FOSSEE, IIT Bombay
OpenFOAM Case Study Project
July 15, 2022

Study of Multi-Form Single Chamber Oscillating Water Column

Bindusara Reddy
Departement of Ocean Engineering
INDIAN INSTITUTE OF TECHNOLOGY, Madras

Abstract

An Open Source Multiphase solver of the OpenFOAM library interFoam, is used for three-dimensional simulation of an Oscillating Water Column (OWC).
The program solves the Navier–Stokes equations for the interaction of waves and structures for two incompressible phases (water and air). Air and water pressures and velocities at various locations are validated with experimental results from The Ocean Engineering Journal's 'Validation of OpenFOAM for oscillating water column three-dimensional modeling'. The model is further used to perform CFD analysis on related geometries for the chamber. The geometries analysed were two trapezoidal cases with symmetry about the vertical axis but asymmetry along the horizontal axis. These results are compared with the rectangular chamber and shown to generate a greater pressure and air flow velocity.

1 Introduction

Oscillating Water Columns are Wave Energy Converters with a fixed Hollow Chamber open to the Ocean below. The action of the wave that advances toward the chamber causes the air trapped between the free surface and the chamber to get compressed and rarefied which can be used to drive a turbine.

Wave Energy Converters that use a Oscillating Water Column for Wave Energy extraction offer many advantages such as low maintenance and robustness due to the absence of any submerged moving parts and the structure can be made fixed/floating or installed offshore/onshore.

Experimental Modelling of such structures faces many challenges such as scale effects and construction issues. Consequently, it does not have a reliable way to measure all parameters of the interface dynamics such as the velocities, wave elevations and pressures conjointly. They also have specific challenges as hydrodynamic and pneumatic flows require different model scales and thus would require a full scale model for accurate analysis.

Computational Fluid Dynamical simulations become very attractive for such analysis. They offer all the results in an in depth and comprehensive manner.

2 Problem Statement

Stokes Ist Order Waves are directed towards a OWC chamber and the pressure and velocity profiles of various locations at specific time instances are compared with variations of geometry for the chamber to obtain performance improvements in the pressure and velocity of the airflow through the top slot. The Geometry analysed is a Single Chamber Oscillating Water Column (OWC).
The chamber is 0.34m long in the x direction, 0.68m wide in the y direction and 0.35m tall in the z direction. The chamber is perfectly rectangular with walls of thickness 0.02m. The chamber is a fixed, immovable, rigid structure.

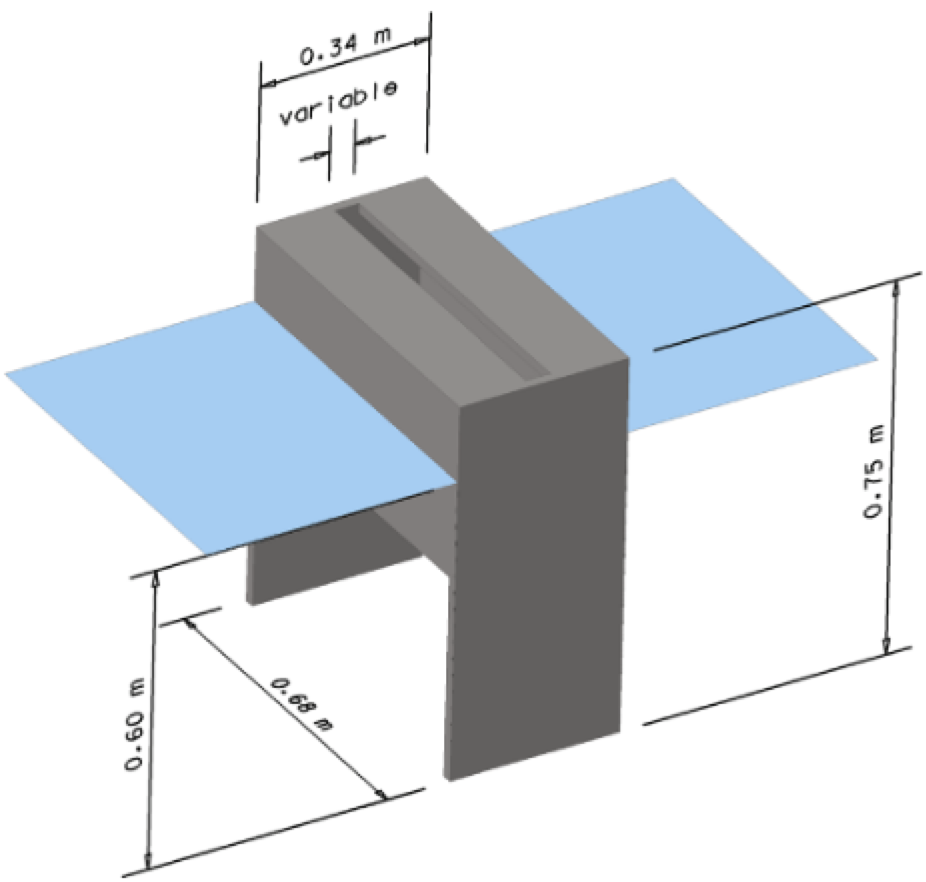


Fig. 4. Chamber model geometry.

The chamber is present in a wave flume of 9m length 0.68m width and 1.1m height. The water level is 0.6m in the z direction. The wave flume inlet is used to generate waves of time period 1.3s, 1.7s, 2.2s and 3.2s. The waves are impinged on the chamber and the pressure and velocity variations are compared for top slot openings of width 50mm and 9mm respectively.

Table 1
Characteristics of the regular wave tests used for model validation.

CASE	*H* (m)	*T* (s)	Slot1 (mm)	Slot2 (mm)
C24	0.08	1.3	50	9
C26	0.08	1.7	50	9
C27	0.08	2.2	50	9
C28	0.08	3.2	50	9

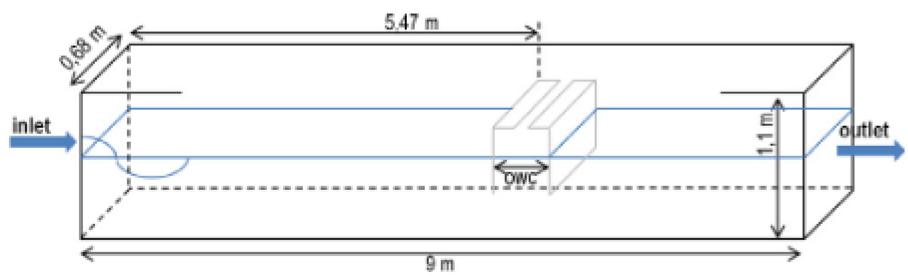


Fig. 8. Numerical setup scheme.

The geometry of the chamber is changed for a trapezoidal and inverted Trapezoidal Case and compared with the results for the Rectangular Case to check for increase in the pressure and velocity profile

3 Governing Equations and Models

OpenFOAM (Open source Field Operation and Manipulation) is the model used for this analysis. OpenFOAM has a multiphase solver interFoam with boundary conditions for wave generation and absorption.
The Solver solves the Navier Stokes Equations for two incompressible, isothermal, immiscible fluids.
The Equations solved are :

3.1 Continuity Equation

The constant-density continuity equation is

$$\frac{du_j}{dx_j} = 0 \tag{1}$$

3.2 Momentum Equation

The constant-density continuity equation is

$$\frac{d(\rho u_i)}{dt} + \frac{d(\rho u_i u_j)}{dx_j} = -\frac{dp}{dx_i} + \frac{d(\tau_{ij} + \tau_{ji}}{dx_j} + p g_i + f_{\sigma i} \tag{2}$$

u represent the velocity, g_i the gravitational acceleration, p the pressure and τ_{ij} and $\tau_{t_{ij}}$ are the viscose and turbulent stresses. $f_{\sigma i}$, is the surface tension.
The density ρ is defined as follows:

$$\rho = \alpha\rho_1 + (1-\alpha)\rho_2 \tag{3}$$

α is 1 inside fluid 1 with the density ρ_1 and 0 inside fluid 2 with the density ρ_2 . At the inter-phase between the two fluids α varies between 0 and 1.

The surface tension $f_{\sigma i}$, is modelled as continuum surface force. It is calculated as follows :

$$f_{\sigma i} = \sigma\kappa\frac{d\alpha}{dx_i} \tag{4}$$

σ is the surface tension constant and κ the curvature. The curvature can be approximated as follows :

$$\kappa = -\frac{d\eta_i}{dx_i} = -\frac{d(\frac{d\alpha/dx_i}{|d\alpha/dx_i|})}{dx_i} \tag{5}$$

3.3 Equation for the interphase

In order to know where the interphase between the two fluids is, an additional equation for α has to be solved.

$$\frac{d\alpha}{dt} + \frac{d\alpha u_j}{dx_j} = 0 \tag{6}$$

3.4 Airy Wave Equation

The Laplace Equation is solved using the Cauchy Poisson equation at free surface

$$\frac{d\phi}{dz} + \frac{1}{g}\frac{d^2\phi}{dt^2}|_{z=0} = 0 \tag{7}$$

with appropriate boundary conditions to obtain the following potential function

$$\phi = \frac{agcosh(k(d+z))}{cosh(kd)}sin(kx-\omega t) \tag{8}$$

The stokes Ist order wave model is used for generating and solving for the waves.
They follow the following equation :

