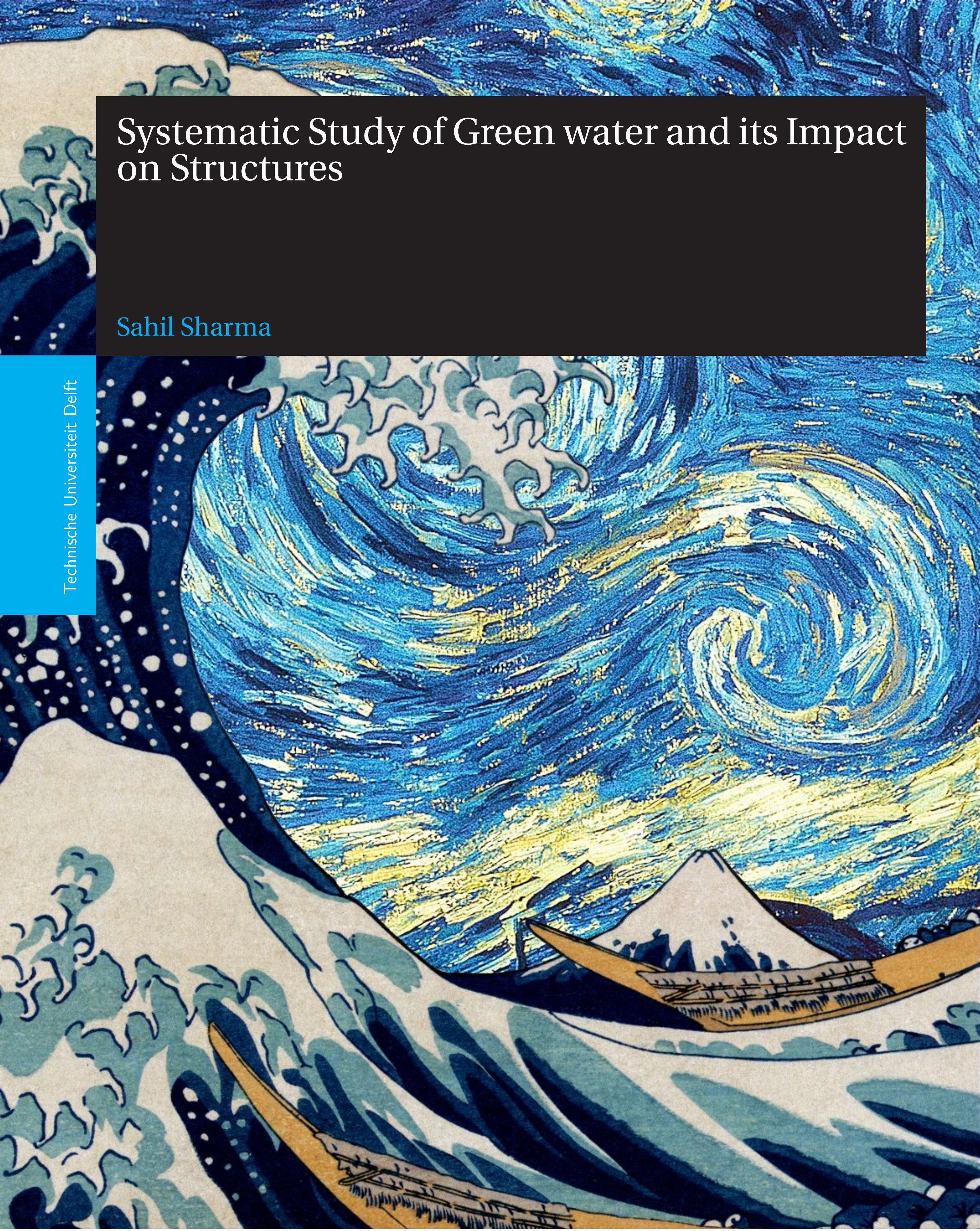
Systematic Study of Green water and its Impact on Structures

Sahil Sharma



Technische Universiteit Delft

TU Delft
Delft University of Technology
Challenge the future

" *The Cover picture has a great significance in this research. It is a combination of Vincent Van Gogh's masterpiece 'The Starry Night' and Katsushika Hokusai's 'The great wave of Kanagawa'. The Starry Night is based on van Gogh's direct observations as well as his imagination, memories, and emotions while the wave of Kanagawa is a series of painting depicting an enormous wave threatening boats off the coast of the town of Kanagawa, Japan."*

Vessels, Structures and Crew face this challenge every day. Observing and keeping in mind threatening offshore waves, this research is dedicated to the Offshore Industry around the globe.

Systematic Study of Green water and its Impact on Structures

by

Sahil Sharma
SN: 4504658

in partial fulfillment of the requirements for the degree of

Master of Science
in Offshore and Dredging Engineering

at the Delft University of Technology,
to be defended publicly on Tuesday January 30, 2018 at 10:00 AM.

Supervisor:	Dr. ir. P. R. Wellens	
Thesis committee:	Prof. dr. ir. A. P. R. van't Veer,	TU Delft
	Dr. ir. P. R. Wellens,	TU Delft
	Dr. ir. J. F. J. Pruijn,	TU Delft

ACKNOWLEDGEMENTS

This research would not be possible without the help and support of several important people that I would like to thank for helping me make this tough journey.
I would first like to thank my thesis supervisor Dr. ir. P. R. Wellens. The door to his office was always open whenever I ran into a trouble spot or had a question about my research or writing. He consistently allowed this thesis to be my own work, but always steered me in the right direction whenever he thought I needed it. I cherished every Friday meetings which have been of a great value to me and have always driven me to perform better every week. I would also like to thank Prof. dr. ir. A. P. R. van't Veer for his feedback that has greatly helped me in shaping my research in the final stages.
On a personal level, I would like to dedicate this thesis to those who have helped me get here. To Femiwa, a great friend, and person; for your patience and support that I wish I could ever repay. To my friends Alden, Jayesh, Briggs, Joost, Fredo, and Emeric. Those small and long coffee breaks with life and party talks were really refreshing. To my roommates Abhi and Novy who have been a part of this journey celebrating highs and also the lows together in Delft.
Finally, my family. Papa, Mummy, Didi, and Jiju: I owe it all to you. Thank you! Without your love and support, I could not have achieved this.

Sahil Sharma

ABSTRACT

The Offshore Industry is one of the most challenging industries. Offshore vessels must survive all weather conditions over its lifetime. In heavy storms, waves and ship motions become so large that water flows onto the deck, It is referred to as Green Water and is a serious challenge that needs to be considered in designing offshore vessels. The methods for making green water prediction are limited. More data needs to be collected to ensure safe navigation and operation.

ComFLOW is a numerical tool in the field of Offshore Engineering used to study these non-linear and complex green water phenomena. ComFLOW is a Navier-Stokes solver with Volume of Fluid(VOF) for free surface motion, It has been used for simulation of dam break flows as a model for green water flow on deck and falling objects in calm waters. In this research, it is used to generate dam-break waves along with simulations involving regular and irregular waves leading to water on deck. It is used to evaluate whether dam break model is a complete model for studying green water loading on deck. A grid convergence study for ComFLOW is conducted along with a verification of ComFLOW for 2 dimensions. Optimum grid cell size is advised for accurate and stable results in ComFLOW for 2 dimensions. Dam break simulations are performed on an idealized deck to observe changes from that of a regular domain boundary.

Wave prediction and it's impact on the topside facilities due to regular and irregular waves is the objective of this research. Absorbing boundary conditions are used to prevent wave reflections. Non-breaking waves are generated for irregular waves. Impact of water on the structure on deck is of primary importance. The wave height is an important parameter in dealing with such green water events. The role of wave height along with the distance of structure on deck is studied in this research. The input parameters such as dam height and distance of the structure from dam height are varied. Localized impact pressures are considered and changes in impact pressure are observed. A structure in regular and irregular waves is also compared with a dam break model. The main conclusions are:

- Variations in dam break loads compare well to theory.
- The dam break is not a complete model for assessing green water loads.

Contents

LIST OF FIGURES

List of Tables

Nomenclature

A	Cross sectional area
V	Volume
u	Velocity of fluid in x-direction
v	Velocity of fluid in y-direction
w	Velocity of fluid in z-direction
ρ	Density of fluid
P	Pressure
∇	Gradient operator
μ	Viscosity of fluid
F	Force
C	Wave phase operator
E	Empty cell
S	Surface cell
F	Fluid cell
F_c^b	Volume aperture
F_e^b	Volume aperture for eastern cells
F_w^b	Volume aperture for western cells
H	Wave height
H_s	Significant wave height
L	Wave length
kh	Dimensionless wave number
C^{out}	Phase velocity
ω	Natural frequency
ϕ	Phase of wave
β	Angle of incidence
k	Wave number
a	Wave amplitude
g	Gravity acceleration

1

INTRODUCTION

The main objective of this research in the field of Offshore Engineering is systematically varying the parameters of green water loading and observe how it impacts the structures present on the deck of a vessel. Numerical simulation and verification of ComFLOW are presented in this thesis.

