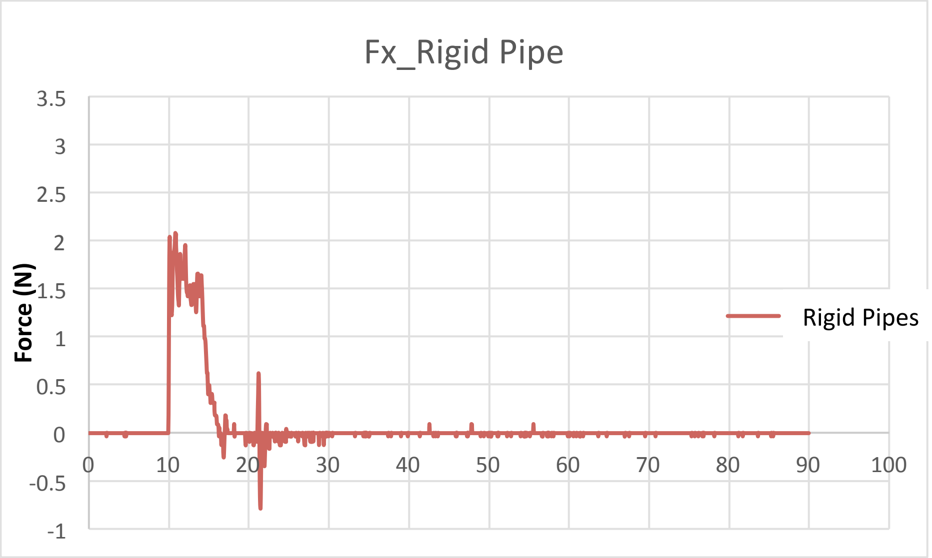


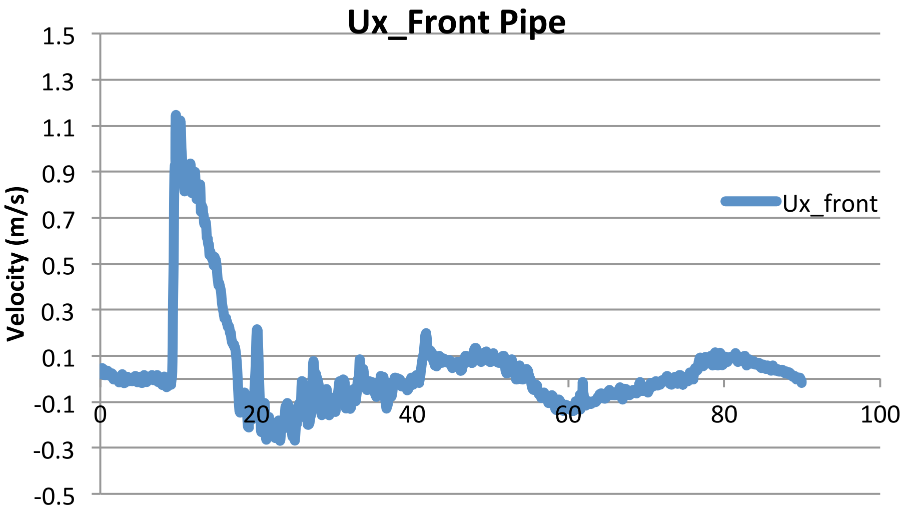
Tsunami Basin Experiment

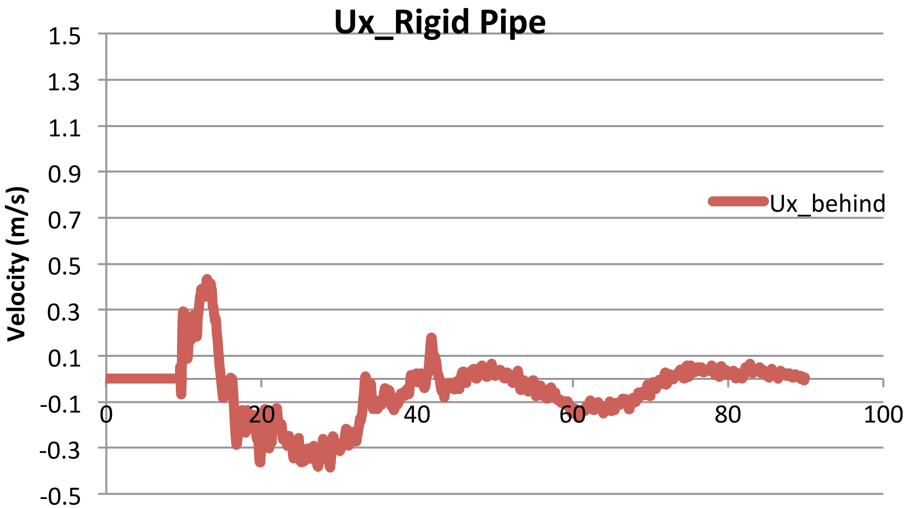


Tsunami Basin Experiment









3D Numerical Analysis of Tsunami Attack on an Array of Rigid Pipes

Computational Fluid Dynamics (CFD)

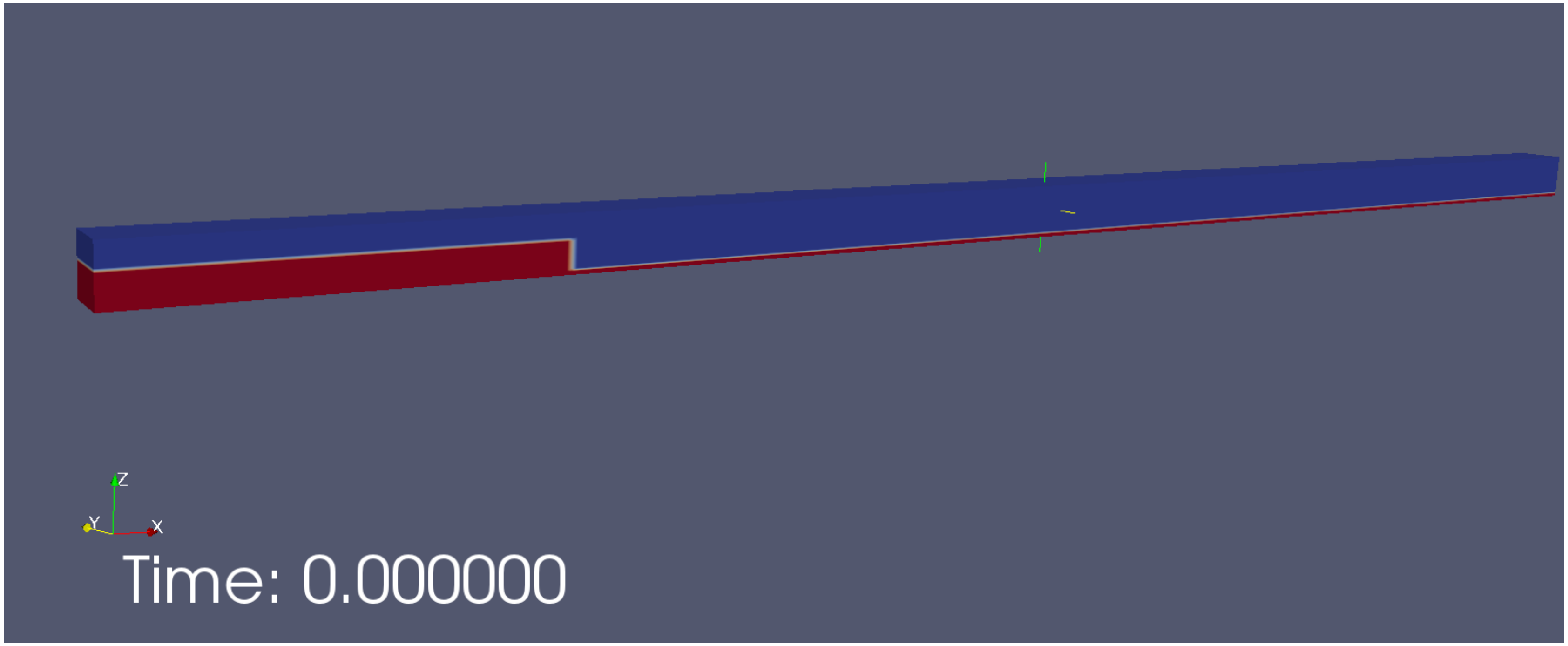
- CFD is used to to establish a numerical model of tsunami attack on the rigid pipes and to understand the effectiveness of such pipes.
- Dam break method is used to generate the tsunami-like wave in the numerical tsuanmi basin.
- Unsteady Reynolds' Averaged Navier-Stokes equations are numerically solved on a 3D computational domain for a transient free-surface flow.
- Current simulation involves 2 fluids : Water and Air.