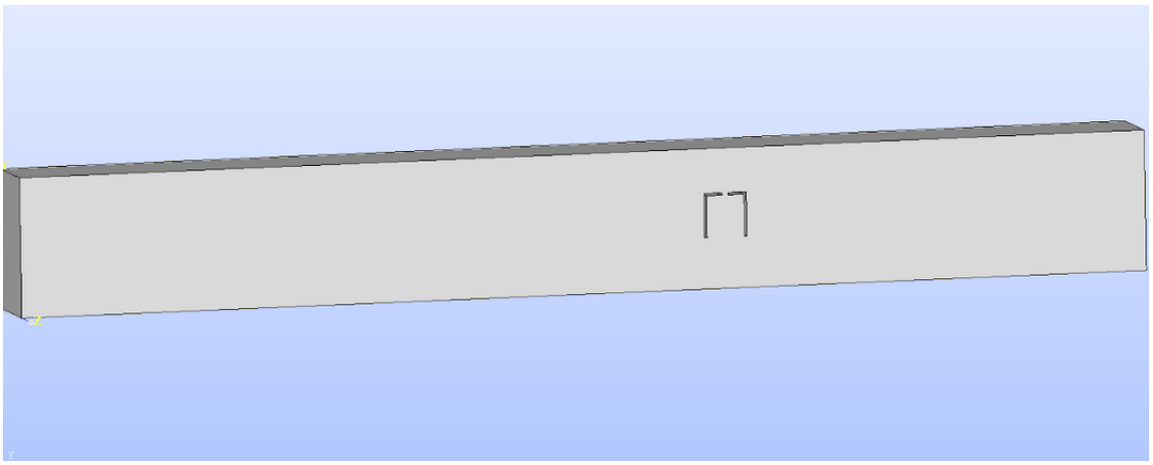
$$\eta = \frac{H}{2}cos(kx-\omega t+\phi) \tag{9}$$

4 Simulation Procedure

The simulation was performed by sculpting and meshing the geometry in Salome, solving the reduced navier stokes equations with the interphase equation using the interFoam solver and then analysing the results in paraView.

4.1 Geometry and Mesh

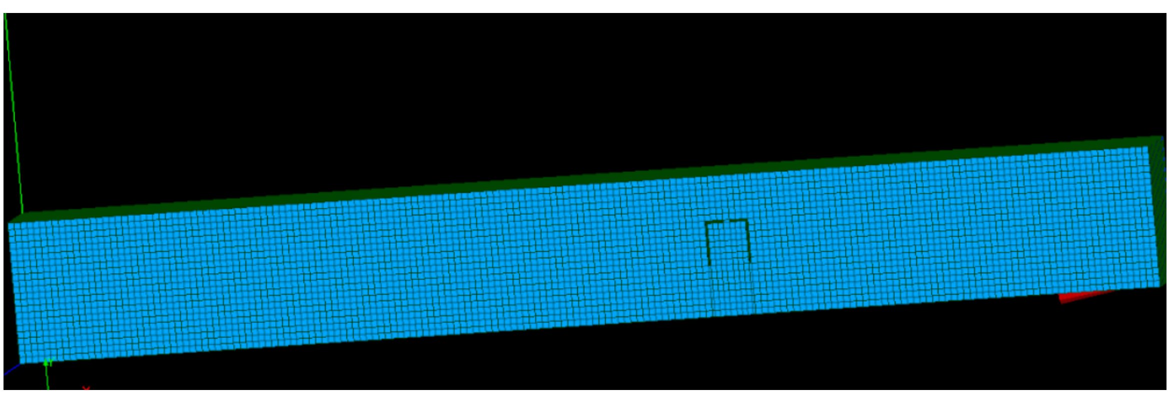
The geometry consists of 9m length wave flume of 1.1m height, 0.68m width. The Chamber is present at distance of 5.47m from the inlet. The water level starts off at 0.6m



The geometry is created in the geometry module of Salome. The faces of the chamberwalls , the outlet , inlet , top, ground, back wall are named and grouped. The geometry is partitioned to enable proper hexahedral meshing.

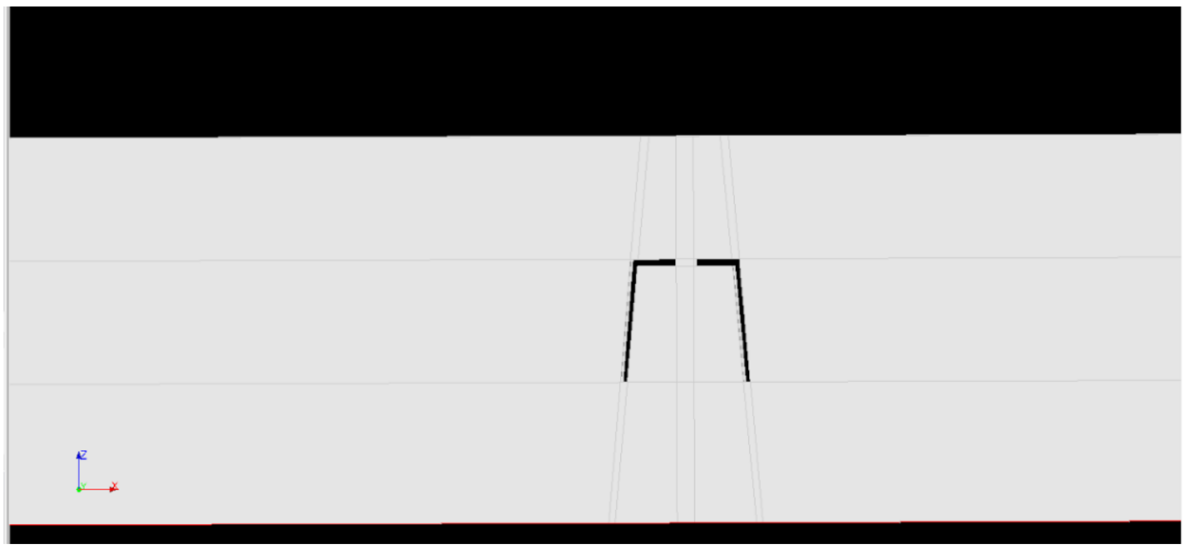
The domain is then exported to the Mesh Module of Salome where extrusion 3D, quadrangle Mapping, and wire discretization algorithms are applied to obtains the points, faces and volumes of the mesh. A local length hypothesis is applied on the 1D meshing algorithm to control the size of each volume and to maintain a cuboidal shaped cell.

The final Mesh contains 444000 cells and are within a tolerance of e-6 having a edge length of 0.025m is exported as a .unv file.

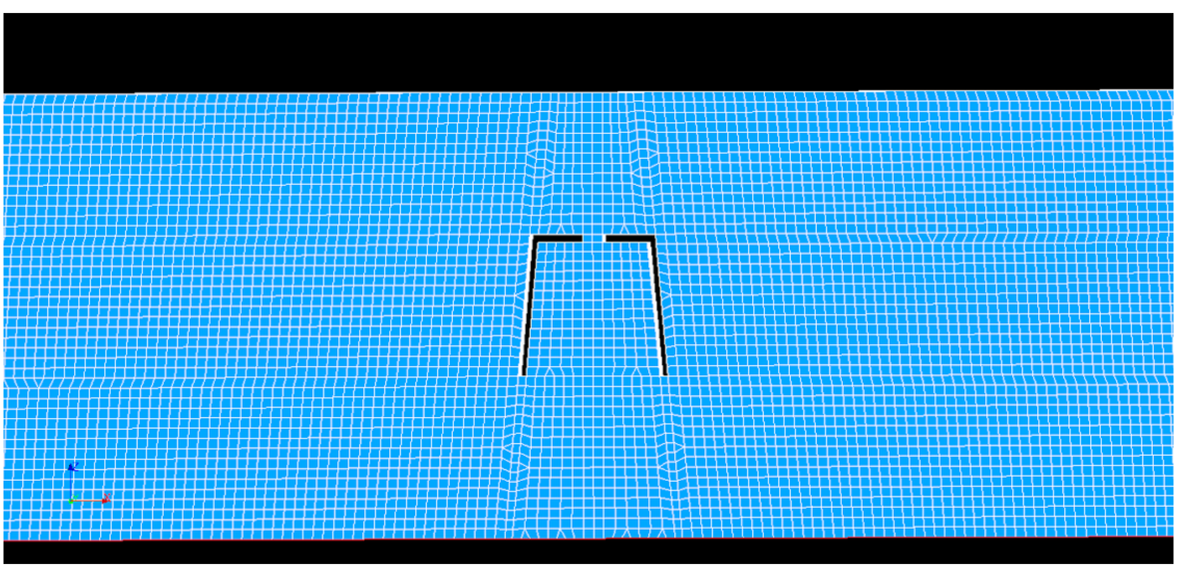


A total of 6 meshes are created for the three geometries of the chamber and the 2 top slot widths that each geometry can contain.

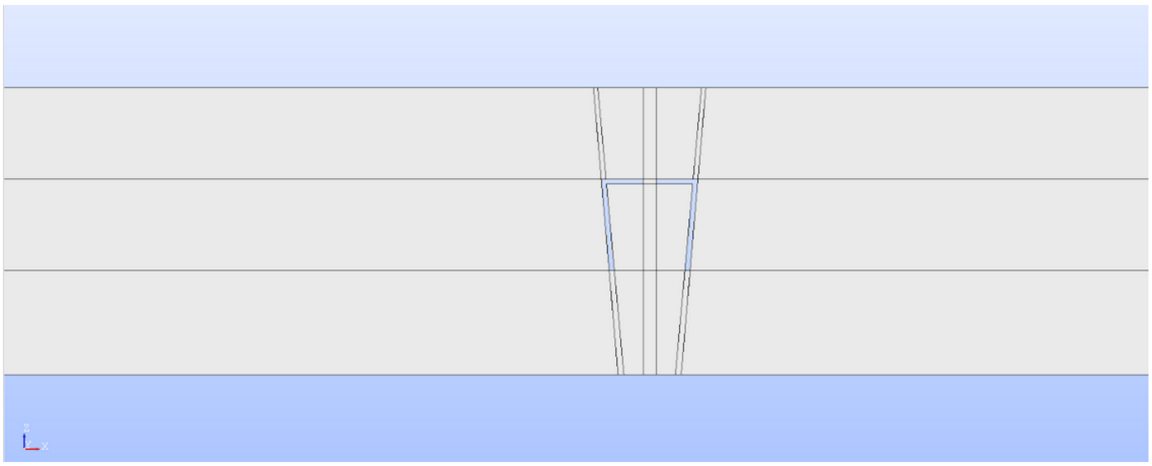
The Geometry of the Trapezoidal case is defined as having the same volume as the Rectangular case, but with its top width being 0.31 m and the bottom width being 0.37 m.
The area of chamber cross section * width therefore gives the same volume.