1.1. GREEN WATER OFFSHORE

The offshore Industry is one of the most challenging industries over the years. Offshore vessels must survive all weather conditions over its lifetime. In heavy storms, the waves and ship motions become to a great extent that water flows onto the deck of the ship[MARIN] which is referred to as Green Water. Due to the colour of the sea water being green rather than blue, the term green water is widely used in contrast to water which consists of White foam and sprays and is highly complex and nonlinear phenomenon[2]. Green water generally occurs at the bow of the ship which is a common phenomenon on vessels and FPSO's during storm surges and has impact large enough to damage sensitive equipment on deck. Green Water is a serious challenge that needs to be considered in designing of offshore vessels. It has been under considerations for the operation and safety of naval merchant vessels over the years. Green water loading was the most important reason for the change in course and speed for Dutch merchant ships to avoid serious damage to the ships on their vessel deck as suggested by Tan (1969)[3]. Green water loads on offshore vessel occurs when storm surge wave exceeds the bow height and reaches the ship deck. For offshore vessels, green water loading to structures, deck plating, and topside equipment is a common occurrence. The methods for making such prediction on deck are limited. More data needs to be collected to ensure safe navigation and operation in these areas. The Discovery Channel television series "Deadliest Catch" has recorded several incidents of huge waves hitting the bow of deck [4] resulting in green water phenomena. In these heavy storms, wave and vessel motions can become so large that solid amounts of seawater flow over the deck of a ship. However, water on deck for smaller vessels is mostly related to the ship stability and ship motions. From the year 1995 to 2002, seventeen green water incidents have been identified on twelve FPSOs in the North Sea, UK. Some FPSOs have experienced the green water loading more than once as reported by Morris, Millar, and Buchner[5]. In January 2000, the bow of Varg FPSO which was located in the Norwegian North Sea was severely hit by green water, damaging the window on the second floor of living quarters on a bow of the ship resulting in flooding of the area [Ersdal and Kvitrud][6]. There have been accidents causing losses to 200 carrier vessels in the past 20 years which have been reported by W. Rosenthal.

Figure 1.1: Green water on a ship offshore (Source:www.restnova.com)

ComFLOW is a numerical tool in the field of Offshore Engineering used to study these non-linear and complex phenomena. Interface tracking and interface-capturing methods are used for simulation of surface problems such as dam breaking, tank sloshing, wet deck slamming and green water on ship decks. Boundary-fitted grids are adjusted with each time step whenever the surface moves, which is used for the interface-tracking method for free surface motions[7]. In divergence, the interface capturing methods do not define a sharp free surface boundary. Rather, computation is performed on a fixed grid, which is extended beyond the free surface and the shape of this surface is determined by cells that are partially filled. The most commonly used interface capturing methods are marker and cell (MAC) method (Harlow and Welch,1965)[8], volume-of-fluid (VOF) method (Hirt and Nichols,1981)[9], and level-set method (Osher and Sethian, 1988)[10]. The interface capturing level set method is employed in concurrence with the finite analytic Navier Stokes (FANS) method for the time domain simulation of the storm surge wave impact loads on a fixed platform under monochromatic, 3D short crested and 2D long-crested waves in a recent study by Chen (2010,2011)[11].
From the literature regarding ComFLOW, 3D numerical models are based on averaged Navier-Stokes equations integrated by using Eulerian methods, Smoothed Particle Hydrodynamics (SPH) Lagrangian methods or hybrid Eulerian-Lagrangian methods. Navier-Stokes/Continuity equations for an incompressible and viscous fluid which can be solved as a Cartesian grid with staggered variables using ComFLOW. The software ComFLOW discretizes the equations in space and time using the finite VOF (Volume of Fluid) method introduced by Hirt and Nichols,1981[9]. ComFLOW has been previously used for applications such as sloshing on board tumbling spacecraft[12] and blood flowing through arteries[13]. Whereas in Offshore Industry it is used for sloshing in anti-roll tanks, simulation of dam break flows as a model for green water flow on deck and falling objects in calm waters[1].

1.2. LITERATURE REGARDING GREEN WATER

Green water has been a subject of interest since early days of research in the ship behavior in the 19th Century. The work of Newton is the first-ever recorded systematic study in this field which goes as early as 1959 and involved five different bow forms with regular head waves. His work was mainly based on visual observations of deck wetness which was labeled as "dry", "wet" and "very wet"[14]. To analyze the deck wetness incident wave elevation and relative ship motion are the main parameters involved. The basic model was constructed to identify the role of these parameters. The best way to calculate the occurrence of green water on deck is to evaluate the relative vertical motions by linear wave theory. O'Dea and Walden(1984)[15] carried out the experiments involving the regular head waves by a model which was in above-water bow shape helping them calculate relative motion near the bow and deck wetness. Lloyd (1985) carried out the similar experiment but in the irregular head waves. He altered the design of the model and presented the results in terms of bow wave height, impact frequency and swell up coefficient. Buchner(1995) was the one to study the phenomenon of green water loading on Floating Production Storage and Offloading (FPSO) units. He investigated the relationship between relative ship motions and deck wetness and also carried out the study of water impact phenomenon with structures on deck[16].

All the studies involved in this field of research is important to understand the phenomenon of green water, however many assumptions were made in the past which does not provide concrete results. Research done by Watanabe (1990) concludes that increased bow flare form reduces relative bow motion and deck wetness[17]. He achieved this result by conducting an experiment for S-175 container ship at forward speed in which he increased the bow flare of the original model. Buchner observed the similarity of the flow on the deck during shipping of water to the one like dam breaking wave which was important in assessing the structural effects due to the water on deck. Greco (2001)[18] studied the experimental and numerical approach for water on deck on a 2 dimensional ship model via a plunging breaker wave input. Vertical bow and stem angles of plus and minus 45 degrees were studied which concluded that dam breaking model gives a qualitative description of what exactly happens to the ship motions when the wave hits the deck. Standsberg and Karlsen (2001)[19] observed the water flow on the deck of floating production, storage and offloading unit (FPSO) in steep random waves which has non linear effects. Only few cases resembled the dam-breaking type flow and were mostly the result of large negative bow motion with small initial horizontal fluid velocities. Dam-break behavior was absent where the kinematics of incident waves caused increase in shipping of water onto the deck. Greco (2001)[18] combined the non-linear two-dimensional potential flow simulations with two-dimensional experiments. Water shipping on the deck was studied using the idealized rectangular-shaped structure and was investigated experimentally and numerically. Also the the wave propagation with impact on the structure was studied by full numerical simulations and measurements. The main observation of this research is the behavior of pressure peak. The pressure peak is the impact of water on the structure. Series of pressure peak were studied and rise of the pressures were detected. Kleefsman[1] studied the water impact by dam break wave numerically in ComFLOW which is based on Navier Stokes equations that represent the flow as in-compressible viscous fluid. Water entry problems related to dam break were also investigated by dropping wedges with different angles, a cone and cylinder into calm water with definite velocity. The simulations were then compared to the experiments conducted. The convergence between the simulations and experiments was concluded.
The practical approach to this research is motivated by the need to study and understand the green water loading and their impact on the structures and alleviate the green water accidents and damages on offshore vessels which in offshore location are weather-vaning and head sea represents the most severe conditions in terms of green water and deck wetness events. Conclusion of two most important literature study research which has great significance in this research can be shortly summarized as below:

1. Greco [18] studied shipping of water on deck in regular waves of various forms. Impact pressure on a vertical wall at a distance from the bow was studied. It was concluded that dam break models provide a qualitative description of ship motions when a wave hits the deck.

2. Kleefsman[1] studied the dam break models in ComFLOW software and observed the convergence of pressure peaks obtained from simulations with experiments.

1.3. PROBLEM DEFINITION

Waves can induce large forces and stresses on and near hydraulic structures on vessels. The hydrodynamic load is responsible for unwanted effects such as vibrations, fatigue, and damage to the structures.The safety of the structures, substructures, operability and also the safety of the crew on these vessels indicates the main issue while dealing with the green water loading. The research on green water loading has been significantly increased and carried out to minimize the accidents and increase the safety of the crew and equipment on deck. Research has been previously conducted on green water as dam break models and observation of green water on offshore vessels in regular waves using numerical tools and experiments. However, green water phenomenon caused by waves is far from regular. It is totally irregular in offshore conditions which have not been studied yet. The wave height is an important parameter in dealing with such green water events. The role of wave height along with the distance of structure on deck will be studied in this research.
To study this phenomenon which involves green water and wave impact, ComFLOW will be used. It is an advancement of the SAFE-Flow project with improved Volume of Fluid(VOF) and better codes for Computational Fluid Dynamics (CFD) to compute wave loads on offshore structures. ComFLOW focuses on the development and validation of complex free surface turbulent flows in the offshore industry with improved functionality and speed-up of the algorithms [20]. The performance of ComFLOW is not validated in the field of Green water from the sides. This research will also focus on verification of ComFLOW in 2-Dimensional cases of green water phenomenon on offshore vessels.

1.4. RESEARCH OBJECTIVE

There has been a significant research in the field of Offshore engineering for Green water loading in fixed and floating platforms and vessels over the years on regular waves. However, the lack of research in predicting wave and their impact on structures kept on the topside facilities due to regular and irregular waves is what the main objective. This research will focus mainly on the ships suffering from green water for 2-Dimensional loading in far offshore damaging the expensive units kept on deck generated from the wave causing an impact. In this research, we can try to predict the pressure from the resulting regular and irregular sea state on the deck equipment by systematically varying the parameters of an incoming wave and the distance of the structure within the deck. To achieve this, several considerations are made which includes setting up the boundary conditions, wave height, time period, distance of the structure from the initial wave, grid size, wave height etc. which is explained in the chapters further. ComFLOW offers the numerical simulation of the fluid on the structure. This is important since the simulations need to be unblemished before it can be validated. The structure kept on deck is restrained from moving. Therefore, the effects of wave-induced motion in the structure is not considered. However, simulations performed are done with realistic wave height relative to the deck to study the realistic green water phenomena. Dam break wave is used as a schematic model to represent green water phenomenon on deck, this research will also focus on an investigation of idealized dam break model as realistic and complete model for green water on vessel deck in regular and irregular waves.