OpenFOAM

- OpenFOAM is a free, open-source CFD software for computational fluid dynamics (CFD) developed primarily byCFD Direct.
- Finite volume method is used to discretize the governing equations over the 3D computational domain.
- Can solve steady state or transient flows problems for single phase and multiphase problems for a variety of fluid models (compressible, incompressible, Newtonian, Non-Newtonian, etc).
- OpenFOAM version 2.4.0 was used in this study.

3D Numerical Analysis of Tsunami Attack on an Array of Rigid Pipes

Simulation Domain



Dimensions of the domain are the same as the tsunami basin dimensions.

The reference frame used for the simulation domain is also shown.

Governing Equations

Continuity Equations

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0$$

Momentum Equations

$$\frac{\partial \rho \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \rho \bar{u}_i}{\partial x_j} - \frac{\partial}{\partial x_j}\left(\mu_{eff} \frac{\partial \bar{u}_i}{\partial x_j}\right) = -\frac{\partial p}{\partial x_i} - g_i x_j \frac{\partial \rho}{\partial x_j}$$

Where x_i is the Cartesian position vector with respect to the reference frame (x_1, x_2, x_3).

u_i is the velocity vector in the Cartesian coordinate system

p is the pressure

$\mu_{eff} = \mu + \rho \nu_t$ is the effective dynamic viscosity which can be separated into fluid viscosity μ *and* the turbulent kinematic viscosity ν_t

Turbulence Modeling

SST-k-Omega Turbulence model was used to for turbulence computation.

Flow Solver

InterFoam Solver

- Transient flow solver for modeling free-surface flows.
- Uses Volume of Fluid (VoF) Method for free-surface capturing.
- Pressure Implicit Separation of Operators (PISO) algorithm is used to calculate the velocity-pressure coupling.

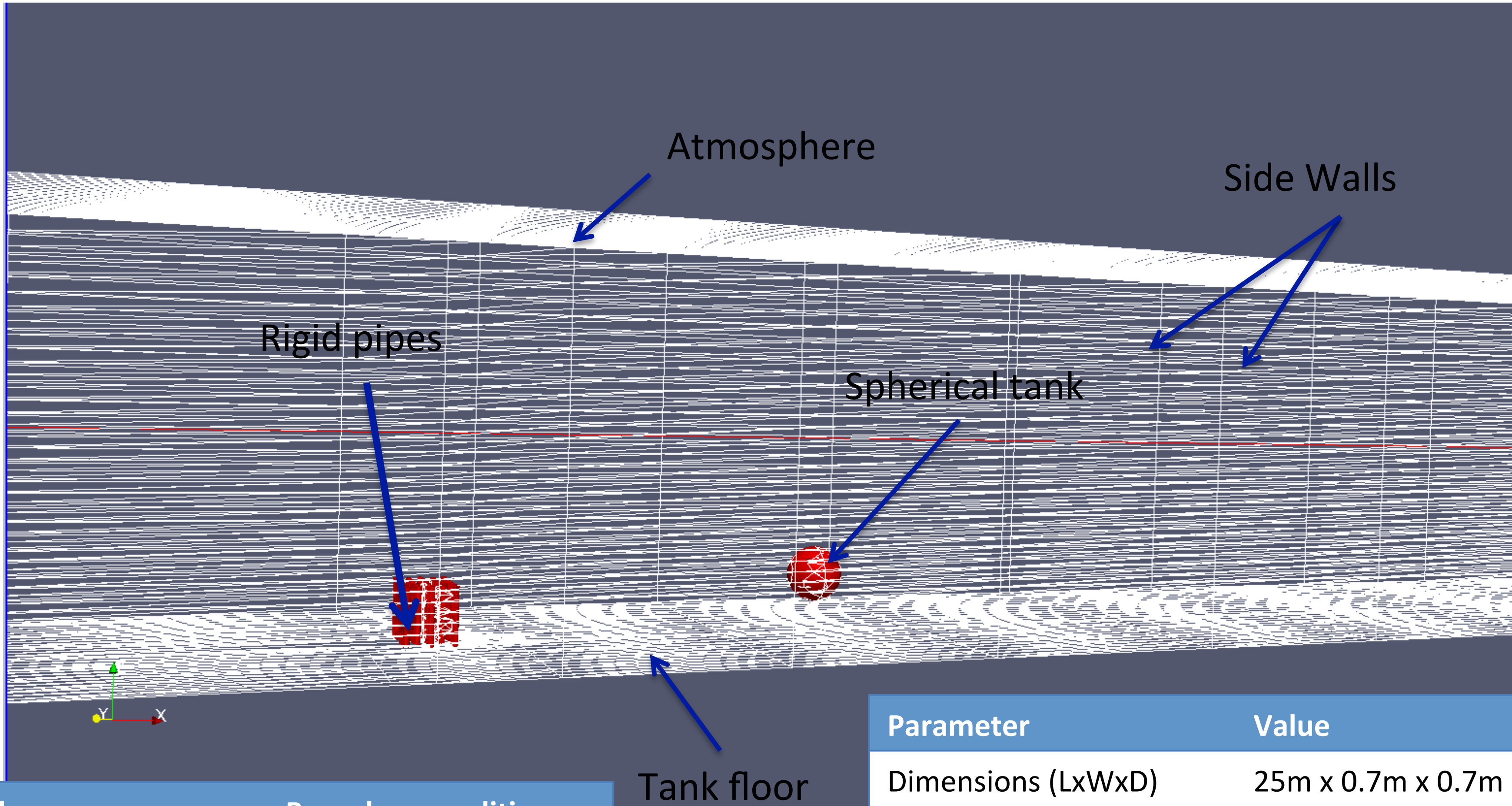
Volume of Fluid (VoF Method)

- VoF method considers only a single phase flow with a single phase function γ as the quantity of the phase of interest (for instance, water).
- For example, γ =1 means the cell is full of water and γ =0 means the cell is full of air.
- In interFoam flow solver, phase movement is described by the following equation:

$$\frac{\partial \gamma}{\partial t} + \frac{\partial \bar{u}_i \gamma}{\partial x_i} - \frac{\partial \bar{u}_{c,i} \gamma (1-\gamma)}{\partial x_i} = 0$$

where $\bar{u}_{c,i}$ is an artificial compression term to preserve a sharp interface between the phases.

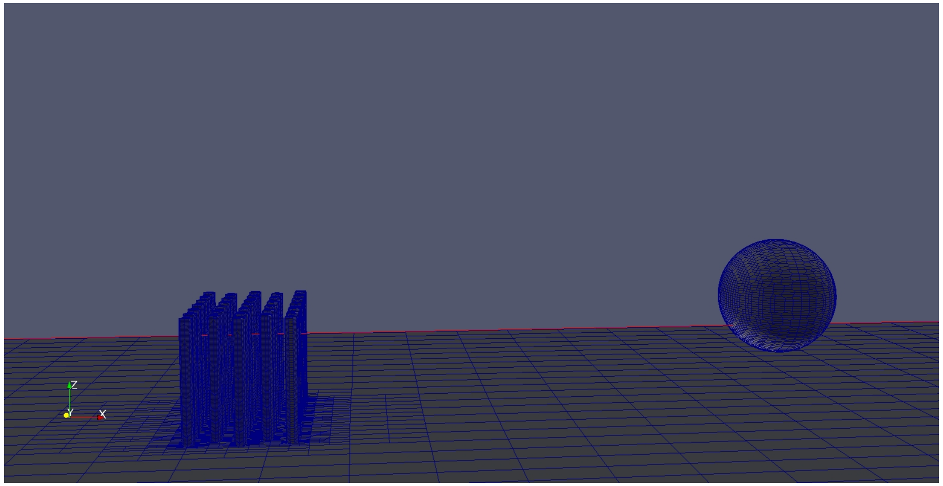
Computational Domain



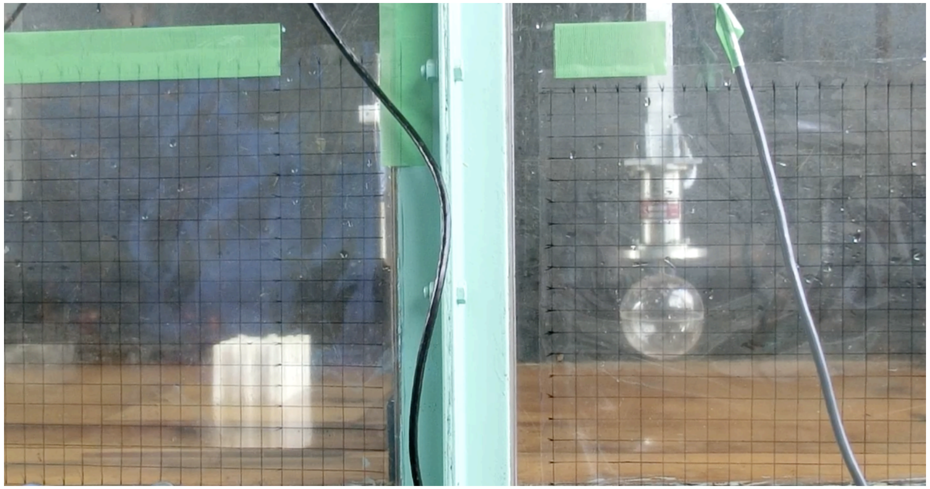
Patch	Boundary conditions
Side Walls	Slip wall
Atmosphere	Pressure inlet outlet
Tank floor	Non-slip wall
Pipes and Sphere	Non-slip wall
Ends	Slip wall