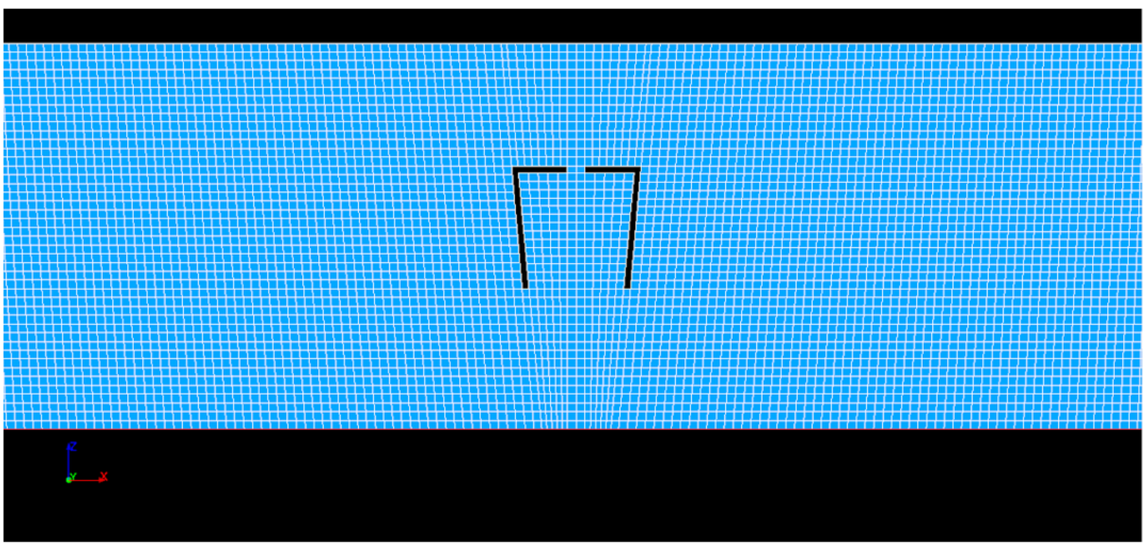


The Hexahedral Mesh is Computed by Partitioning the complex Geometry into Simple Cuboidal blocks using Planes.



The Geometry of the Inverted Trapezoidal case is defined similarly but the chamber has a top width of 0.37 m and a bottom width of 0.31 m





The Mesh consists of 444000 cells with a local length parameter of 0.025 m

4.2 Initial and Boundary Conditions

The Wave period is set as $1.3s$, $1.7s$, $2.2s$ and $3.2s$ for the particular case that is solved. The Wave Height is 0.08m. The wave model is set to StokesI in the Wave Properties file. Wave paddle (n = 1) is set at the inlet patch. The rampTime is set to 0.5 seconds.
The front patch is set to act as a symmetryPlane. The ground and the back plane act as walls along with the faces of the chamber walls.
The alpha value is set to 1 in the region of the water domain and 0 in the air domain.
The value of g is set to a uniform value of -$9.8m/s^2$ in z direction.
The transport values of the two phases are set to $\nu = 1e^{-}6m^2/s$, $\rho = 1000kg/m^3$ for water and $\nu = 1.48e^{-}5m^2/s$, $\rho = 1kg/m^3$ for air. A Newtonian transport model is used and a surface

tension of 0.07 Nm is applied.

4.2.1 $Alpha$ Boundary conditions

Alpha is a dimensionless number that represents the phase fraction of a cell. We set it be 0 everywhere representing air and rewrite the internalField vector using setFields to 1 everywhere water is present.
The inlet is set to waveAlpha - Uniform 0
The outlet, ground, back, chamberwalls are set to zeroGradient
The top is inletOutlet - Uniform 0
The symmetryplane patch is a symmetryPlane

4.2.2 $P - \rho gh$ Boundary conditions

$P - \rho gh$ is the dynamic pressure found by subtracting the Hydrostatic Pressure from the total Pressure.With Dimensions of Kg/ms^2 it's Set to a uniform 0 Field vector.
The inlet and outlet is set to fieldFluxPressure - Uniform 0
The ground, back, chamberwalls are set to zeroGradient
The top is totalPressure - Uniform 0
The symmetryplane patch is a symmetryPlane

4.2.3 U Boundary conditions

U represents the Velocity Field at each point. Dimensions of m/s it's Set to a uniform 0 0 0 Field vector in all directions.
The inlet and outlet is set to waveVelocity - Uniform 0
The ground, back, chamberwalls are set to fixedValue - Uniform 0
The top is inletOutletPressureVelocity - Uniform 0
The symmetryplane patch is a symmetryPlane

4.3 Solver

The Solver utilized in the simulations is the OpenFOAM solver *interFoam*. It is an incompressible, transient, isothermal multiphase solver. The solver utilizes the Volume of Fluid (VOF) Method for tracking the free surface.

The PIMPLE algorithm is run which is a combination of Pressure Implicit with Splitting of Operators (PISO) and Semi Implicit Method for Pressure Linked Equations (SIMPLE).

The solver uses Euler ddtSchemes;
Gauss linear gradSchemes;
Gauss linear orthogonal laplacianSchemes;

Linear interpolationSchemes;
Orthogonal snGradSchemes.
For the Divergence Schemes - divSchemes
div(rhoPhi,U) - Gauss linearUpwind grad(U);
div(phi,alpha) - Gauss vanLeer;
div(phirb,alpha) - Gauss linear;
div(((rho*nuEff)*dev2(T(grad(U))))) - Gauss linear;

The fvsolutions is defined to utilize the Preconditoned Conjugate gradient for $P - rgh$;
the Generalised geometric-algebraic multi-grid solver for $P - rghFinal$;
the Preconditioned Bi-Conjugate gradient for U and $UFinal$.

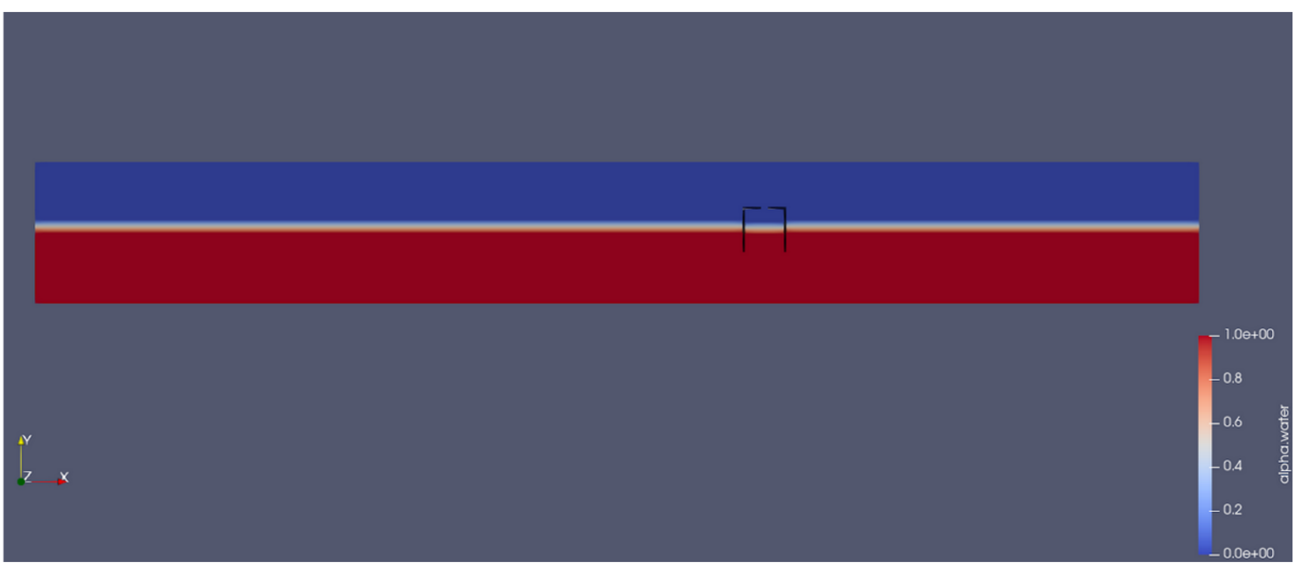
The Momentum Predictor is switched off and the number of correctors in the PIMPLE algorithm is set to 2 while the non orthogonal correctors is set to 0.

The solver timestep is kept 0.01 s and the simulations are run for 80 s with 0.1 s write interval. The timestep is adjustable during runtime. The Max Courant Number and the Max Alpha Courant is set to 0.65.

The Simulation converges in 2 hrs using scotch decomposition to break the domain into 20 sub domains to run it in parallel on 20 cpu threads.