1.5. PROPOSED METHODOLOGY

This research focuses on studying the Greenwater loading generated by regular and irregular sea state and systematically varying the parameters to observe the behavior of the incoming wave on the structures kept on deck and thus allowing us to also focus on the design parameters of the deck to sustain impact wave pressure. Numerical methods are used to determine extreme wave impact hydrodynamics against these structures. The equations are obtained using the finite volume of fluid method on a fixed Cartesian grid. ComFLOW is the software used to simulate these fluid flow onto the structure on idealized fixed deck since it offers better algorithms for the wave propagation. It is the primary focus of this research because the wave impact pressure needs to be considered in the design stages of the ship to reduce the damages.

1.5.1. BREAKDOWN OF PROPOSED METHODOLOGY

To achieve the Research Objective, the whole Methodology is divided into small procedures to finally achieve the result. Series of chapters will be conducted and are briefly described in the following:

1. **Grid Independence**: The ComFLOW will be tested for results where the grid resolution no longer participates in the accuracy of results obtained from the simulations. Based on Grid independence study, grid size will be selected for conducting all numerical simulations for research.

2. **Verification of ComFLOW**: The ComFLOW is not verified for different cases of dam break waves. In this research, it will be compared with Kleefsmann's 3Dimensional simulations with 2Dimensional cases for dam break waves in different grid resolution.

3. **Dam Break Simulation**: The green water phenomenon will be considered as dam break model. The fixed structure will be placed in the domain which will act as a structure on the deck of a vessel offshore. Pressure impact from dam break wave will be monitored.

4. **Systematic Variation**: Two different types of variations in parameter will be conducted. 1. The wave height of the dam break wave will be varied and the pressure generated from different wave heights on a structure will be observed. The relation between Pressure generated vs. Wave height will be observed. 2. The distance of the structure on deck will be varied from the incoming dam break wave. The pressure generated from this wave will be monitored and a relation between the distance of structure will be observed i.e. Impact pressure vs. Distance.

5. **Structure in Regular waves**: The pontoon like structure will be placed in the domain, regular waves of different wave heights and time period will be generated to observe significant green water events and respective pressure generated upon impact on the topside structure. The distance of topside structure on pontoon will also be varied to observe the behavior of pressure generated by green water loading at different locations.

6. **Structure in Irregular waves**: The pontoon like structure will be placed in the domain, non breaking irregular waves of different wave heights and time period will be generated to observe significant green water events and respective pressure generated upon impact on the topside structure. The distance of topside structure on pontoon will also be varied to observe the behavior of pressure generated by green water events at different locations. It will finally be concluded if dam break is an ideal and complete model to study green water for regular and irregular waves.

2

Governing Equations and Simulation in ComFLOW-3

2.1. Governing Equations for Fluid flow

The dam break flow in ComFLOW is described by the continuity and Navier-Stokes equations which describe the conservation of mass and momentum respectively[1]. They are represented in eq 2.1 and 2.2:

$$\oint_{\partial V} \mathbf{u}\cdot\mathbf{n}\, dS = 0, \tag{2.1}$$

$$\int_V \frac{\partial u}{\partial t}\mathrm{d}V + \oint_{\partial V} \mathbf{u}\mathbf{u}^T\cdot\mathbf{n}\mathrm{d}S = -\frac{1}{\rho}\oint_{\partial V}(p\mathbf{n} - \mu\nabla\mathbf{u}\cdot\mathbf{n})\,\mathrm{d}S + \int_V \mathbf{F}\,\mathrm{d}V \tag{2.2}$$

In the above equations, ∂V is the boundary of volume V, $\mathbf{u} = (u, v, w)$ is the velocity of the fluid in x, y, and z coordinate directions, ρ is the density, p is the pressure and ∇ is the gradient operator. $\mathbf{n}$ is the normal at the boundary ∂V. In addition, μ represents the dynamic viscosity and $\mathbf{F} = (F_x, F_y, F_z)$ is the external force.

2.2. Boundary conditions

Two types of boundary conditions are used in this research, No-slip boundary conditions for dam break simulations while non-reflecting boundary conditions for regular and irregular wave simulations. No slip boundary conditions are used for the object inside the domain and the solid walls. They are described by $\mathbf{u_n}$= 0 and $\frac{\partial \mathbf{u_n}}{\partial \mathbf{n}}$ for fixed boundaries. Since there is no moving object in the domain, $\mathbf{u}_n$ is 0 for fixed boundaries.

For wave simulations, boundary condition is needed which allows the fluid flow in and out of the domain. The fluid flows in during the simulation when the incoming wave is directed while a non-reflecting outflow condition should be used for the opposite boundary. In our simulation the incoming wave is described as a regular linear wave or a regular Stokes wave of the 5th order[21]. Open boundary conditions involve flow over obstacles, wake instability flow etc.[22] This research is focused on problems related to wave propagation and the resulting disturbances that are generated in the domain. The equation is:

$$\frac{\partial \phi}{\partial t} + C\frac{\partial \phi}{\partial x} = 0 \tag{2.3}$$

Here, ϕ is any variable and C is the wave phase velocity (Velocity of the incoming wave).

2.3. FREE SURFACE

The equation for conservation of mass and momentum is considered. The interface between water and air is called the free surface [23]. which is the phenomenon that is considered during the lifetime of a vessel in far offshore. $s(\mathbf{x}, t) = 0$ is the actual position of the free surface, while the displacement of this free surface can be represented in the equations:

$$\frac{\mathrm{D}s}{\mathrm{D}t} = \frac{\partial s}{\partial t} + (\mathbf{u} \cdot \nabla)s = 0 \tag{2.4}$$

If the water is considered to be in-compressible, $\nabla \cdot \mathbf{u} = 0$ and can be rewritten as:

$$\frac{\mathrm{D}s}{\mathrm{D}t} = \frac{\partial s}{\partial t} + \nabla(\mathbf{u} \cdot s) = 0 \tag{2.5}$$

2.4. NUMERICAL MODEL

Cell Labelling: The numerical simulation domain is distributed in Cartesian grid cells. These cells are labeled to identify the type of equation used. The velocity components are assigned on cell boundaries while pressures are defined in cell centers. Structured grids are preferred to unstructured grids. A structured grid is straightforward and higher order discretisation can be achieved. However, for a small grid distance is required for accuracy at a particular location where large gradients are expected. Grid cells in structured grid are more compared to un-structured grid. Based on the details, geometry labels are assigned to the cells that describe its kind. For fluid, a boundary (B) or an exterior (X) cell. Fluid cells are labeled as empty cells (E), surface cells (S) and fluid cells (F) to describe free surface[23].The velocity between an F-cell and S-cell is called as FS-velocity. The labeling can be further simplified in figure 2.1.

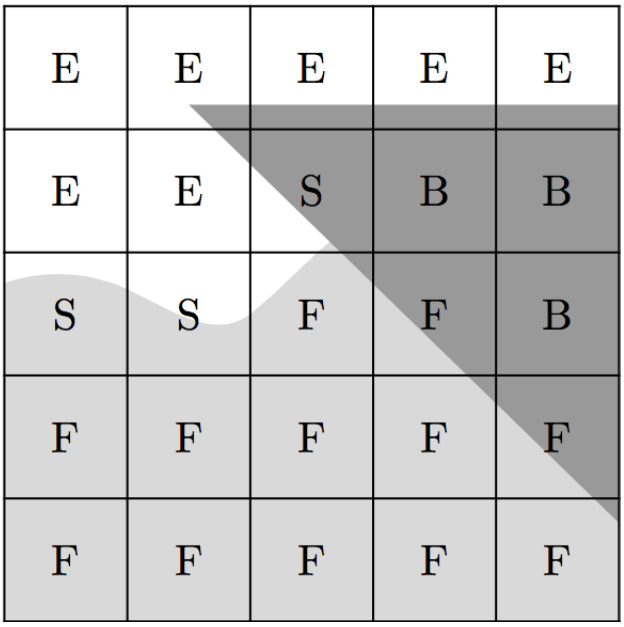


Figure 2.1: Cell labeling: Boundary, empty, surface and full cells. The dark grey area denotes the ship, shaded area denotes the water and white area represents air [1]

2.4.1. DISCRETISATION OF NAVIER-STOKES EQUATIONS

Navier-Stokes equations are discretised in space and time. Discretisation is carried out in a control volume around a velocity that is defined at a cell face. For uncut cells, the control volume consists of half of neighboring cells to left and right of velocity. In case of cut cells, control volumes are defined as half of open part of a left neighboring cell and half of open part of the right neighboring cell. The discretisation can be explained for momentum equation in the x-direction. [1]

The volume integral of time derivative can be discretised in space using the midpoint rule which is expressed as:

$$\int_v \frac{\partial u}{\partial t} \mathrm{d}V = \frac{\partial u_c}{\partial t} F_c^b \delta x_c \delta y. \tag{2.6}$$

Here, u_c is the central velocity around which the control volume is placed and $F_c^b \delta x_c \delta y$ is the volume of the control volume. The volume aperture F_c^b of the control volume is defined as $F_c^b = \frac{1}{2}(F_e^b + F_w^b)$ with F_e^b and F_w^b is the volume aperture for the eastern and western cells respectively[23]. It is explained in figure 2.2.

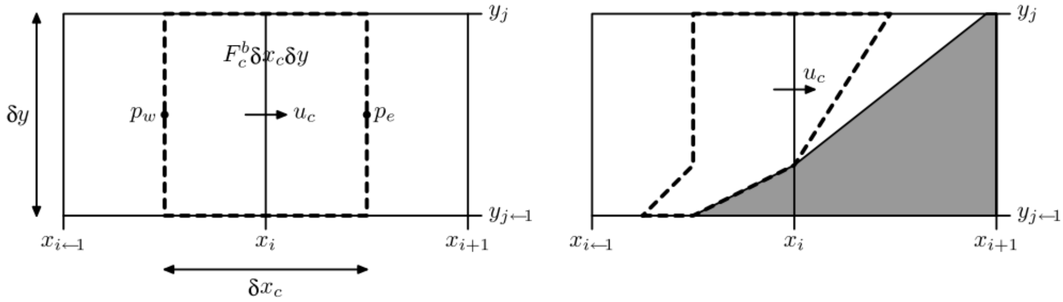


Figure 2.2: Discretisation of Navier Stokes equation in x-direction for uncut (left) and cut cell (right)

2.4.2. DISCRETE EQUATIONS

Finite volume of fluid method is used to discretise the continuity and momentum equations. The conservative formula is given by the governing equations in equation 2.1 and 2.2.