Parameter	Value
Dimensions (LxWxD)	25m x 0.7m x 0.7m
Fluid density (water)	998.2 kg/m^3
Kinematic Viscosity (water)	1.004E-06 m^2/s
Cell Count	Around 870,000 cells
Mesh Type	Unstructured Hexahedral

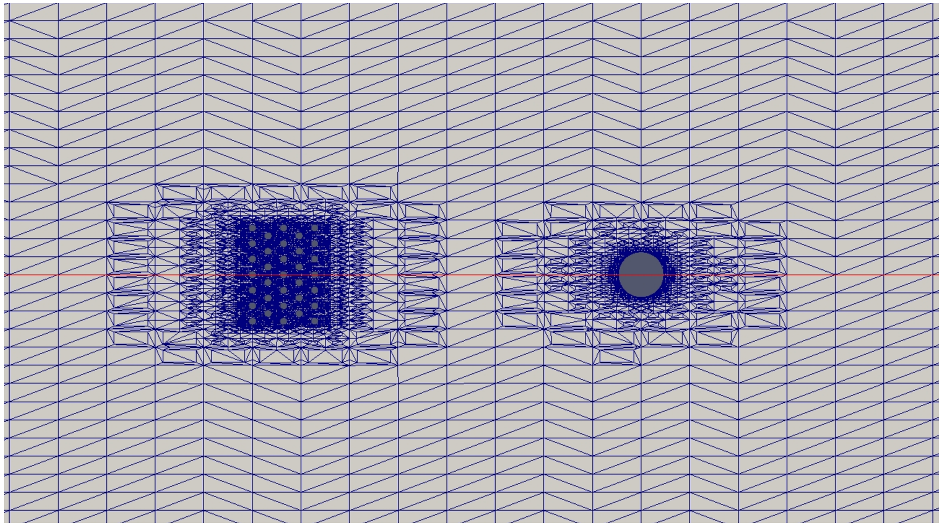
Computational Mesh



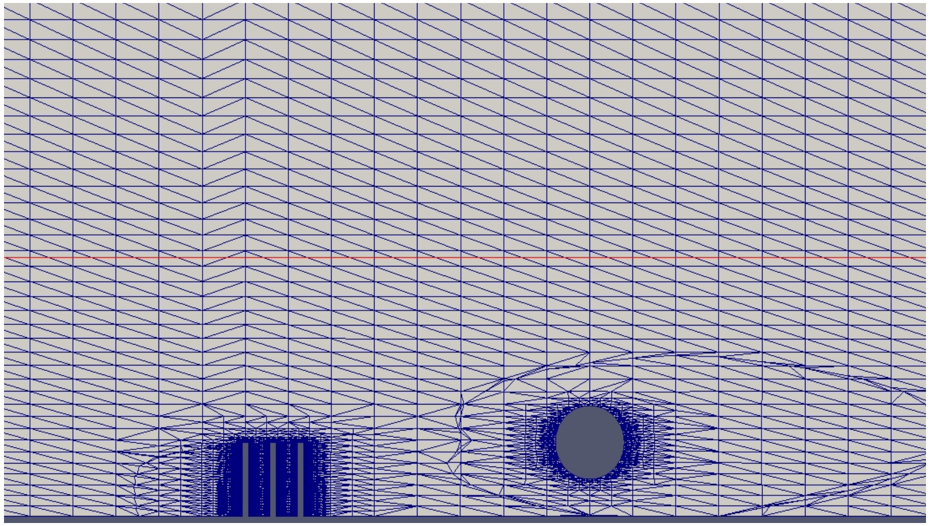
3D computational mesh



Experimental setup

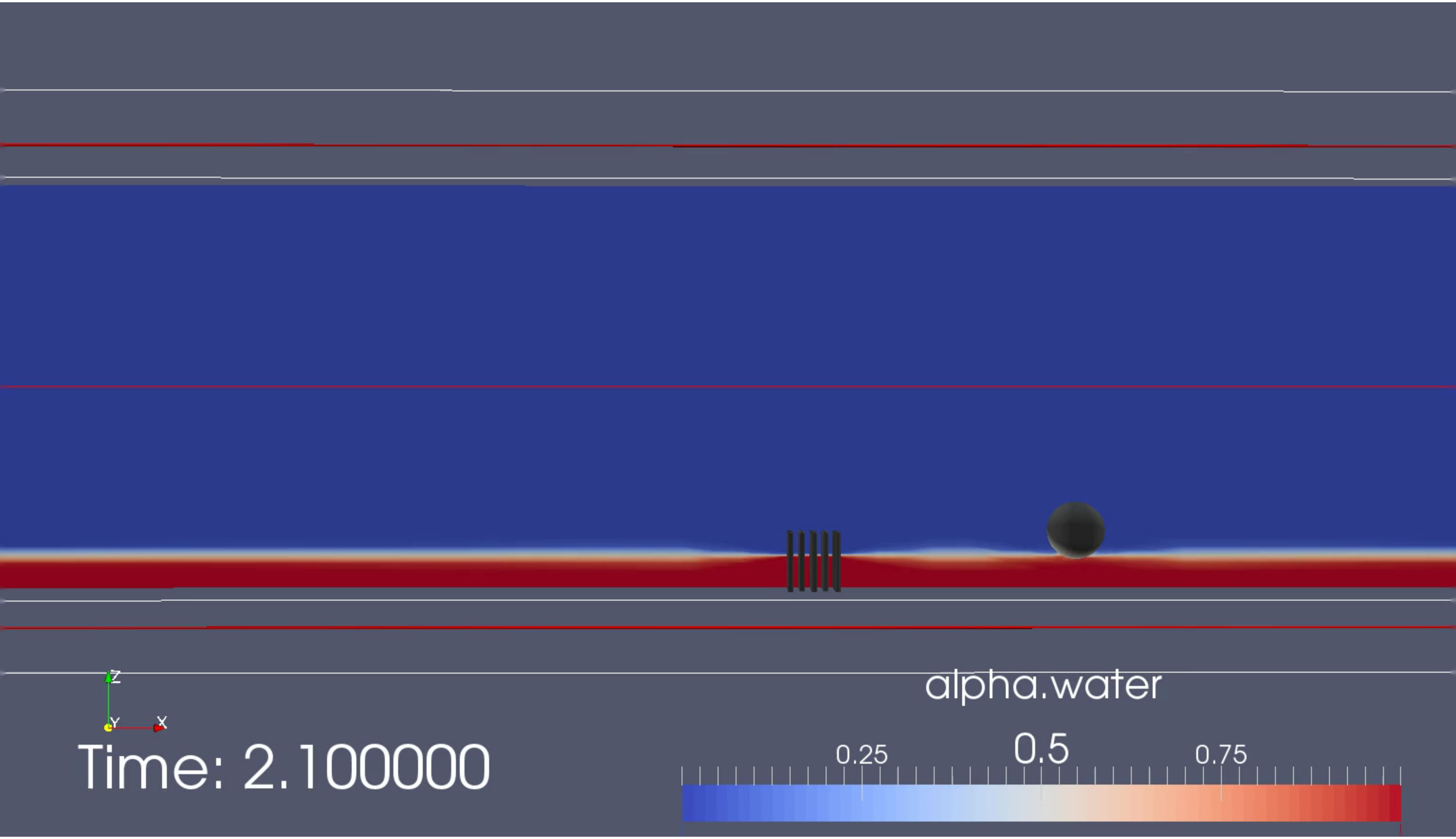


Computational mesh (top view)