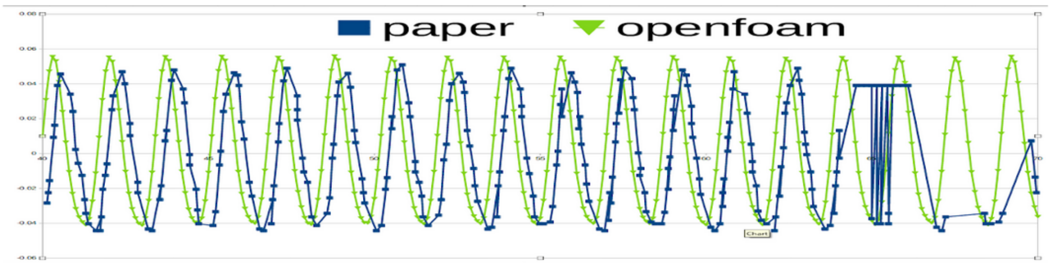
5 Results and Discussions

The Simulation results are compared with the results presented in the paper. The datapoints were extracted from the paper using the opensource software WebPlotDigitizer. The PostProcessing is performed in ParaView.
The water level at $t = 0$:

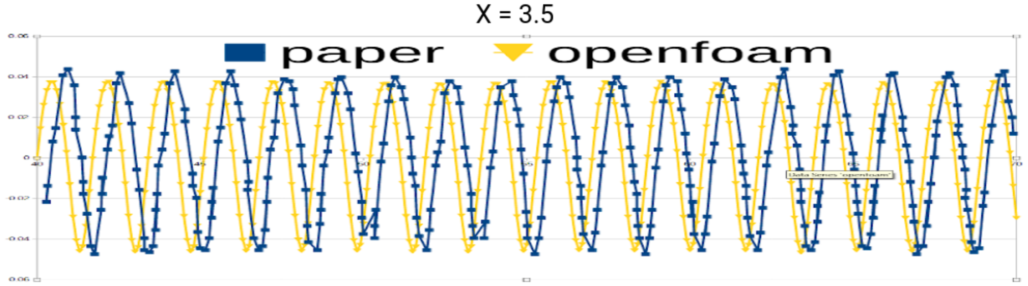


5.1 Rectangular Chamber

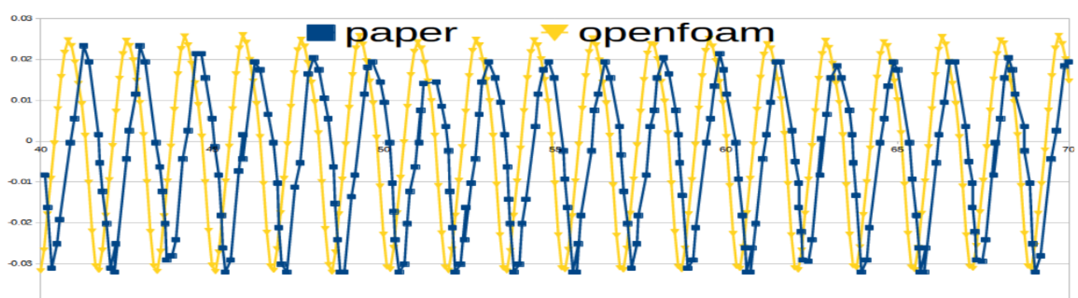
The Free Surface elevation is plotted at the eight locations where wave gauges have been placed and the measurements are compared for the 50 mm top slot chamber, 1.7 s wave in the timeframe 40 - 70 s.



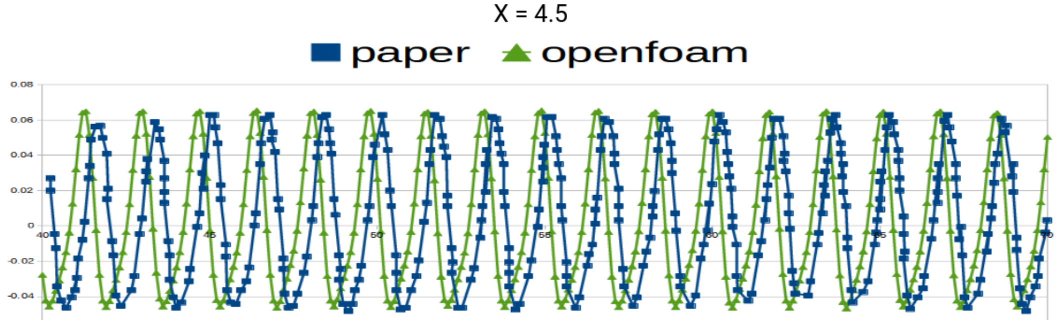
X = 3.5



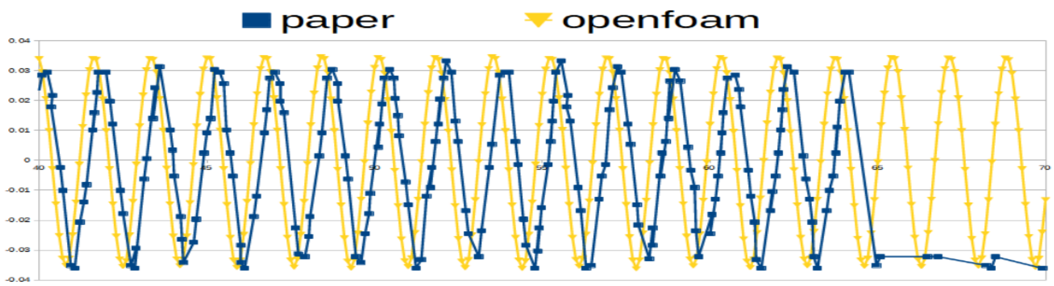
X = 3.85



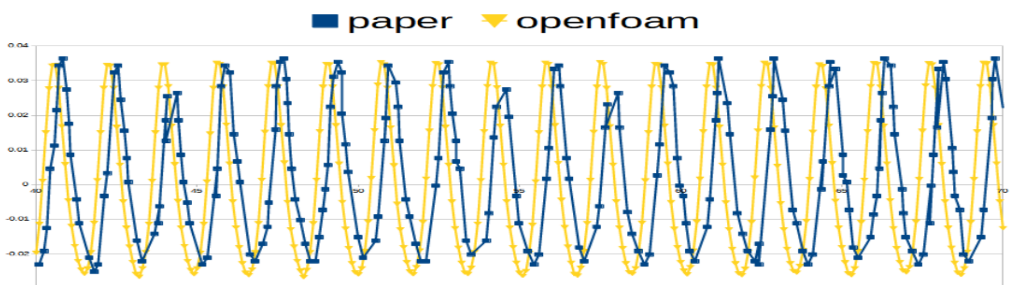
X = 4.5



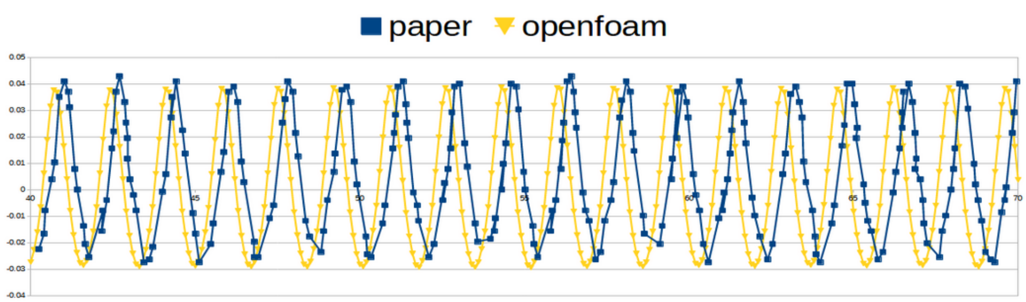
X = 5.42



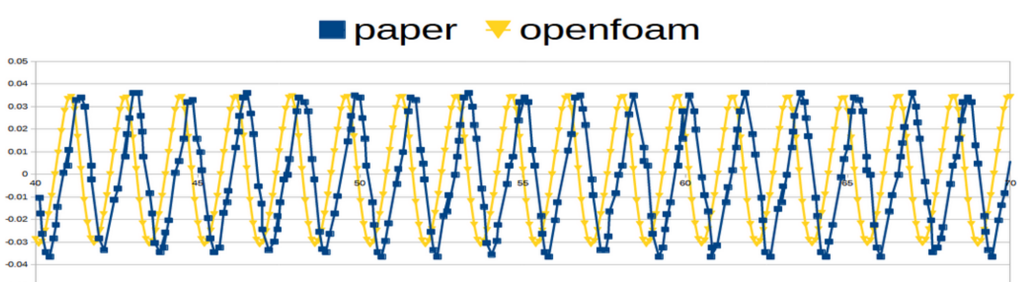
X = 5.86



X = 7.31



X = 7.66



X = 8.31

From the Plots we see the presence of a phase shift in the wave but the amplitudes and periods closely agree with wave gauge measurements.

Next the Surface elevations of chambers with different top slots are compared at three wave gauges present inside the chamber.