The discrete continuity equation is represented as:

$$u_e A_e^x \delta y + u_n A_n^y \delta x - u_W A_W^x \delta y - v_s A_s^y \delta x + u_b (A_e^x - A_W^x) \delta y + v_b (A_n^y - A_s^y) \delta x) = 0. \tag{2.7}$$

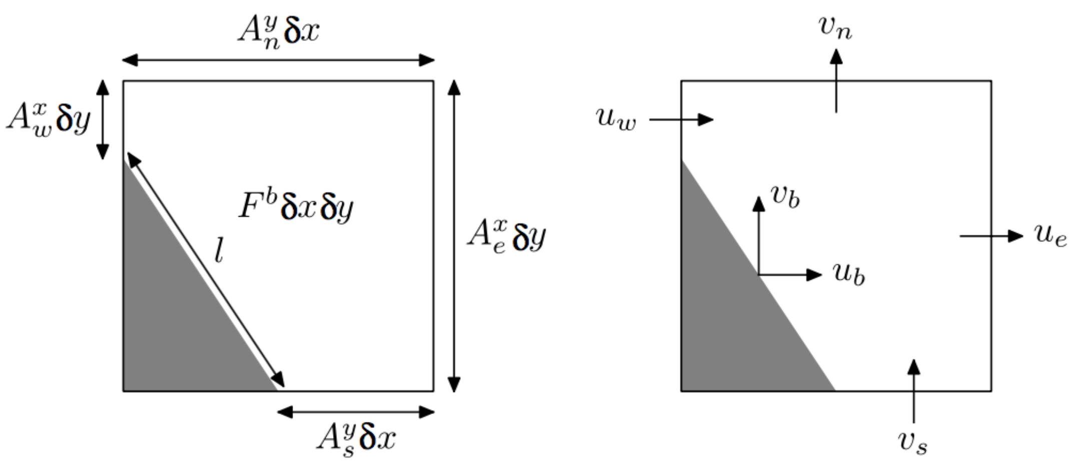


Figure 2.3: Conservation cell of the continuity equation

Momentum equations are represented as:

$$u_h^{n+1} = \tilde{u}_h + \delta t \Omega^{-1} \frac{1}{\rho} (M^0)^T p_h^{n+1} \tag{2.8}$$

where

$$\tilde{u}_h^n = u_h^n - \delta t \Omega^{-1} (C(u_h^n) u_h^n - \frac{\mu}{\rho} D u_h^n - F_h^n) \tag{2.9}$$

Poisson equation for pressure is given:

$$M^0 \Omega^{-1} (M^0)^T p_h^{n+1} = \frac{\rho}{\delta t} (M^0 \tilde{u}_h^n + M^b u_b^{n+1}) \tag{2.10}$$

2.5. SIMULATION IN COMFLOW

The ComFLOW domain is where the simulation is performed, it is always the box type structure which makes it easy to define the geometry and boundaries of the substructure and easier to define the grids. The Volume of fluid method and the domain definition is defined on cut-cell method in which the boundaries of the object are kept sharp. The numerical convergence of the grids is checked by performing simulations which are kept constant in space and time. The simulation involves dam break flow in a box with the structure to observe and analyze the impact of dam break wave onto the structure. The simulation performed is the model for the event of Green water loading onto the deck of the vessel. The created domain is 5m long, 1m wide and 2m high while an incoming wave of 1m height is considered. The structure is kept at a distance of 2.5m from the domain in x-direction or 1.5m from dam break wave to achieve grid independence.

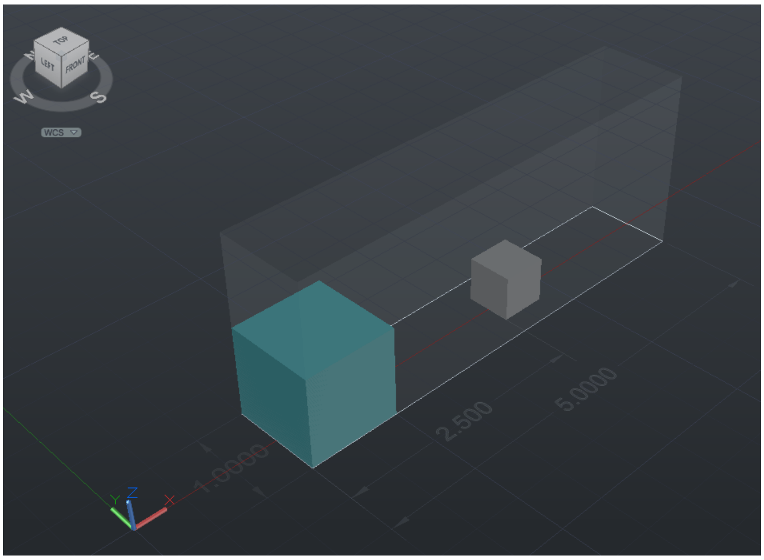


Figure 2.4: 3D isometric view of the ComFLOW domain

2.5.1. GRID INDEPENDENCE STUDY

To observe the numerical convergence of grid resolution in ComFLOW, Dam break simulations for different grid resolutions are performed to check for results where grid resolution no longer affects the values of impact pressure and force on the structure. For Grid refinement, grid cell parameters are defined in x,y, and z direction. These are the $imax$, $jmax$ and $kmax$ in ComFLOW respectively. Since we are only interested in dam-break waves in 2 dimensions, $jmax = 1$ i.e. in the y-direction.

Thus, simulation is performed in x-z plane. To attain accurate results, $\frac{\Delta Z}{\Delta X} = 1$ as a grid parameter is used. The simulations are performed for various Grid sizes. These being imax = 50, 100, 200, 250, 300, 350, 400, 500, 600, 700 and 800. The pressure is monitored by placing the monitor point in front of the designed structure.

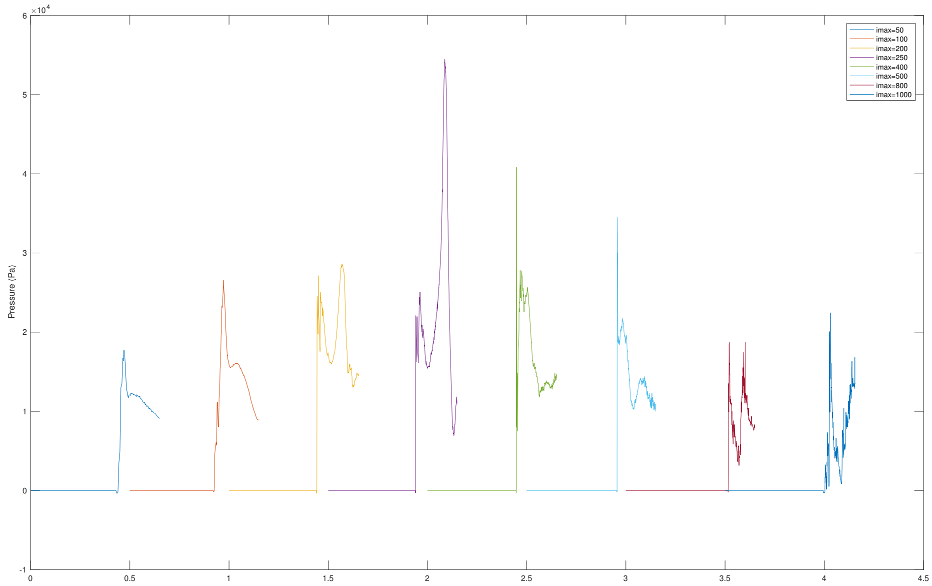


Figure 2.5: Grid convergence for Pressure impact for different grid sizes with impact time shift for clarity

Figure 2.5 represents simulations of pressure impact for different grid cells. The time duration of simulation is reduced from 5s to 0.65s to closely indicate the values of impact pressure from dam break wave on the structure. It can be observed that for grid cells 0.00625m and 0.005m which is imax= 800 and 1000, the pressure peak is approximately the same but not a concrete conclusion can be drawn since the convergence is failed. The sharp peaks can be ignored which are caused due to irregularities in ComFLOW.

To Further check for convergence, values from force box are also observed to study grid independence. The force exerted by dam break wave upon impact are also observed for grid resolution of imax = 50, 100, 200, 250, 400, 500, 800, 900, 950, 1000 and 1050. It can be seen in figure 2.6 that for last two grid cells of imax = 1000 and 1050 convergence is not achieved. Also, high peaks of forces can also be observed which has to be neglected. This is due to the numerical issues in ComFLOW and cannot be trusted. Since ComFLOW is a numerical tool, the accuracy of simulations should improve as the grid resolution is refined. However, it is not the case observed from these dam break simulations which indicate abnormal trend in the grid independence which is not expected. Thus, No concrete conclusions can be drawn from these results for either pressure and force calculations.