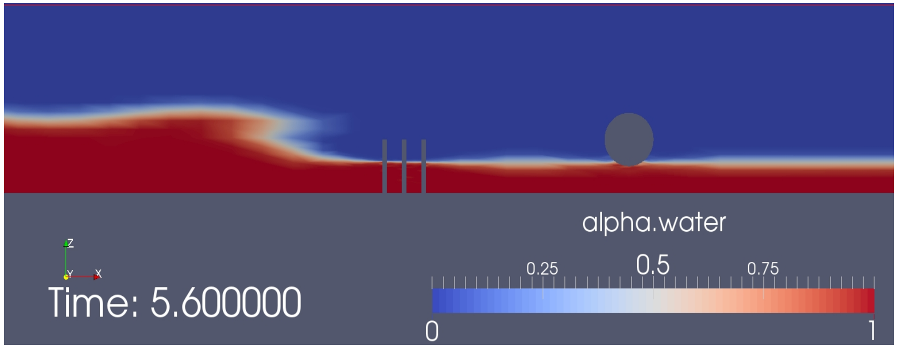


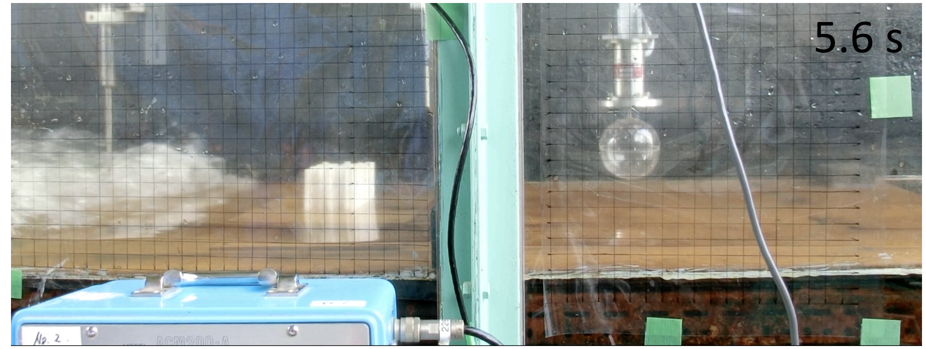
Computational mesh (side view)