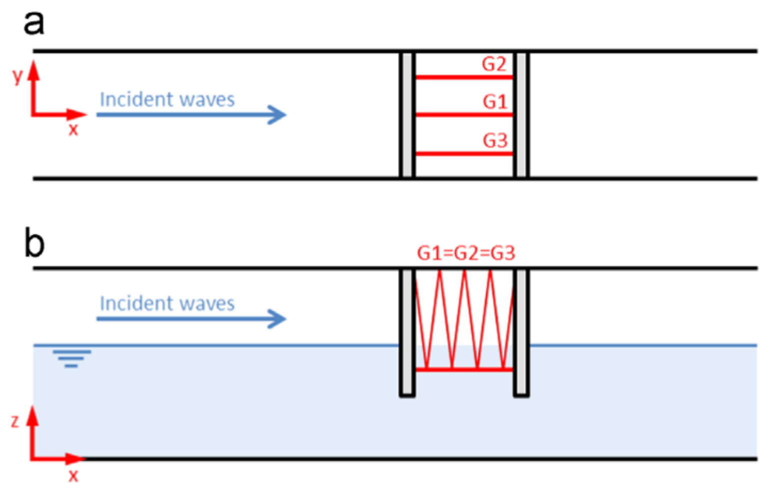
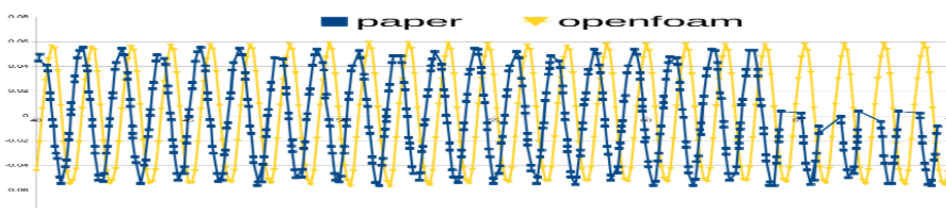
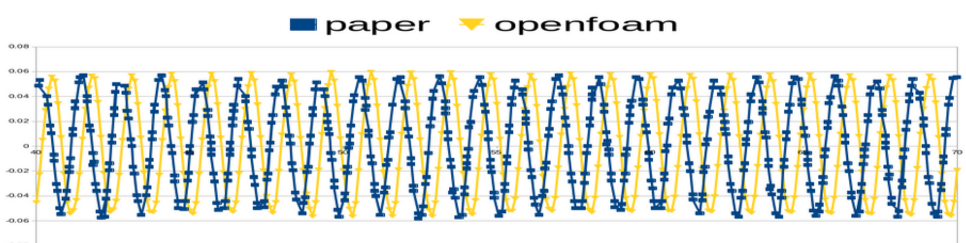
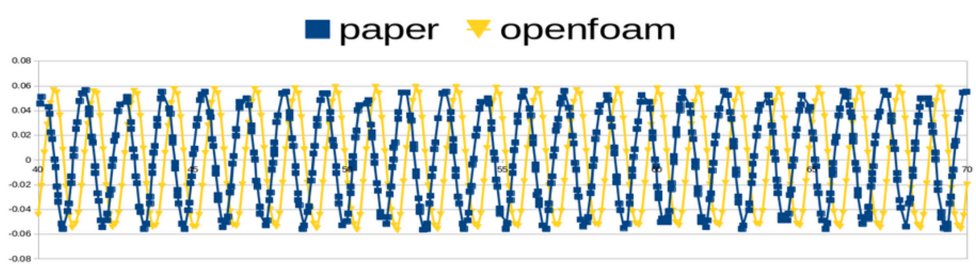


Figure 1: Location of in chamber Wave gauges

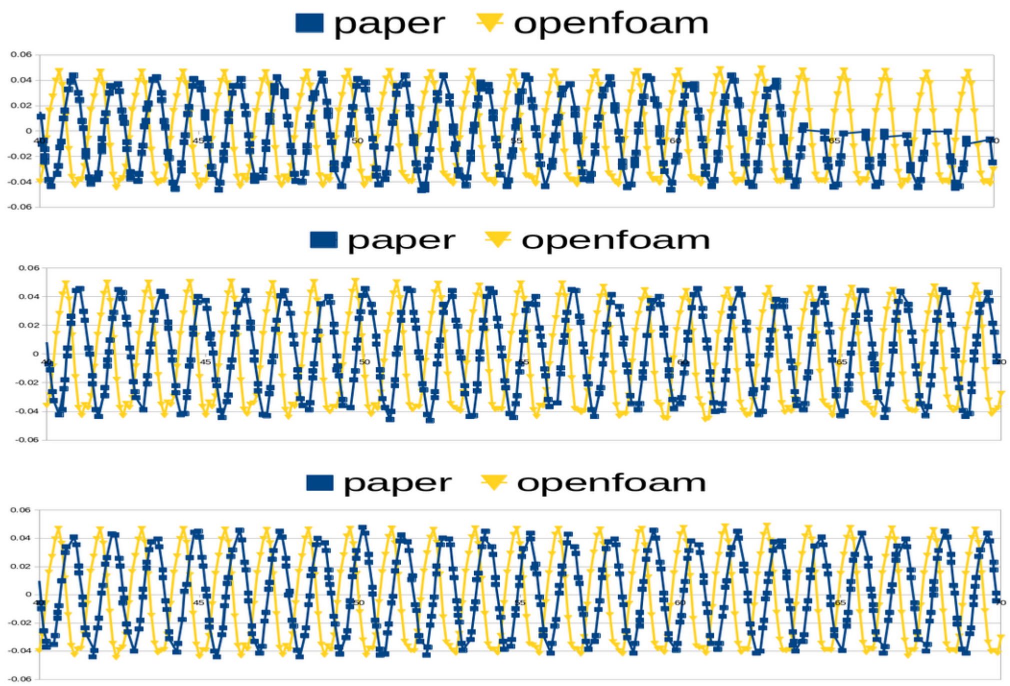


50mm
C24





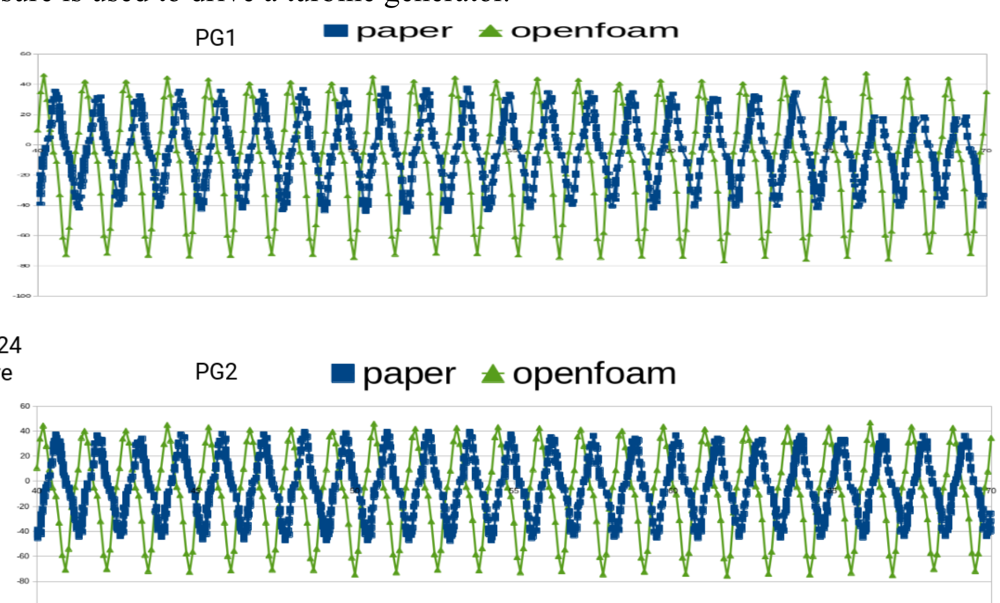
9mm
C24



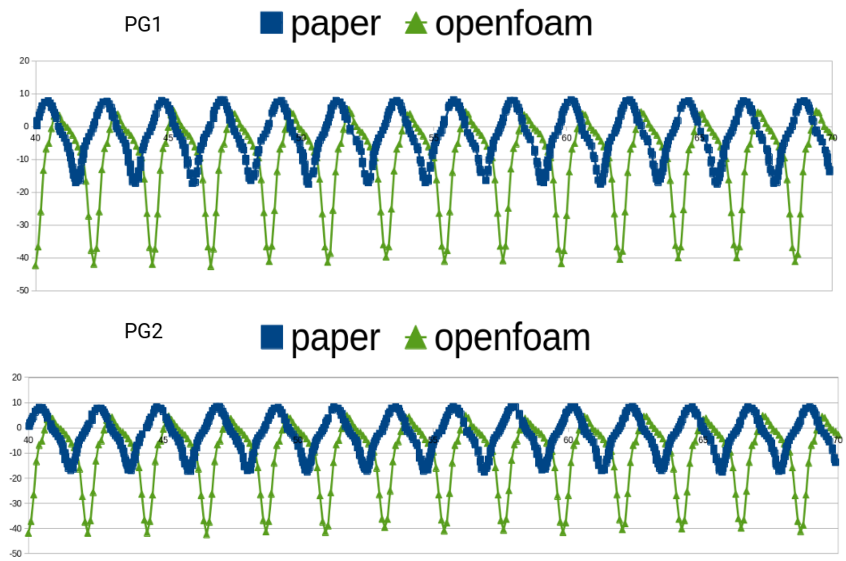
The resulting waveform in the chamber closely matches the results observed in experiments.

Here we compare the Pressure Response of the Air at the 9mm chamber top slot for three different wave periods, which is direct measure of the efficiency of the OWC for power extraction as the pressure is used to drive a turbine generator.