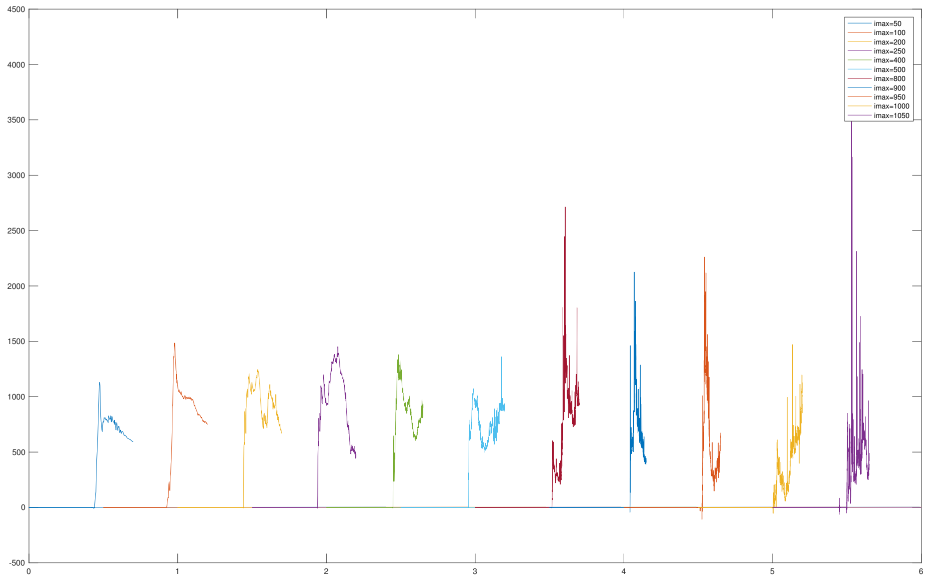


Figure 2.6: Grid convergence of Forces for different grid sizes with impact time shift for clarity

2.6. CONCLUSION

After detecting uncertainties in ComFLOW software for grid cells of a size smaller than 0.05m, higher pressure and force peaks are observed which is not reliable anymore. And since, the convergence in grid resolution is not detected from grid independence study for both pressure and force figures, verification of ComFLOW needs to be conducted with Kleefsman's numerical simulations to obtain optimum grid cell size to perform a further study in this research.

3

VERIFICATION OF COMFLOW

After performing the grid independence study, it is concluded that ComFLOW does not offer grid convergence. Thus, totally different and new method is elected which focuses on verification of simulations performed by Kleefsman [1] in 3 dimensions on dam-break waves with same grid resolution with 2 dimensional simulations to attain optimum grid resolution which can be used to further to study regular and irregular waves on the structure.

3.1. EXPERIMENTAL SETUP BY KLEEFSMAN

Kleefsman performed experiments at Maritime Research Institute Netherlands(MARIN) for dam break waves on a structure. The structure was placed in a domain and seen as a scale model of container on the deck of a ship (Issa et al. 2006). The pressure was observed from these plunging waves as a function of time. The domain consisted of specific dimensions 3.22 x 1 x 1 meters.

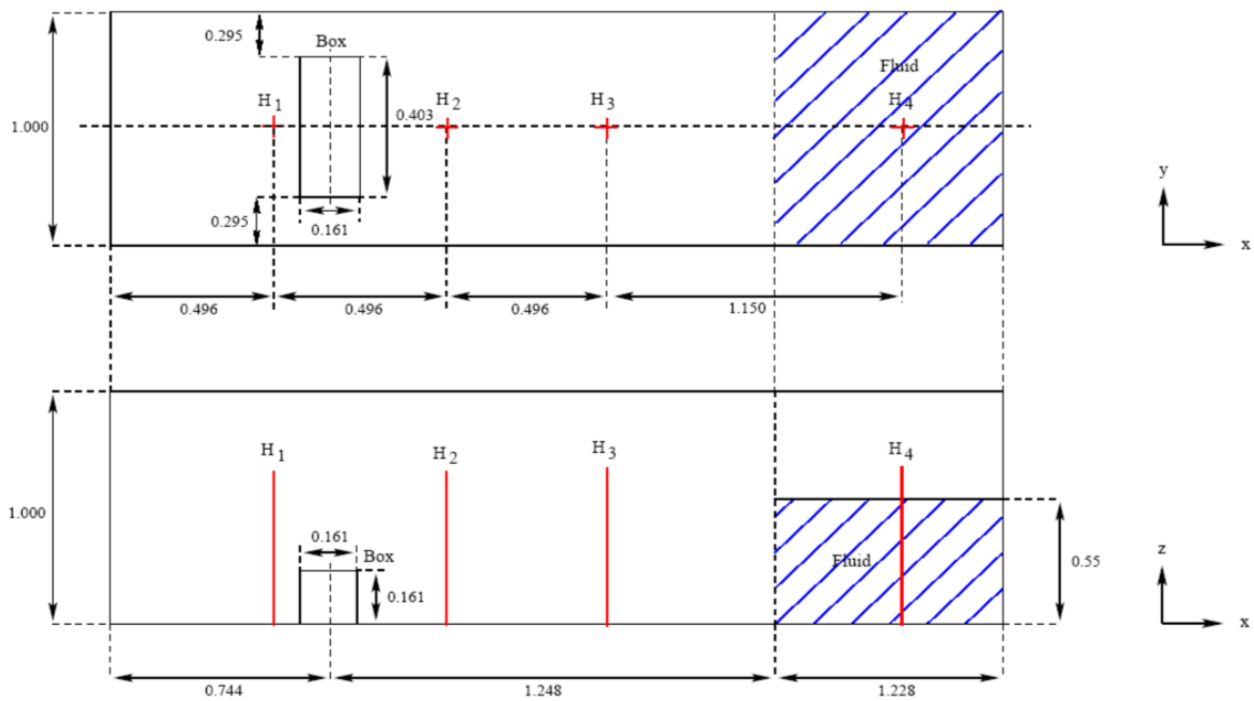


Figure 3.1: Top and side view of experiment conducted at MARIN

Dam break wave of 0.55m high and 1.22m wide was simulated on the structure which is idealized as green water phenomenon. This case is used to validate ComFLOW. The figure represents top and side view of an experiment conducted. All dimensions are in meters. The structure is placed at 1.1675m from the dam break wave column. She included 4 wave gauges represented by $H_1, H_2, H_3, and H_4$. Fluid represents dam break column. Detailed descriptions were also provided on pressure gauges placed on the structure to monitor exerted pressure.

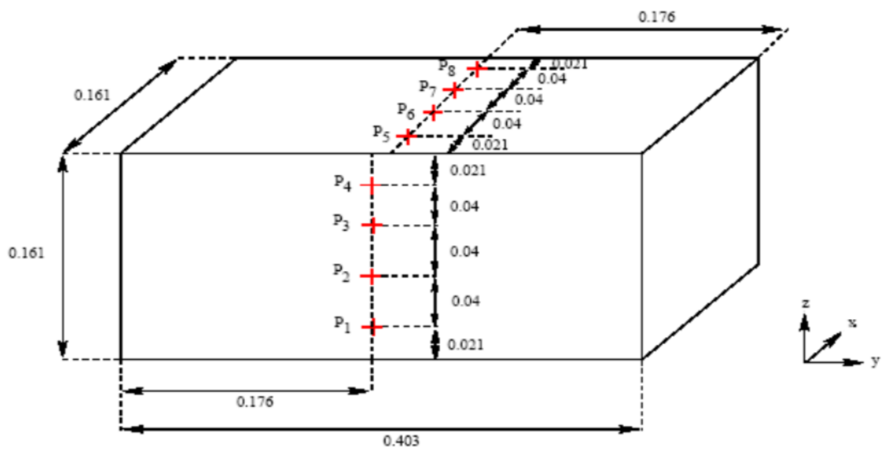


Figure 3.2: Position of pressure gauges on structure

8 pressure gauges were monitored by Kleefsman in her experiment to monitor and validate her numerical simulations with experiment.

3.2. KLEEFSMAN COMFLOW SIMULATION

The same exact experiment is generated in ComFLOW software with the same exact specifications and results were compared with each other by Kleefsman. For this research, both numerical simulations and experimental data are of interest. Verification of ComFLOW in two dimensions is not been done yet. The numerical simulations performed by Kleefsman in 3 dimensions will be used to validate ComFLOW in 2 dimensions because the simulations performed in 2 dimensions reduces the computational time. Exactly the same scenario is generated in ComFLOW which was used by Kleefsman for her experiments at MARIN.
Three different grids mesh sizes have been used by Kleefsman, in increasing order of $59 \cdot 19 \cdot 17$, $118 \cdot 38 \cdot 34$ and $236 \cdot 76 \cdot 68$ grid points. The last one being the finest of them all. However, it was claimed that the coarse grids are not good enough when zooming in to observe flow behavior. Simulations performed by Kleefsman were for 6s real time. However, we are just interested in impact pressure from water thus duration was reduced to 1s real time. The Pressure results obtained from pressure point P_1 are discussed below.
In figure 3.3, blue pressure line indicates grid resolution of $59 \cdot 19 \cdot 17$, while the red line represents $118 \cdot 38 \cdot 34$ grid resolution and the black line shows finest grid resolution of $236 \cdot 76 \cdot 68$. It is observed that there is a delay in impact peak pressure for two coarse grid sizes. Convergence is not achieved for these three cases but results for finest grid i.e. $236 \cdot 76 \cdot 68$ is closest to the experimental result.