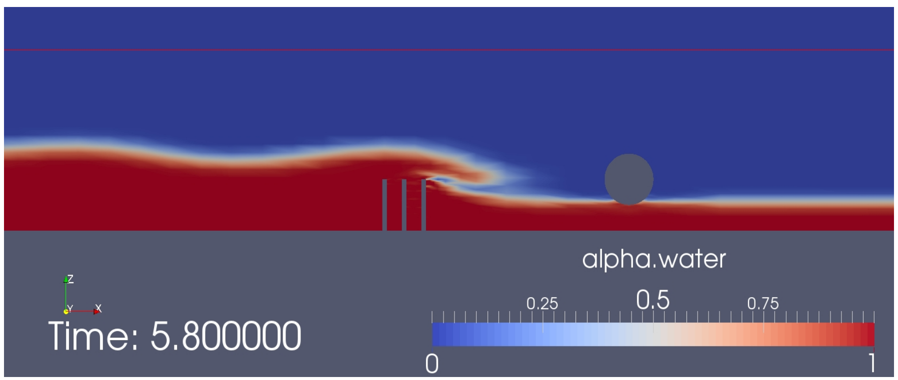
3D Numerical Results

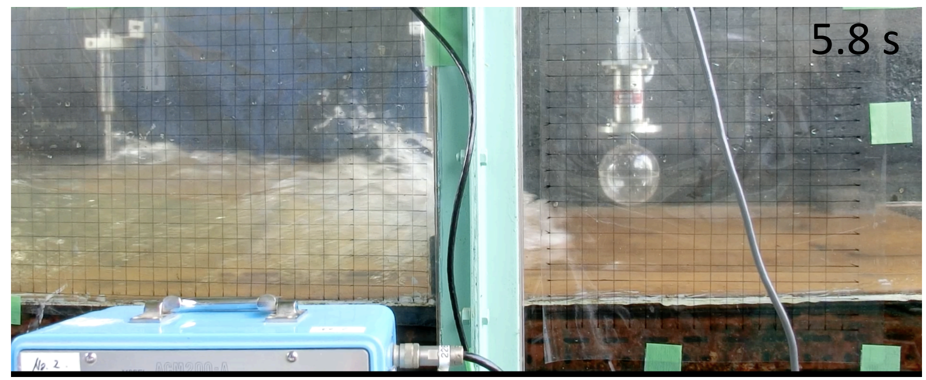


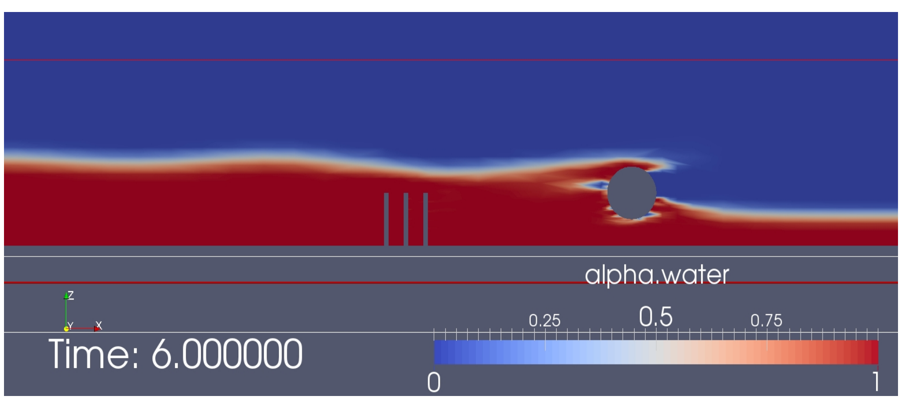
Comparison of wave pattern



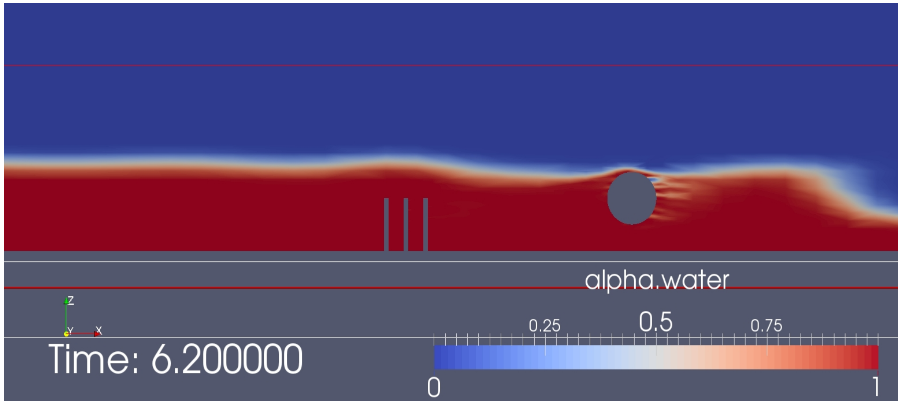


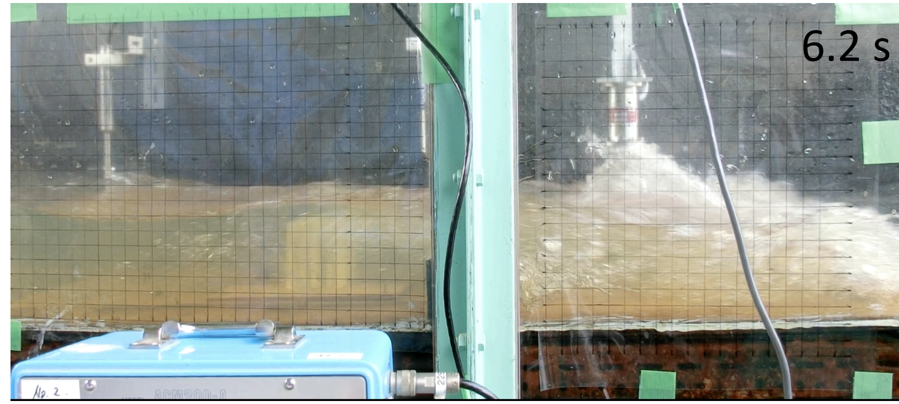


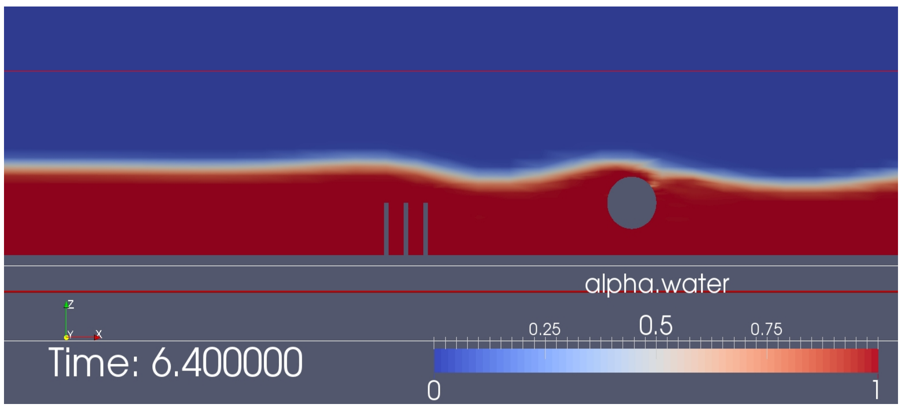


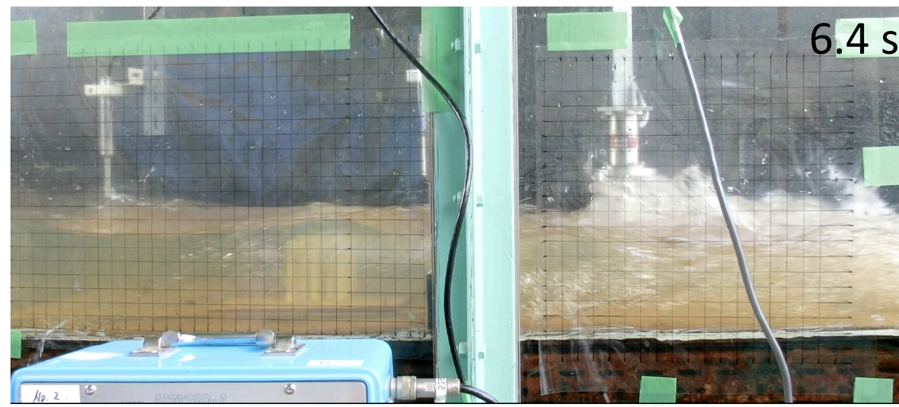


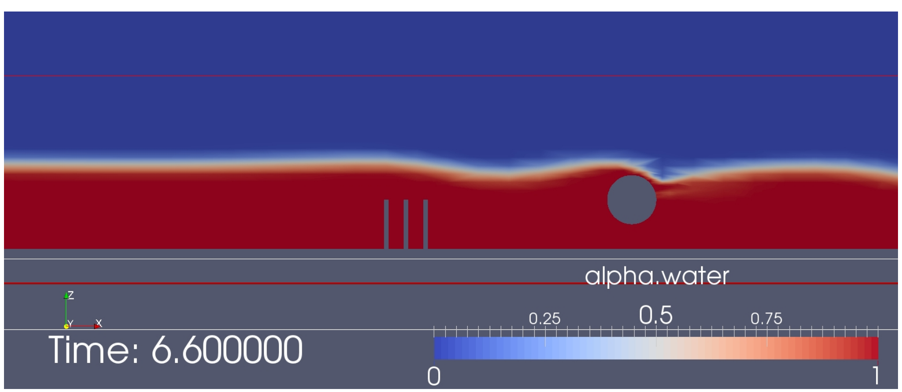