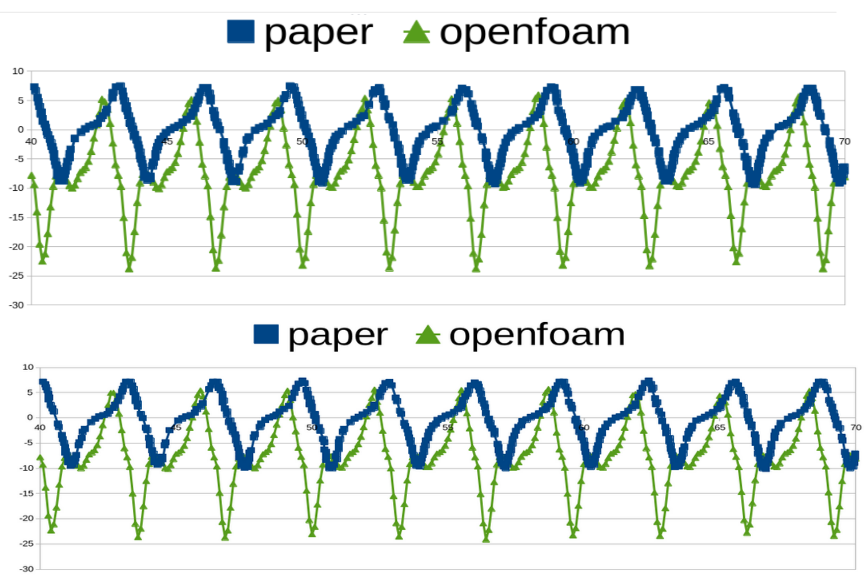
9mm C24
Pressure



9mm C27
Pressure

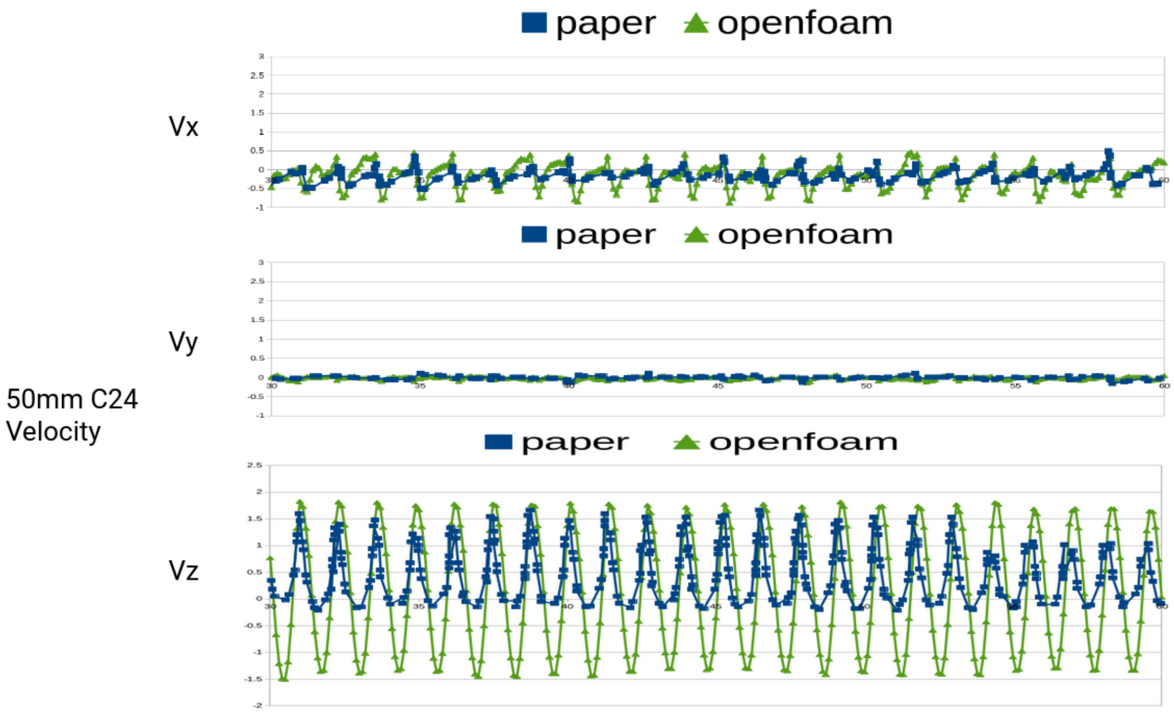


9mm C28
Pressure

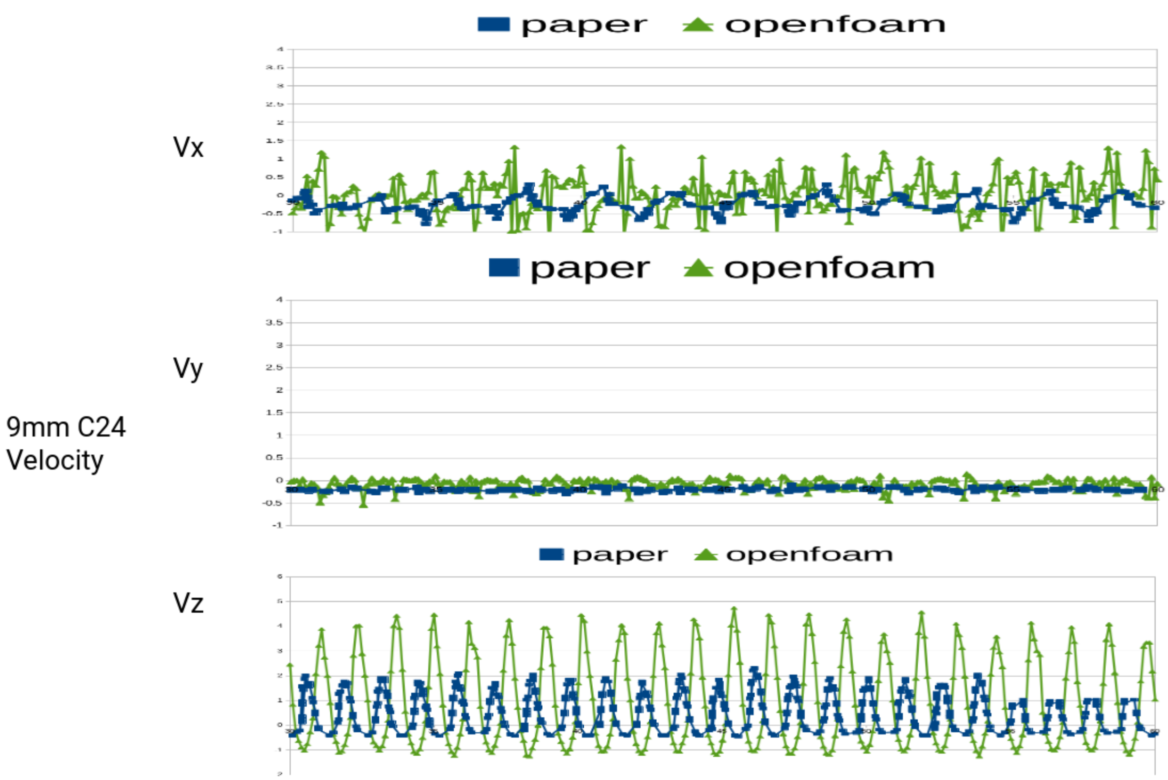


The Pressure Response is overestimated by the solver during the inflow of air consistently due to the specificity needed to solve for fluid, structure and air interactions but the general shape of the pressure form is matched closely.

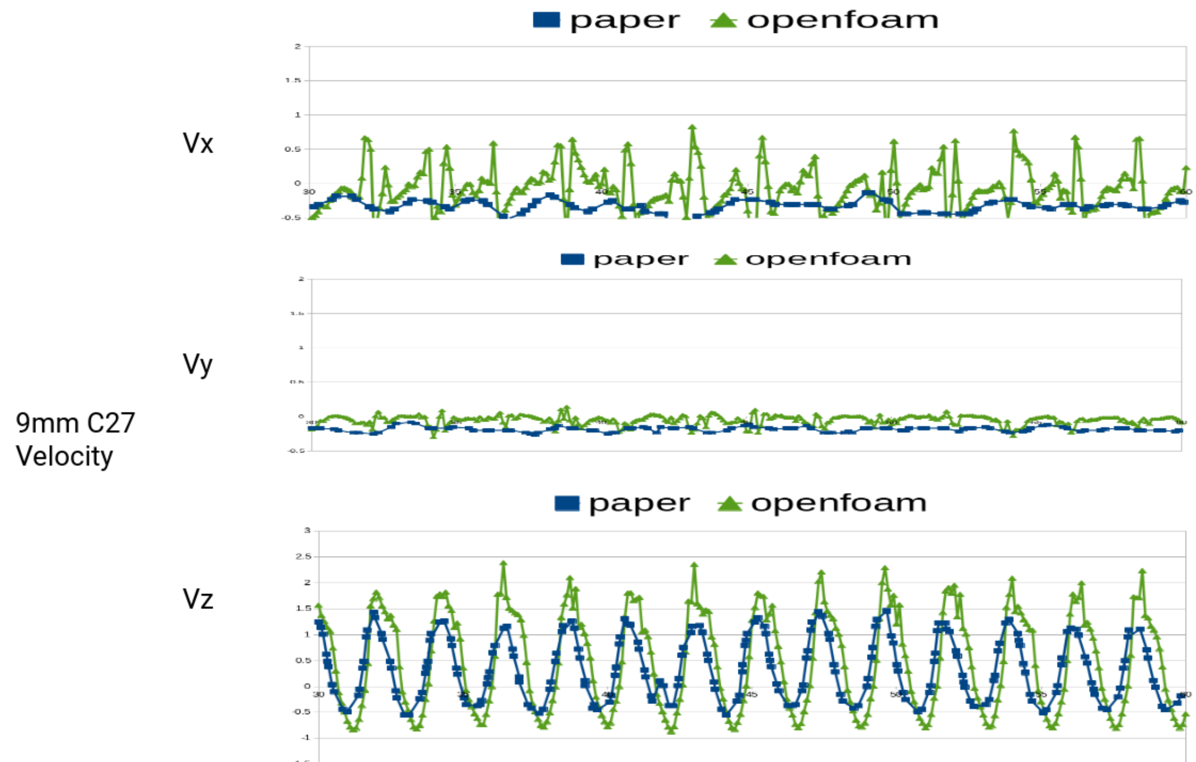
The Velocity of air at the 50mm top slot is plotted for a 1.3 s wave period 0.08 m high wave.