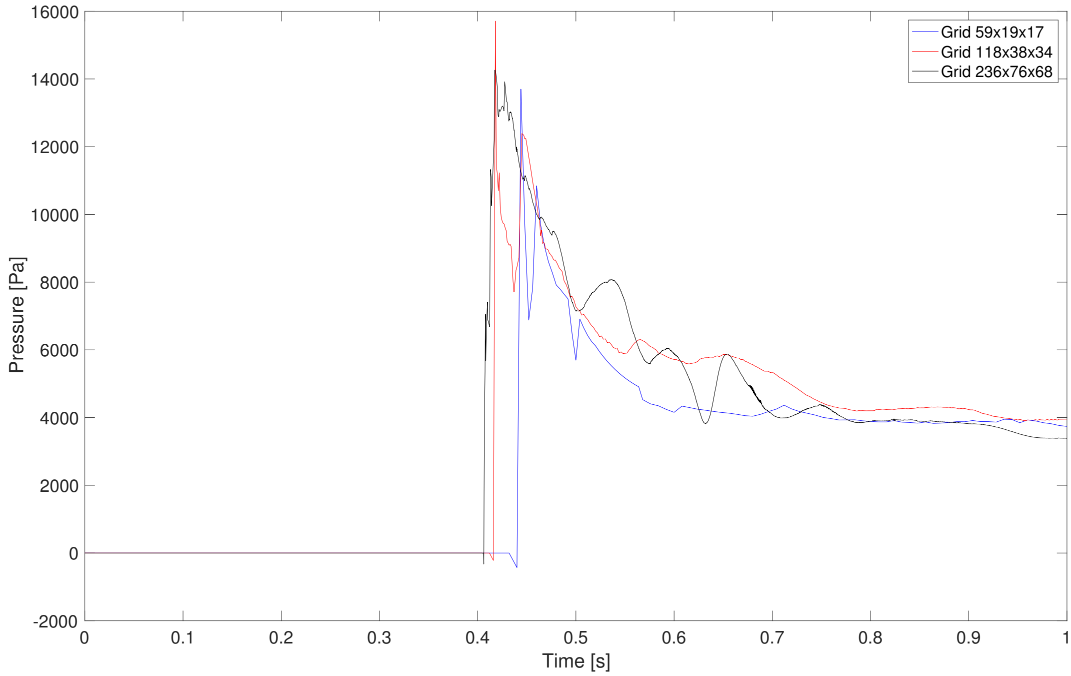


Figure 3.3: 3 dimensional simulations for different grid sizes performed by Kleefsman

3.3. COMPARISON WITH KLEEFSMAN

The experimental data and numerical simulations performed in 3 dimensions by Kleefsman are compared with 2 dimensional domain. The pressure exerted by the dam break wave should be independent of 2d and 3d cases and the pressure point should provide with the same result since these points are placed at an exact same location for both cases. Comparison of three grid mesh sizes is conducted which were previously used by Kleefsman.

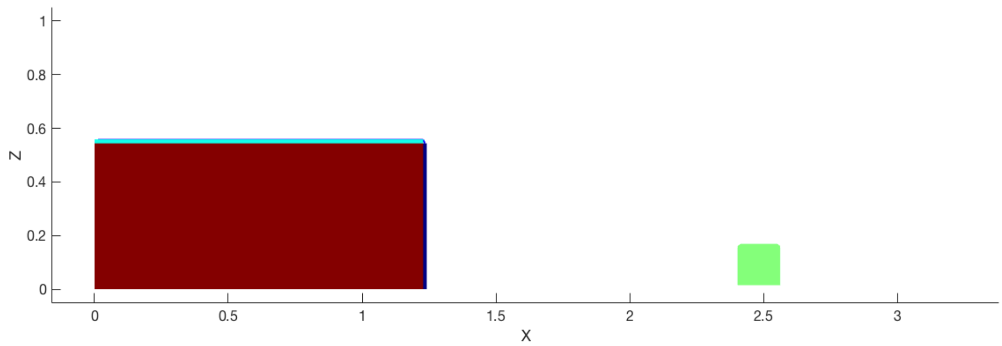


Figure 3.4: 2 dimensional view of Kleefsman experiment in ComFLOW domain

The red column represents the dam break water column and green block indicates the structure. The domain is 3.22m long and 1m high. The water column is 1.228m long and 0.55m high. The distance of structure is exactly same as in experiment explained before i.e. 1.1675m from the water column.

3.3.1. COMPARISON: GRID RESOLUTION $59 \cdot 19 \cdot 17$ VS. $59 \cdot 1 \cdot 17$

This is the coarse grid mesh size for ComFLOW.The comparison is done for pressure point P_1 on the structure. The pressure exerted by dam break wave is analyzed in a 2 dimensional case which means the grid resolution is $59 \cdot 1 \cdot 17$. Here, the grid is 1 in the y-axis. Thus the simulation is performed in X-Z plane. Figure 3.5 represents pressure behavior for three cases including the experimental result.

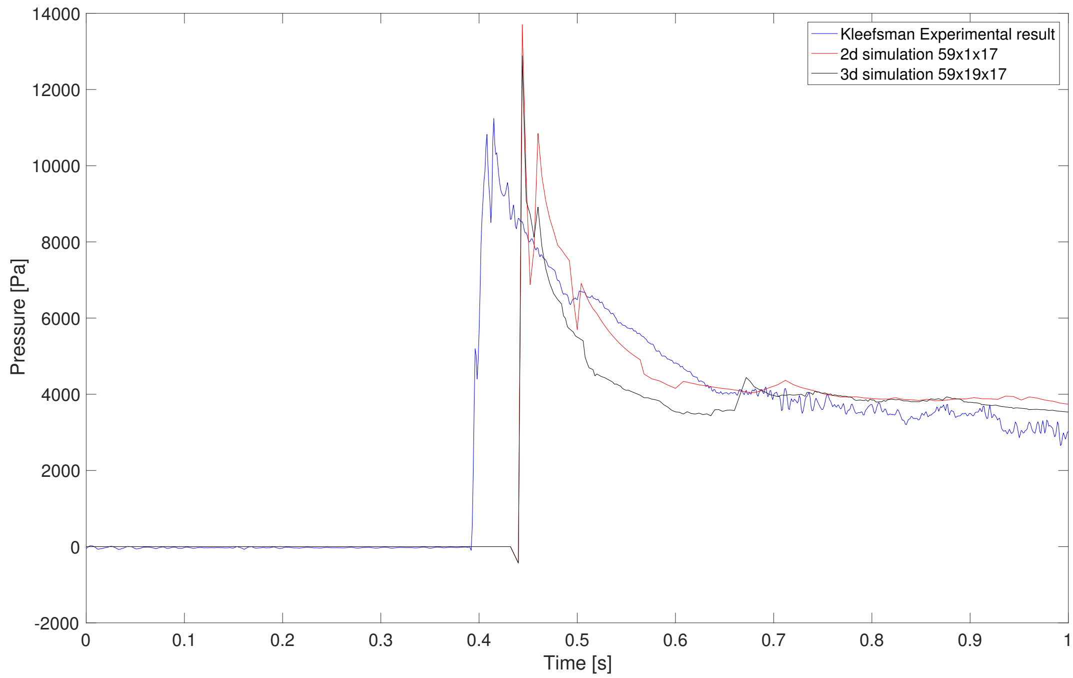


Figure 3.5: Comparison of two simulations in different dimensions for grid mesh size with experimental data

The duration of simulation is 1s, with CFL number being 0.75 with a time step of 0.01. This is kept exactly same as explained by Kleefsman [1]. It is observed that 2d and 3d simulations offer similar impact pressure peaks which change later in time with grid mesh sizes of $59 \cdot 19 \cdot 17$ and $59 \cdot 1 \cdot 17$ respectively. However, these results do not match with the experimental results as there is a delay in the initial impact of the water on the structure.

3.3.2. COMPARISON: GRID RESOLUTION $118 \cdot 38 \cdot 34$ VS. $118 \cdot 1 \cdot 34$

The grid mesh size is increased and amplified to the previous scenario. A comparison is done for Pressure point P_1 on the structure. The pressure exerted by dam break wave is analyzed in 2-dimensional case. Simulation is performed in X-Z Plane. In figure 3.6, Black pressure line indicates the 3d simulation while the red line represents the simulation in 2d case and blue pressure line is the experimental result from Kleefsman. Impact peak occurs exactly at the same time and behavior is seen to be identical, the second peak after impact is also similar. However, black pressure line seems to be stable after 0.8s while the red line shows an increase in pressure after 0.8s. There is a small spike in impact pressure peak for both the numerical simulations. But these results seem to be satisfactory since the peaks are closer to experimental data, thus can be trusted.

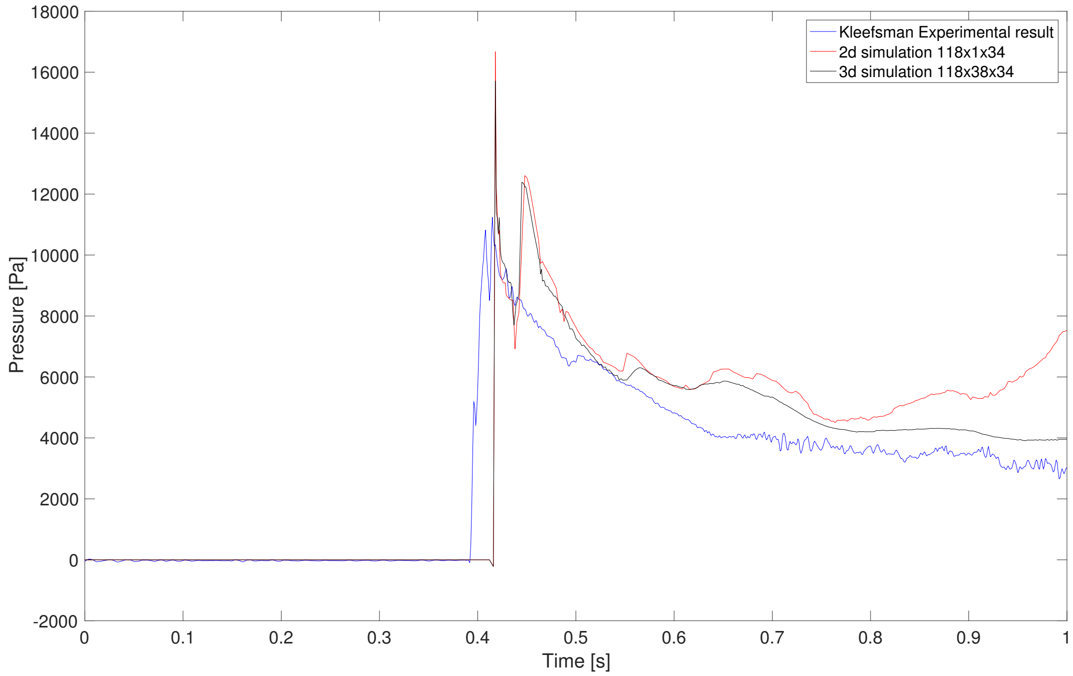


Figure 3.6: Comparison of two simulations in different dimensions for grid mesh size with experimental data