Comparison of wave pattern

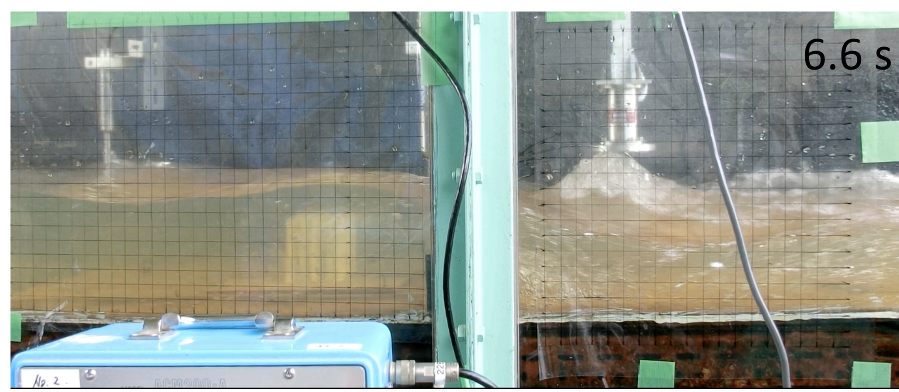


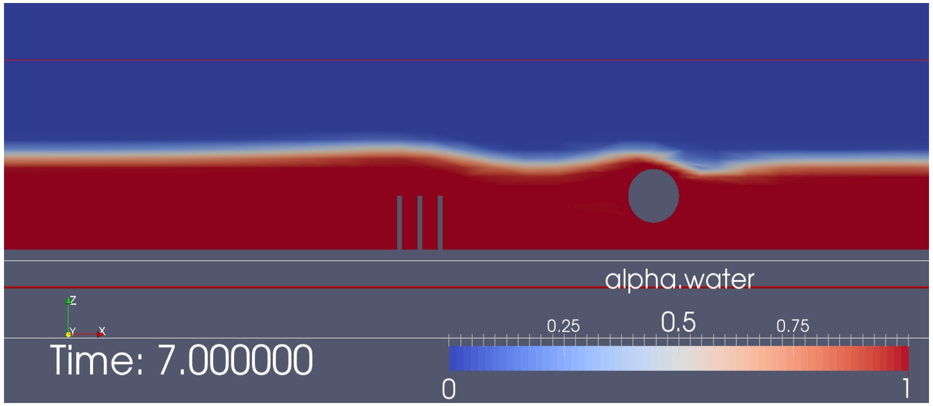


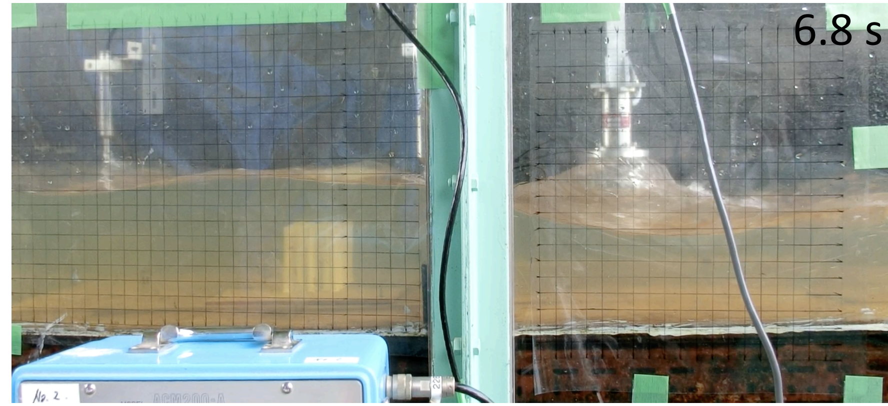


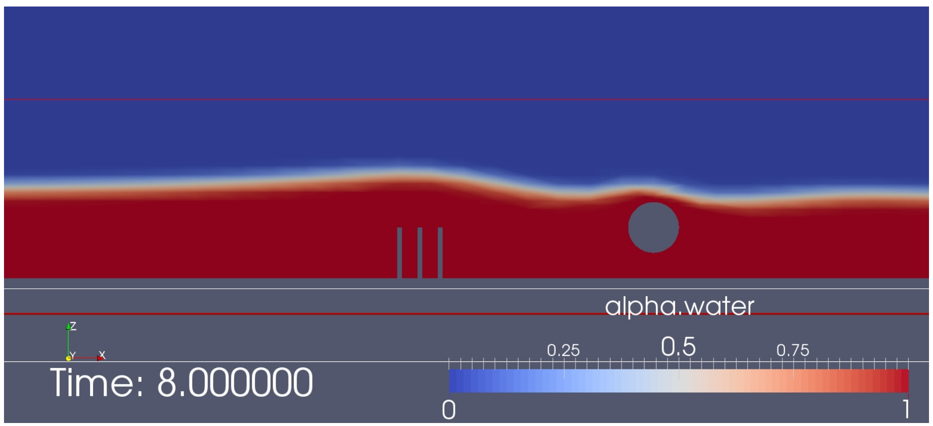


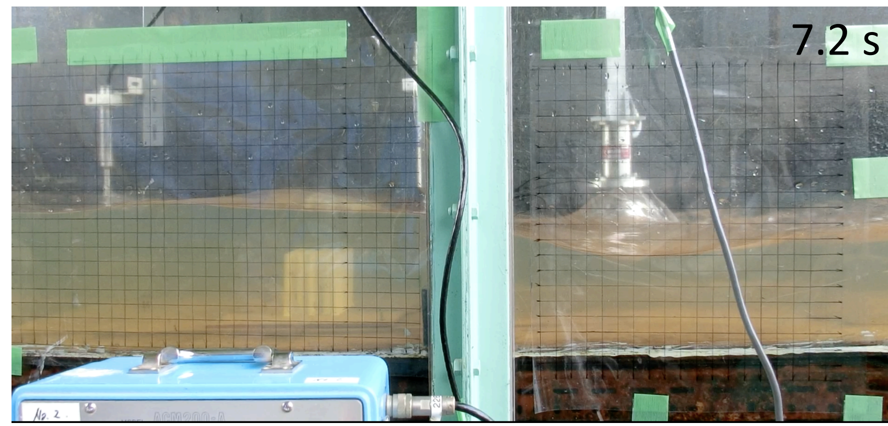


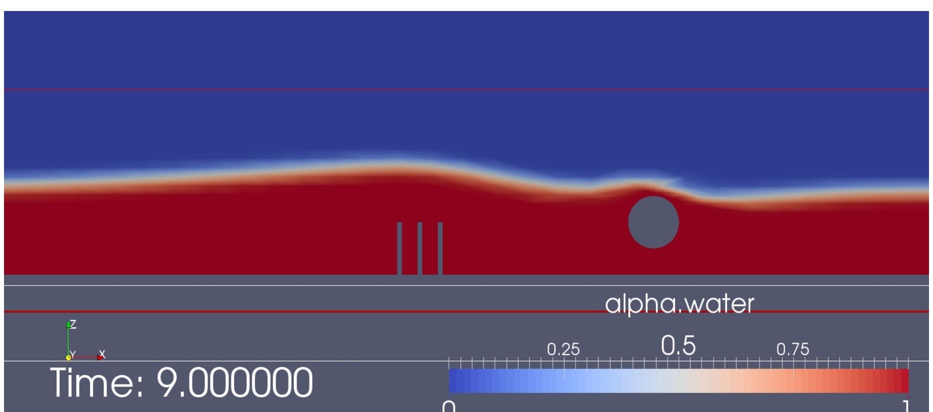



alpha.water
0.25
0.5
0.75
Time: 7.000000
0
1

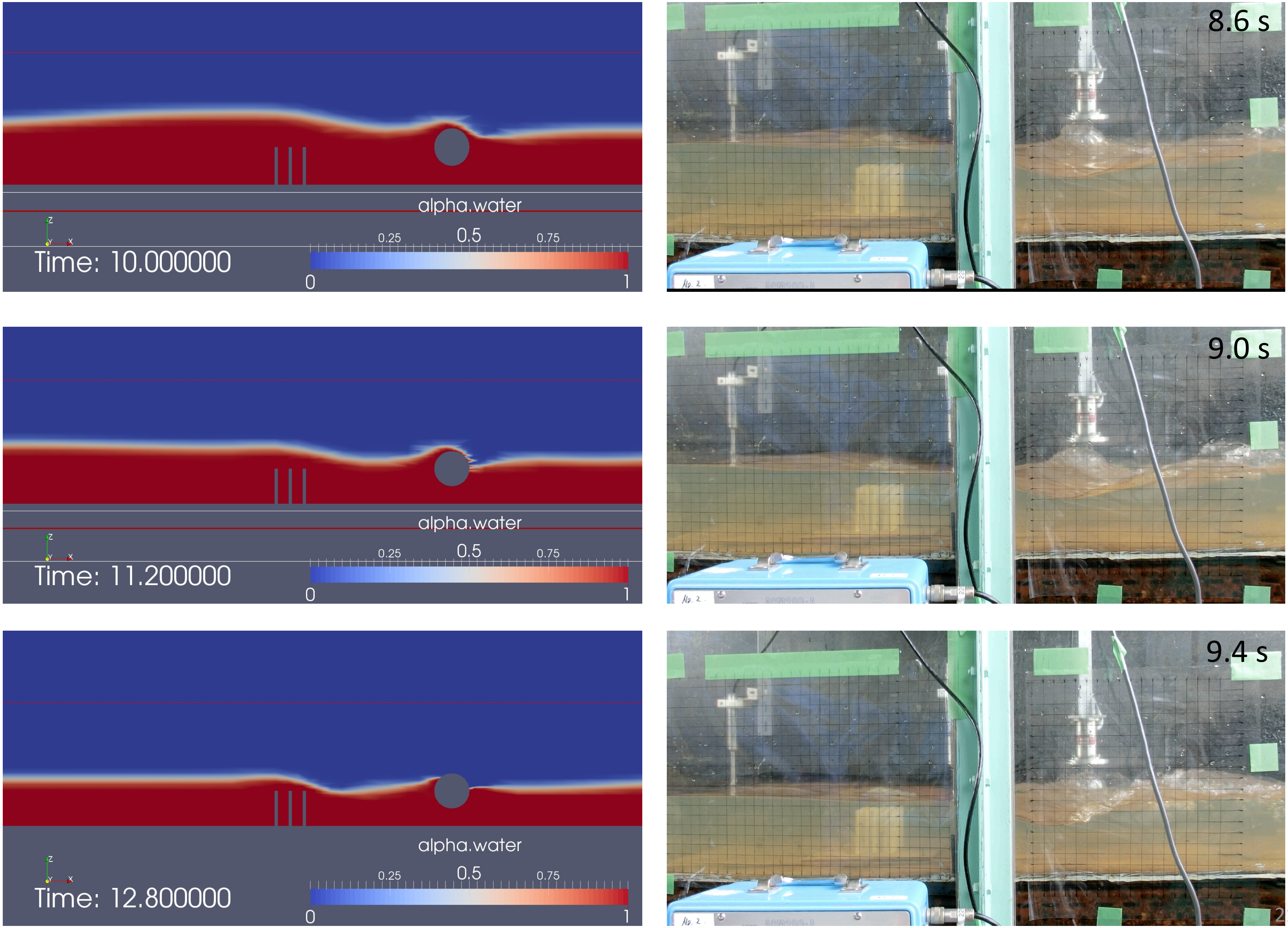

6.8 s


alpha.water
0.25
0.5
0.75