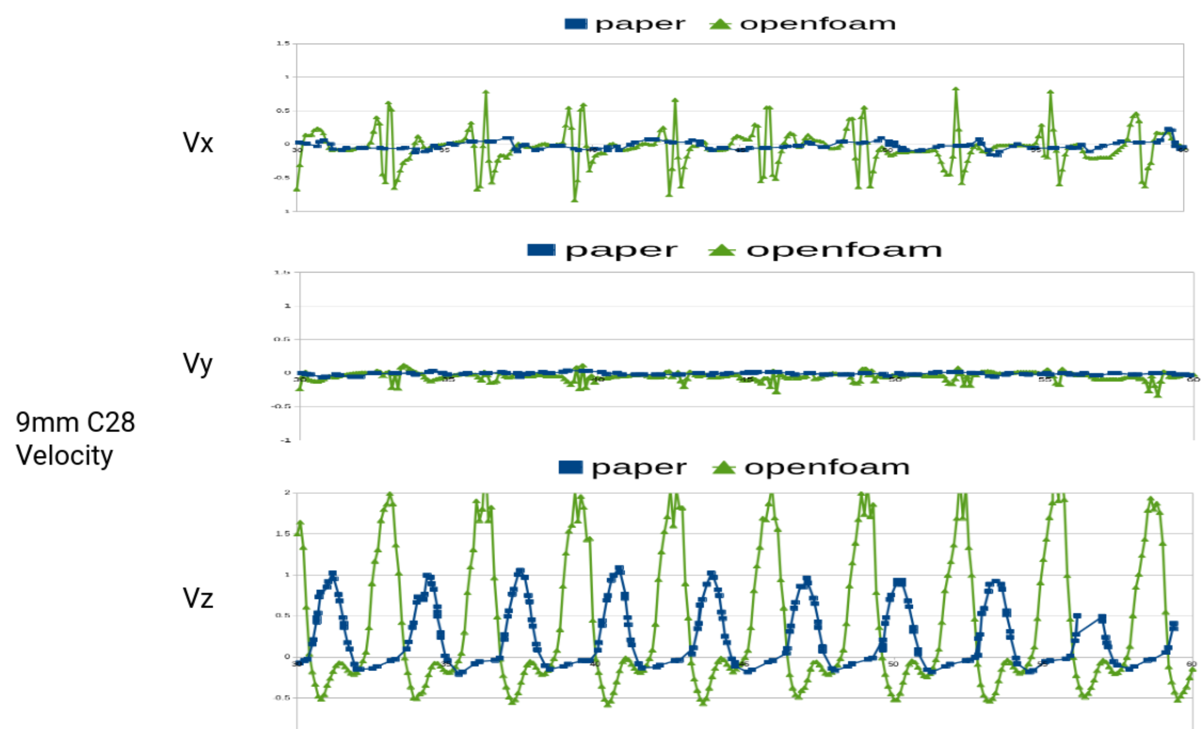
The Velocity of air at the 9mm top slot for a rectangular chamber being influenced by a 1.3 s, 0.08 m wave:



The Velocity of air at the 50mm top slot is plotted for a 2.2 s wave period 0.08 m high wave.



The Velocity of air at the 50mm top slot is plotted for a 3.2 s wave period 0.08 m high wave.



5.2 Trapezoidal and Inverted Trapezoidal Chamber

5.2.1 $50mmTopSlot$

The Analysis of pressure response of the trapezoidal geometry and the inverted trapezoidal geometry compared to the rectangular chamber with a 50 mm top slot for a 0.08m 1.3s wave yields:

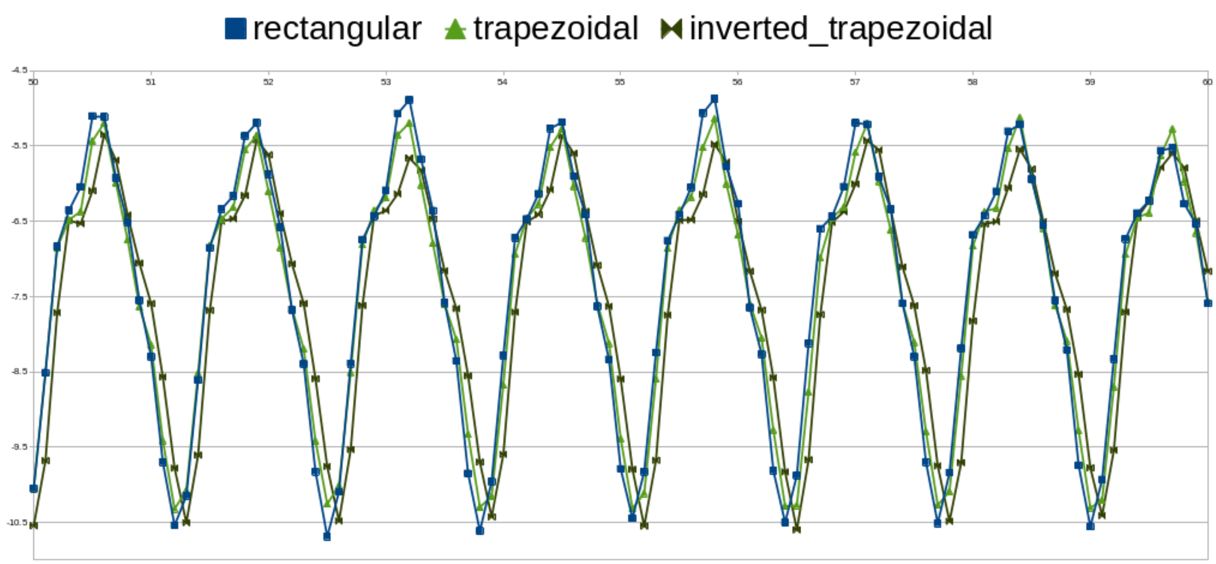


Fig. 1 - Pressure Gauge 1

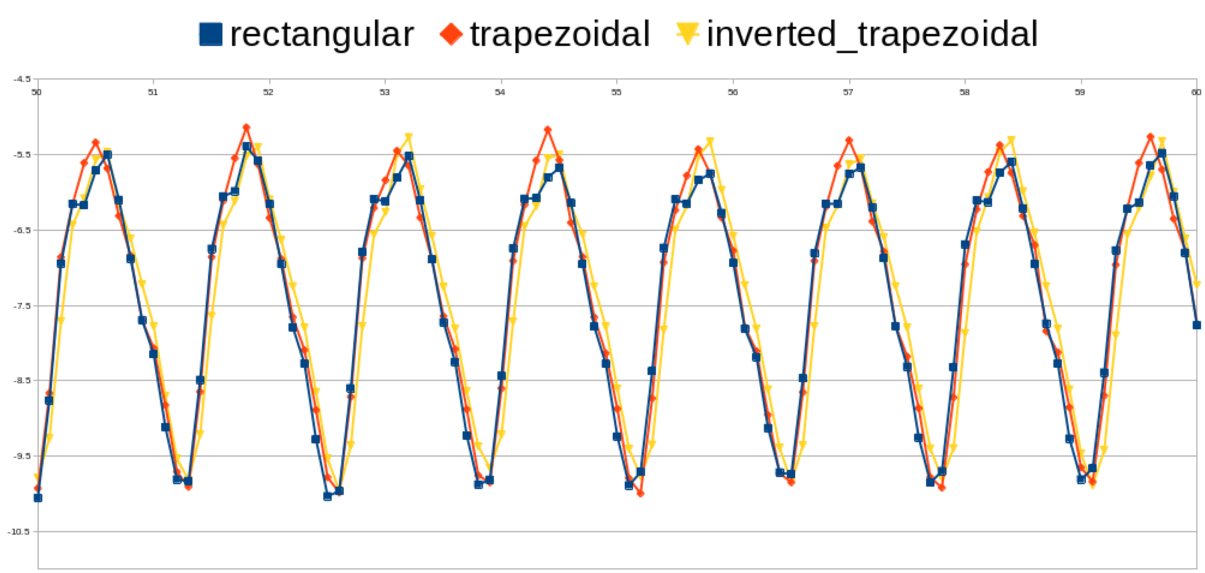


Fig.2 - Pressure Gauge 1

The Velocity response of the three geometries are found to be:

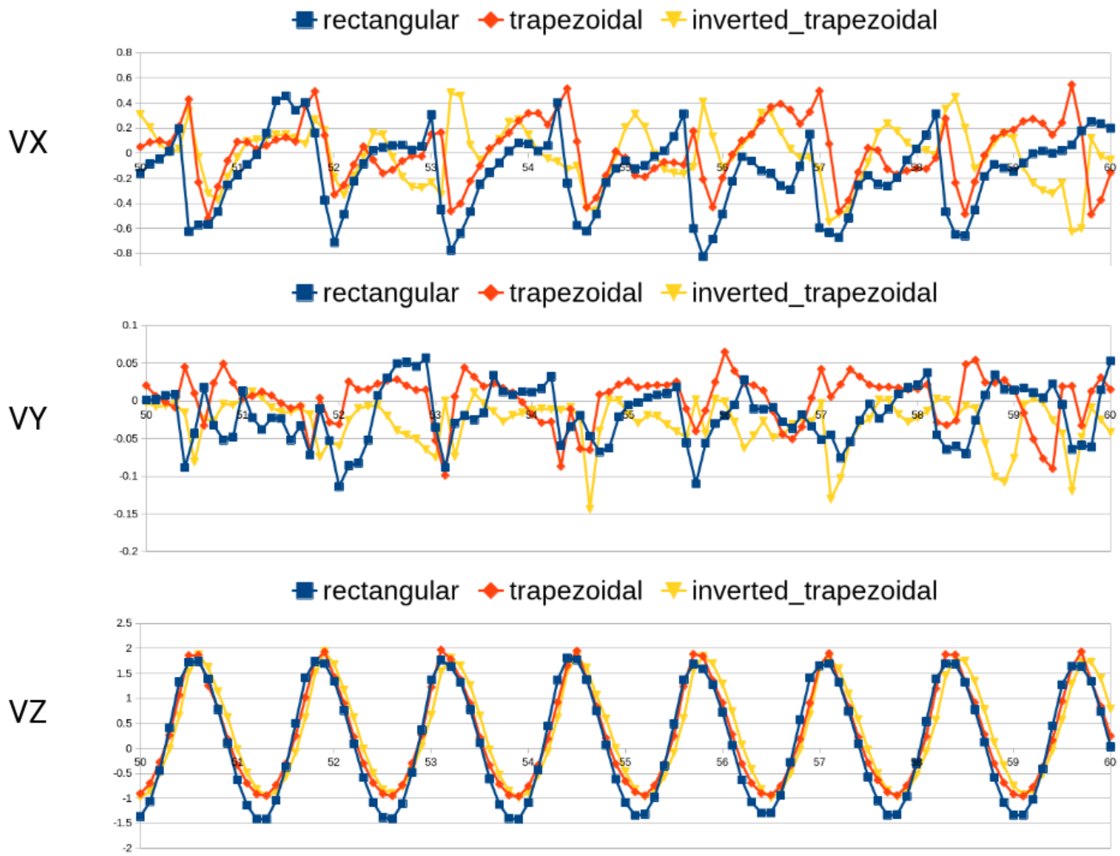


Fig.3 - Velocity Plots

5.2.2 $9mmTopSlot$

The Analysis of pressure response for a 9 mm top slot for a 0.08m 1.3s wave yields:

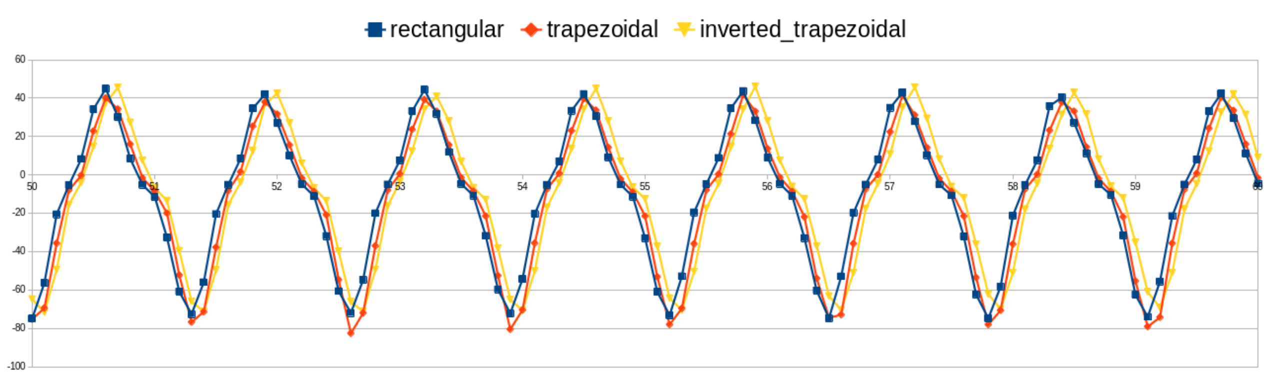


Fig. 1 - Pressure Gauge 1

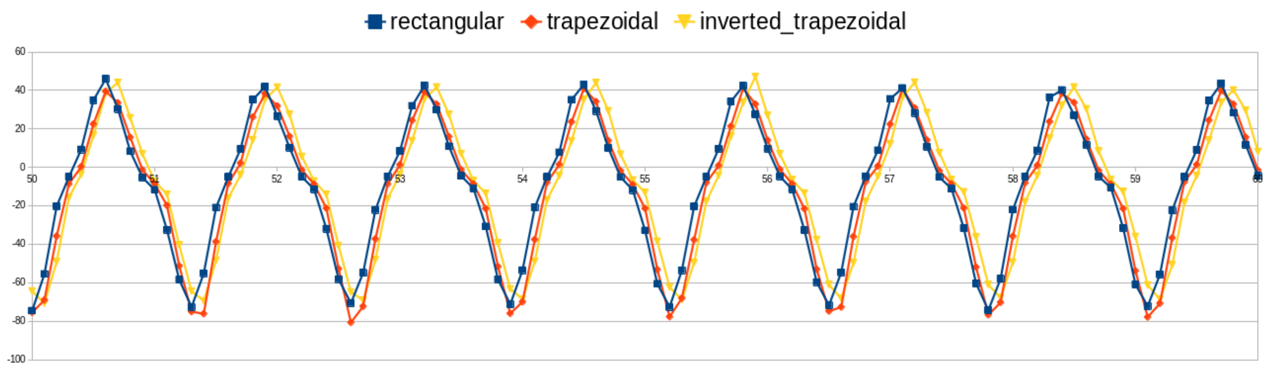


Fig.2 - Pressure Gauge 2

The Velocity response of the three geometries are found to be:

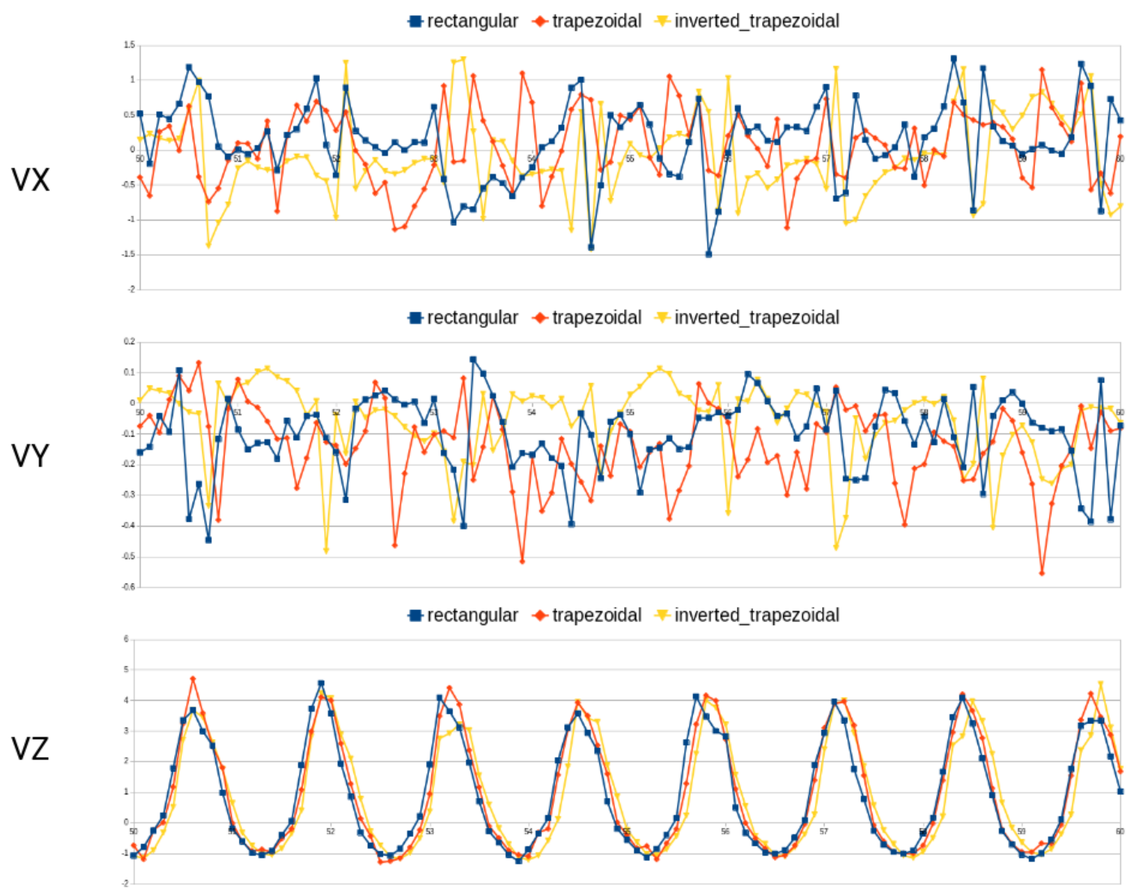


Fig.3 - Velocity Plots

The Response is seen to be closely matching in form for all three geometries but diverges slightly in magnitude for each.

Conclusion

The Results show that there is an apparent effect on the pressure and velocity due to the change in geometry and the validation with experimental results show that OpenFOAM is a useful tool to reduce costs of experimental setups and still model the effects of waves and multiphase flow interactions with ocean structures to a good degree of accuracy. The results from such a CFD analysis would need to be supplemented by a more detailed analysis before it is implemented in real scenarios. The solver as well can improved by compromising on generality to model the wave dynamics and pneumatics in the OWC chamber. The turbulence effects on the shape can also be improved upon to get more closely matching results.

Acknowledgement

The FOSSEE group of IIT Bombay is greatly acknowledged for providing the opportunity for the research and giving support through the mentors and professors who guided me throughout the project.
The author would also like to thank the mentor Ashutosh and Guiding Professor Manimaram from the Mechanical Engineering Department of VIT for providing the help that was needed and going over and above to help me with completing the research work. The Case Study has provided a lot of insight into Computational Fluid Dynamics in OpenFOAM and the ability to survey literature to prepare a Paper. I thank the FOSSEE group for providing me with the opportunity to participate and contribute to the adoption of open source software in Academia.

References

OpenFOAM User Guide :
https://www.openfoam.com/documentation/guides/latest/doc/
A.IturriozR.GuancheJ.L.LaraC.VidalI.J.Losada , Validation of OpenFOAM for Oscillating Water Column three-dimensional modeling, In: Journal of Ocean Engineering :
https://www.sciencedirect.com/science/article/pii/S0029801815003649?via