3.3.3. COMPARISON: GRID RESOLUTION $236 \cdot 76 \cdot 68$ VS. $236 \cdot 1 \cdot 68$

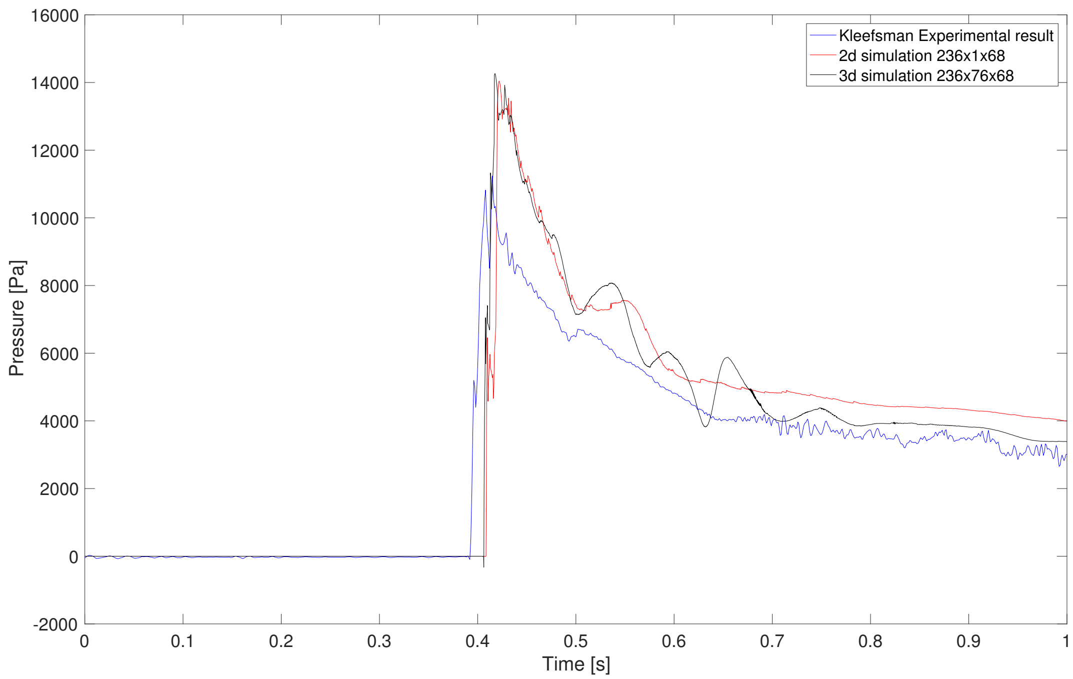


Figure 3.7: Comparison of two simulations in different dimensions for grid mesh size with experimental data

The grid mesh size is further amplified to the previous scenario. A comparison is done for Pressure point P_1 on the structure. Simulation is same as compared to previous cases. In figure 3.7, Black pressure line represents the simulation in 3d while red line indicates numerical simulation in the 2d scenario and the blue line represents the results of an experiment by Kleefsman. It is seen that pressure behavior in the 2d case is somewhat similar to the 3d case. The initial impact pressure is almost same and then pressure stability is also observed in 2 dimensional case. Since 2d and 3d cases behave similar to each other and are closest to the experimental results, grid resolution for these simulations can be trusted in ComFLOW.
Results from another pressure point are compared to conclude the selection of this grid resolution. Pressure point P_2 on the structure is selected and comparison for 2 dimensions with 3 dimensions are performed. From pressure point P_2 similar results are observed as from point P_1. Initial impact peaks are the same which is the main interest.

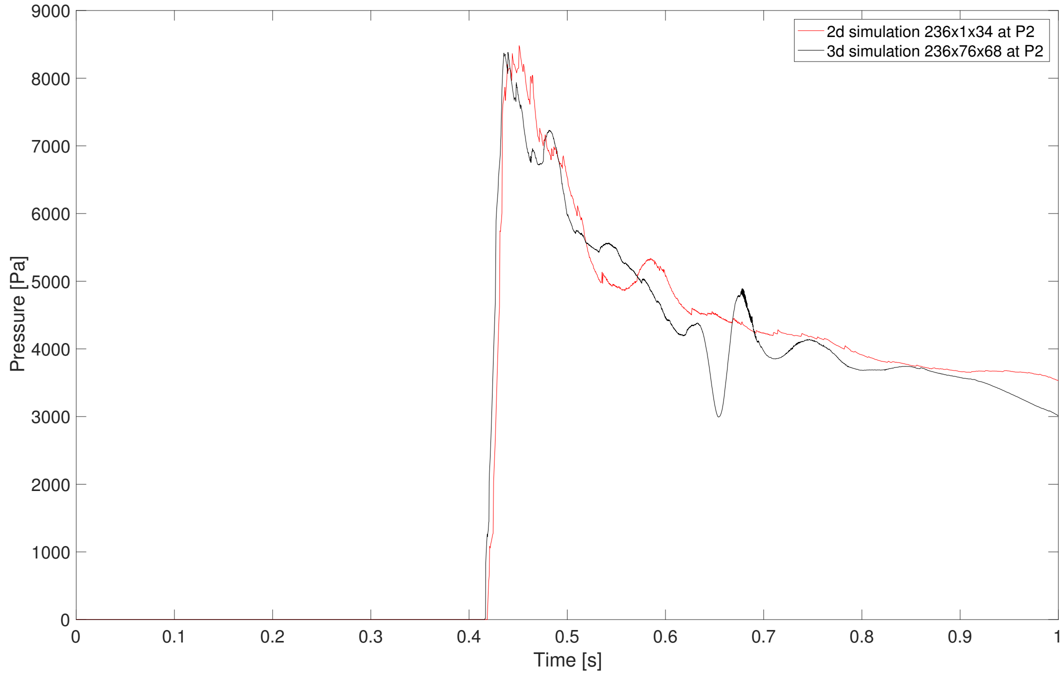


Figure 3.8: Comparison of two simulations in different dimensions for grid mesh size at pressure point P_2

3.4. CONCLUSION

Kleefsman simulations were compared with simulations in 2 dimensions to obtain grid resolution which can be trusted to conduct further study in this research. For the coarsest grid, the 2d and 3d pressure graphs coincide with each other. For higher grid $118 \cdot 1 \cdot 34$, initial pressure peak is almost the same but later, changes can be observed while for finest grid resolution $236 \cdot 1 \cdot 68$, the similarity in pressure behaviour over a duration of 1s is observed for both the pressure point P_1 and P_2. So with these observations in mind, $236 \cdot 1 \cdot 68$ is selected as our final grid resolution which offers grid cells size of Δx=0.01m for x-direction and Δz= 0.01m for z-direction. It is the finest grid and offers results that can be trusted. For simulations in chapters further, grid resolution will vary but the size of grid cells will remain the same i.e. 0.01m in x and z direction.

4

Dam Break Simulations with Systematic variation

Water flow impact on deck and subsequent impact is a key issue in this research. This involves the fast flow of fluid along with the impact followed by the stationary and stability phase later. This whole study is idealized as the Dam-Break wave on a fixed and dry surface. Dam break wave is often studied as a verification of numerical method for which ComFLOW software is used. Water impact and green water loading are numerically investigated using the dam break model approach in the software.

4.1. Variation of Dam break wave height

ComFLOW software is used for the numerical simulation. The main feature of this simulation is to define a numerical model which is able to reproduce the dam break wave model which is idealised as green water on vessel deck. The domain size of 3.22m · 1m · 1m is used for simulation. Dam heights are varied by 0.05m from 0.2 to 0.55m. In the domain, a box structure of 0.161m · 0.403m · 0.161m is placed at a distance of 2.3955m in the x-direction, which is idealized as a scale model of the structure kept on the deck of a ship. However, the structure is kept fixed to reduce the complexity of the problem. Simulation is conducted to observe the impact pressure and impact force. A 2 Dimensional simulation is performed with the grid resolution of $236 \cdot 1 \cdot 68$. The domain and grid resolution are kept similar to Kleefsman's [1] numerical simulations explained in the previous chapter. The duration of the simulation is 1s which is long enough to capture the impact loading caused by the fluid with automated adaptive time step. CFL_{max} is 0.8 and the time step used is 0.01s. A monitor point is placed 0.021m above the surface on the structure to obtain the pressure impact and velocity of the fluid hitting the structure, Similarly, three other pressure points are placed onto the structure at a distance of 0.04m apart in z direction. The water column from the left flows with increase in time, and hits the structure.

Figure 4.1 represents the pressure behavior in time for duration of 1s. It is observed that for dam height of 0.55m, highest pressure is obtained which reduces with time as the dam height is decreased. During this phenomenon, a moment when water hits the structure, its flow direction is deflected 90 degrees which result in peak pressure on the structure in a typical time period. Part of the water shoots up vertically along the wall, resulting in quasi-static load at the lower levels, combined with the effect of flow stagnation. This results in secondary peak which is lower than the initial peak.