Time: 8.000000
0
1

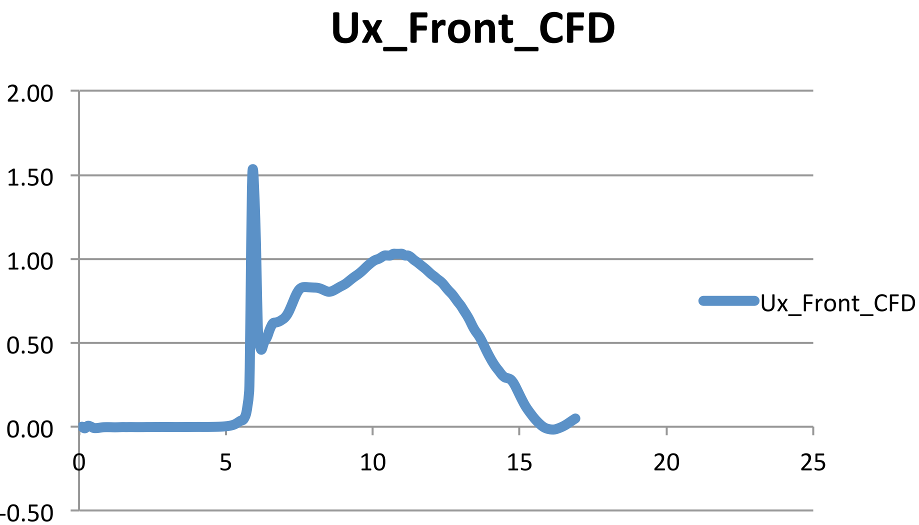

7.2 s

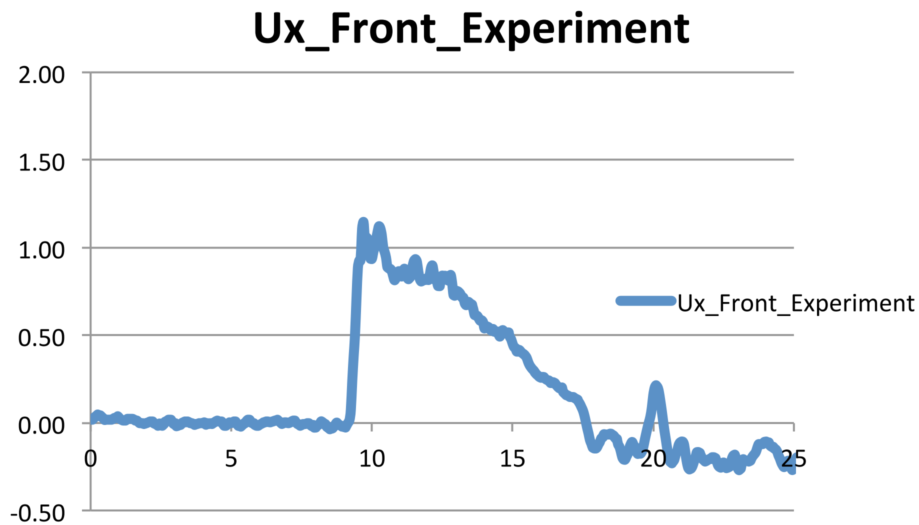

alpha.water
0.25
0.5
0.75
Time: 9.000000
0
1

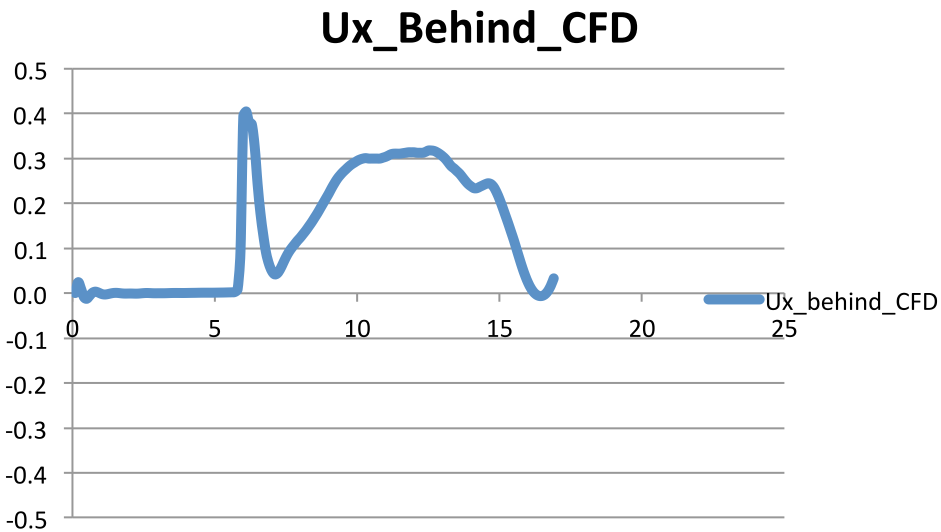

7.8 s

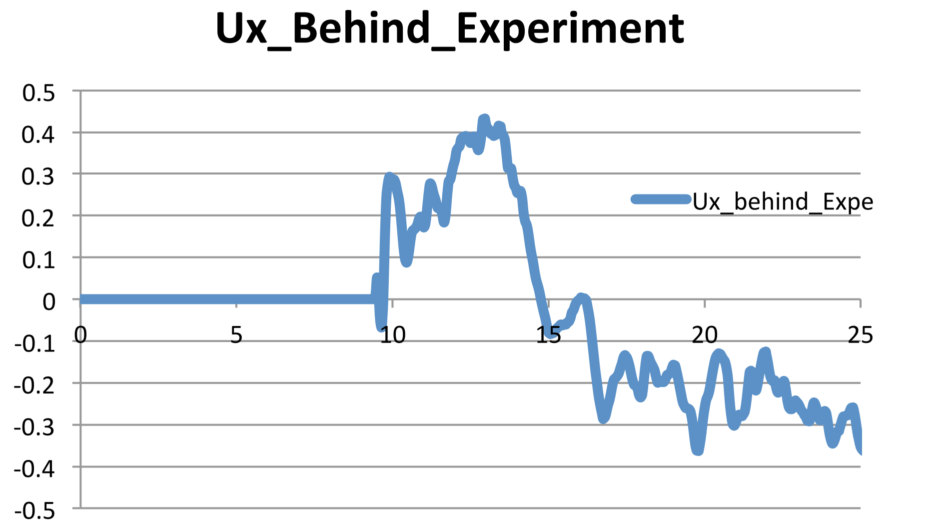

alpha.water
0.25
0.5
0.75
0
1
Time: 10.000000
8.6 s
alpha.water
0.25
0.5
0.75
0
1
Time: 11.200000
9.0 s
alpha.water
0.25
0.5
0.75
0
1
Time: 12.800000
9.4 s

Comparison of Flow Velocity

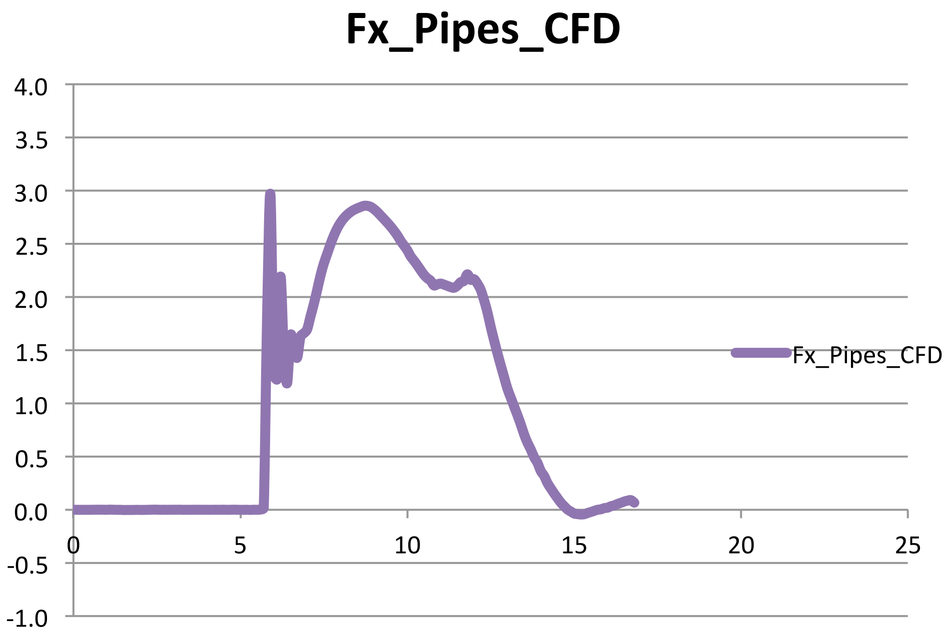


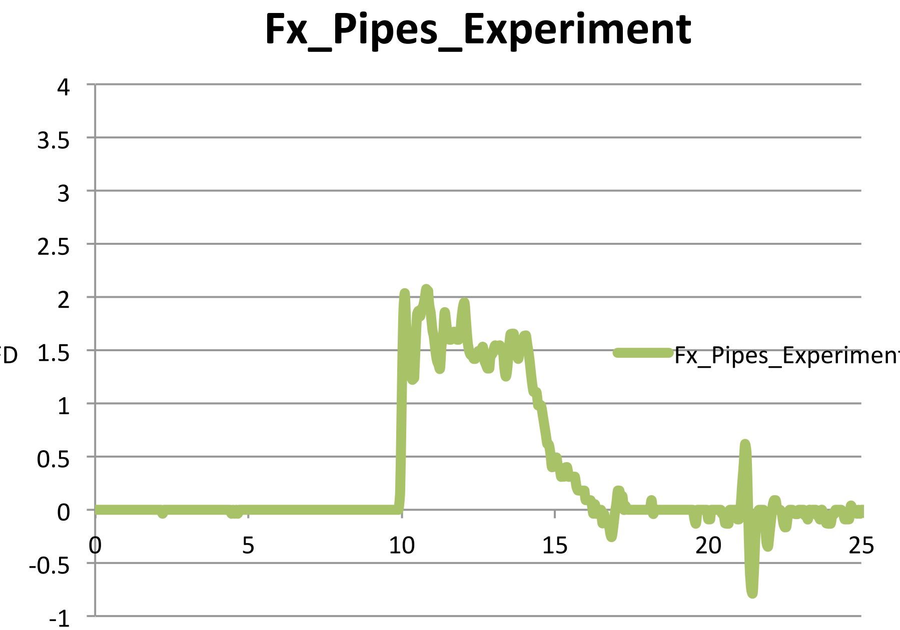


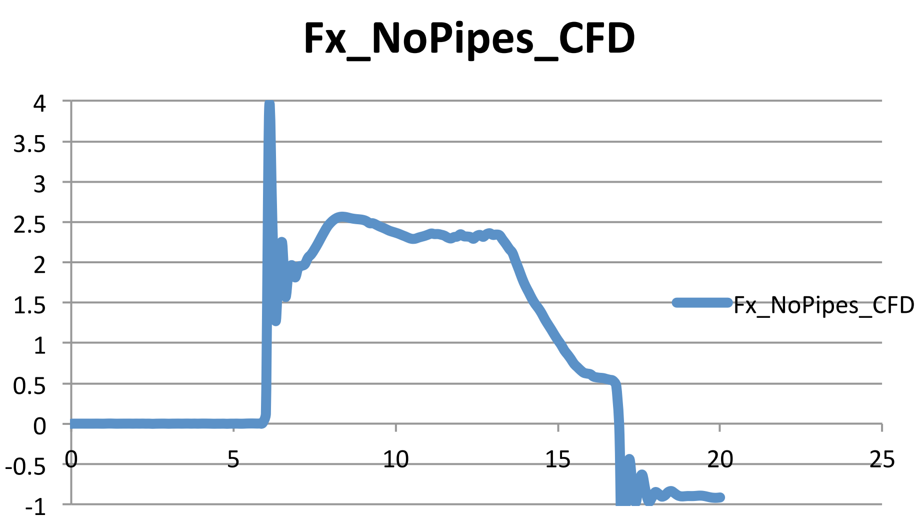


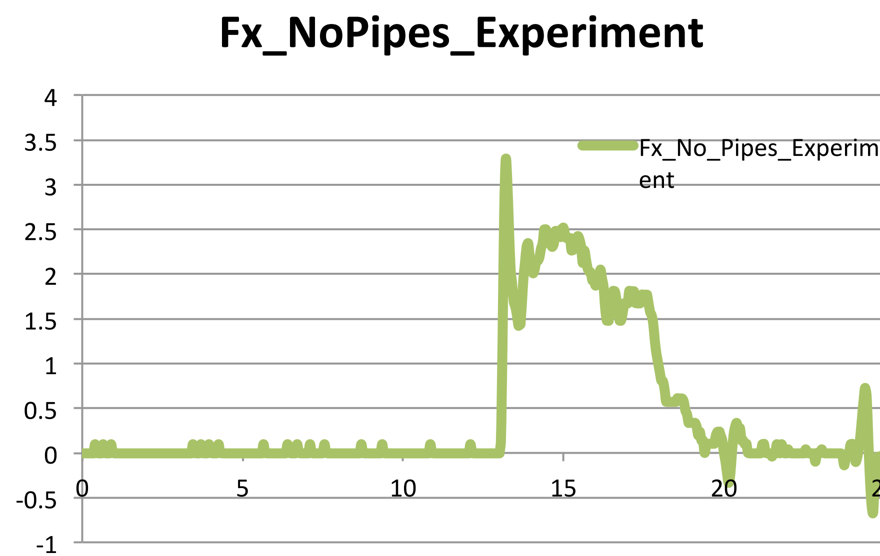


Comparison of Hydrodynamic Force







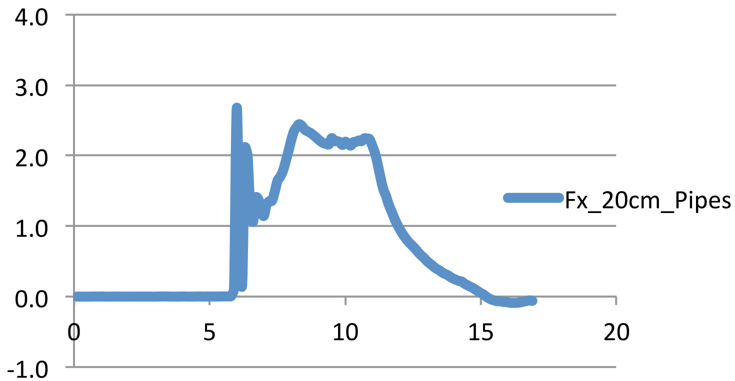


CFD Predictions of Longer Pipes

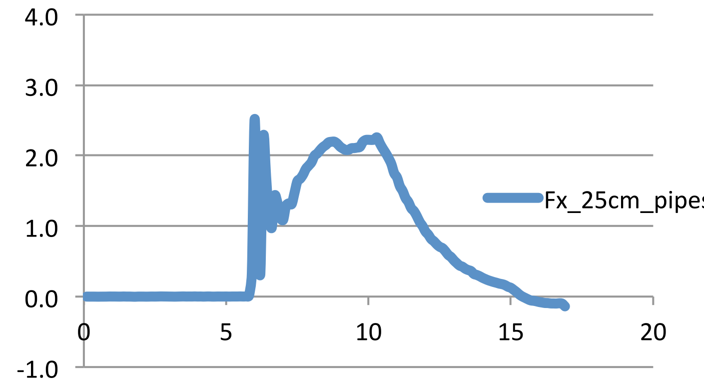
Problem with 10 cm pipes

- Not high enough to protect the whole tsunami wave height.
- Longer pipes will be more effective.
- CFD simulations were carried out to predict the hydrodynamic force acting on the tank for the case of longer pipes and the force predictions are as follows:

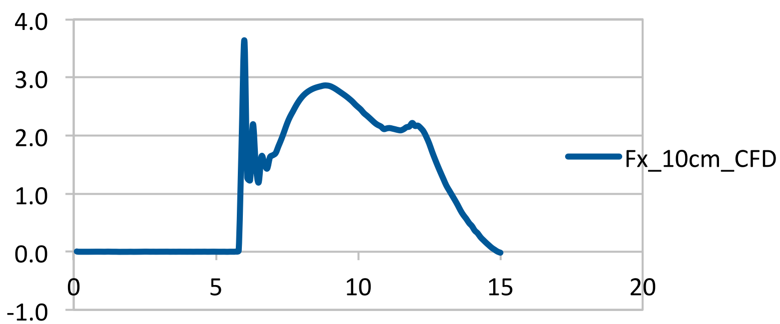
Fx_20cm_Pipes_CFD



Fx_25cm_Pipes_CFD

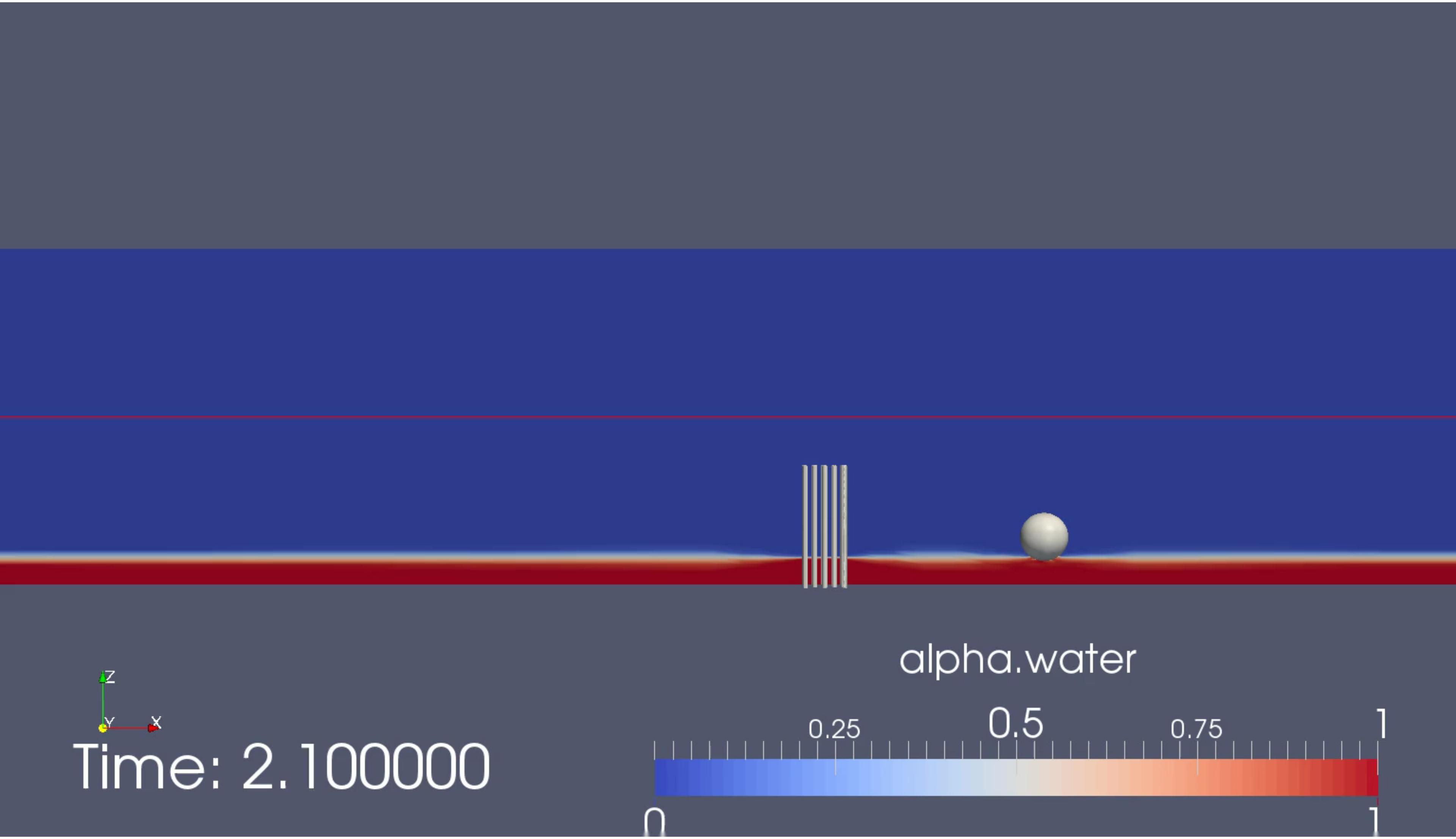


Fx_10cm_CFD



- Better reductions in hydrodynamic force were observed.
- Experiments should be carried out for 20 cm and 25 cm pipes

CFD Predictions of 25 cm Pipes



Conclusions

- Tsunami basin experiments were carried out and the effect of rigid pipes were investigated by comparing the velocity measured and hydrodynamic forces acting on the tank.
- A numerical model that can represent tsunami damage reduction by flexible pipes was established assuming that the pipes are rigid pipes. This numerical model was validated by comparing with tsunami basin experiments in terms of flow velocity, hydrodynamic forces and wave form. The CFD results showed good agreements with the experimental results.
- Based on this numerical model, predictions are carried out for longer pipes (20cm and 25cm). Better reductions of hydrodynamic forces were observed from simulations.

Future Work

- To protect oil and gas tanks from the attacks of large-scale tsunamis, a large number of flexible pipes would be placed along the coast near the oil refineries. In order to simulate the tsunami attack on a large area of flexible pipes, a body force model that can represent the flexible pipes at macro scale shall be developed based on current numerical model.
- Risk assessment for the cylindrical and spherical tanks under the attack of tsunami should be established, including the structural analysis of the base structures.