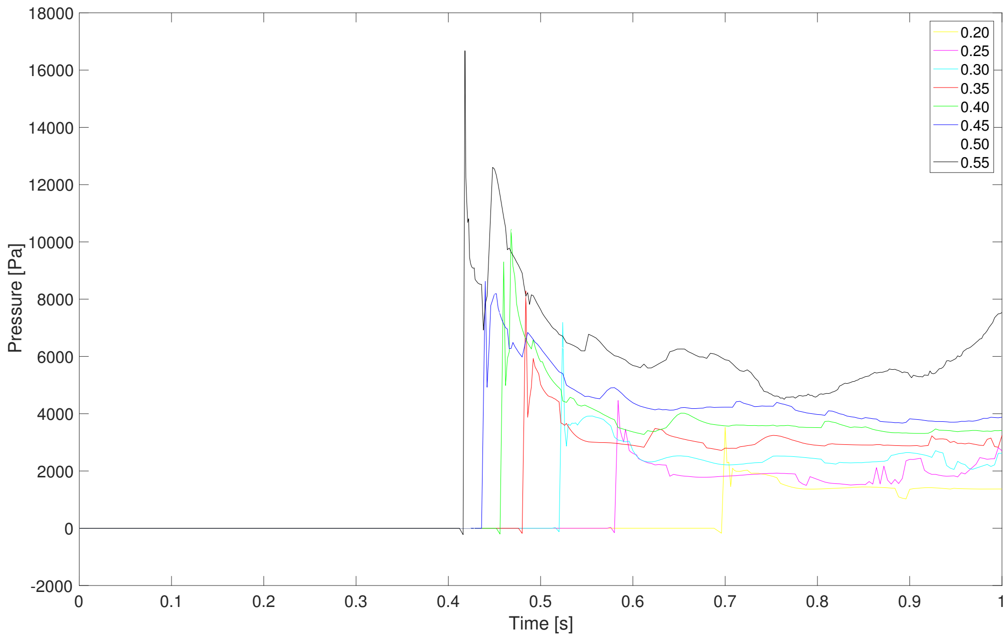


Figure 4.1: Dam Break Comflow simulation indicating Pressure exerted on the structure over a duration of 1s for different wave heights

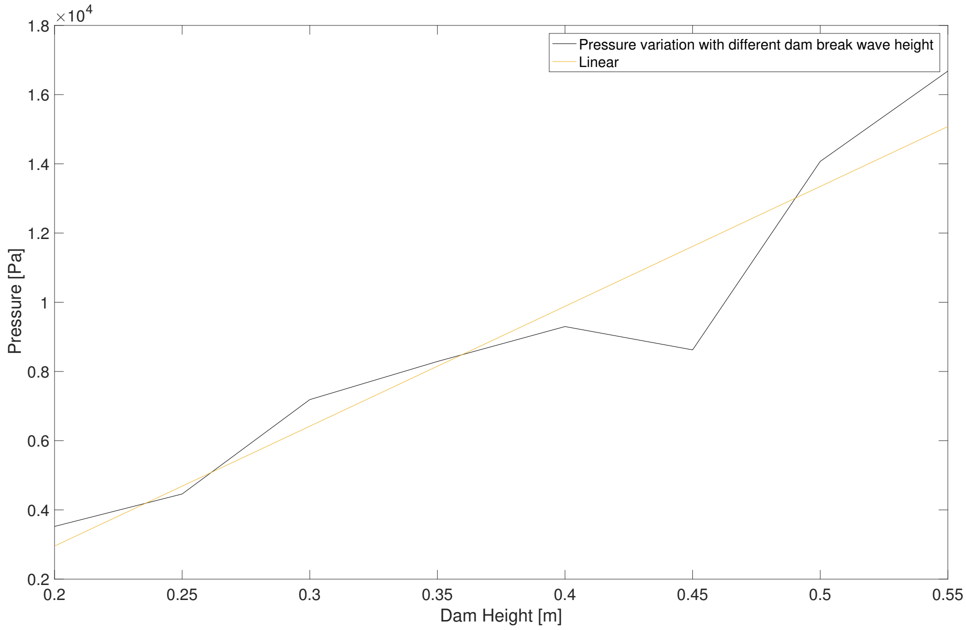


Figure 4.2: Pressure variation with different dam height

Pressure peak is observed which is obtained when the fluid initially hits the structure, then stationary phase is observed along with stability phase later. The fluid flows instantaneously when the simulation is performed and hits the structure between time 0.4s and 0.6s which can be confirmed from the snapshots and also the pressure diagram 4.1. It can be seen in figure 4.2 that pressure increases with increase in dam break wave height. Only the highest impact pressure peak is considered for all the wave heights in this figure which is due to the dam break wave impact on the structure.

Four Snapshots from the same simulation of 0.55m wave height can be observed below in series of figures which indicates how the pressure of the fluid varies on to the structure for a different time in 2 dimensional flow.

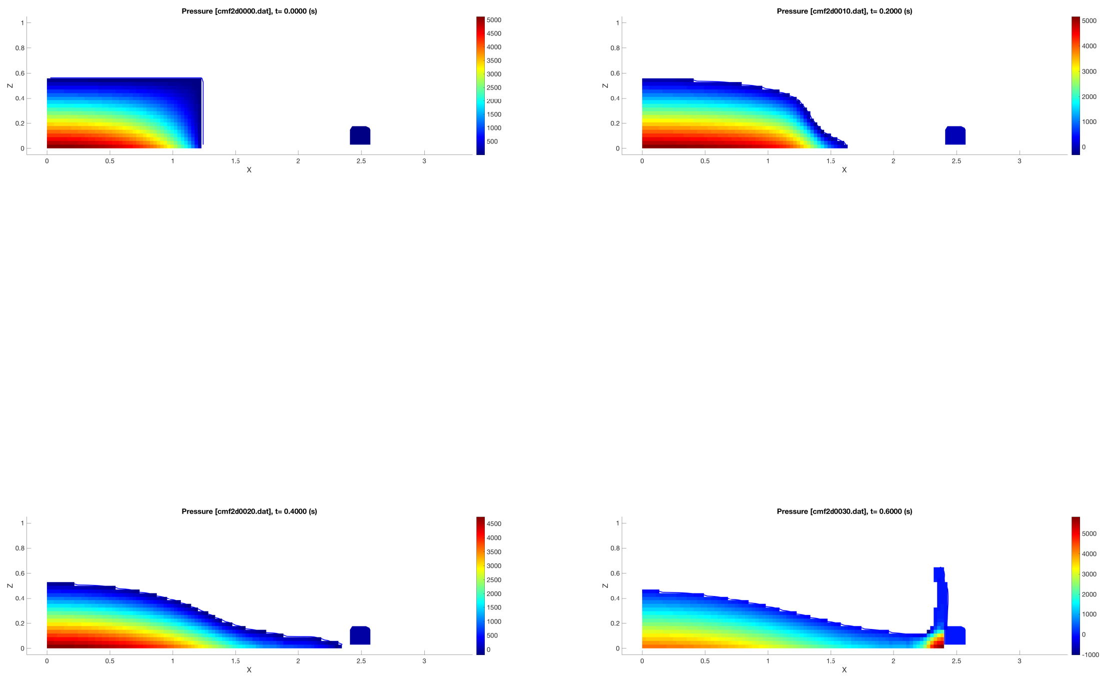


Figure 4.4: Series of ComFLOW snapshots representing variation in pressure for different time period for 0.55m wave height

4.2. VARIATION IN DISTANCE OF STRUCTURE FROM DAM BREAK WAVE

For simulation of idealised green water on the deck of a vessel as a dam break model, the systematic study is performed which involves a variation of distance of the structure in the x-direction. This variation is used to calculate the maximum impact pressure on the structure with respect to regular dam break wave and the distance of the structure from the wave. The domain size is increased for this study which is now 5m long, 1m wide and 2m high as indicated in 4.5. The grid resolution is increased to $500 \cdot 1 \cdot 200$ which offers 0.01m grid cell size in x and z direction. This is the same resolution. The distance of structure is varied from 1.5m to 2.8m in x-direction i.e. 0.5m to 1.3m from dam break wave in a domain. The dam break wave is kept constant which is 1m high and long and flows from the left on to the structure. The simulation duration is reduced to 0.8s just to observe impact pressure in at different locations. The time step is now 0.01s and CFL_{min} and CFL_{max} as 0.15 and 0.4 respectively.

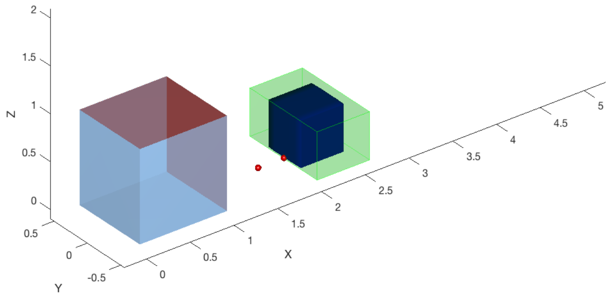
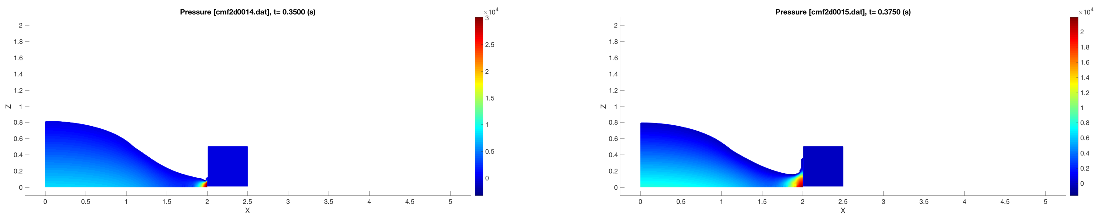


Figure 4.5: 3D view of dam break wave in ComFLOW

Systematic Variation is studied in ComFLOW software with the structure kept initially at a distance of 0.5m from the water column of length and height of 1m respectively. The distance is then varied by 0.1m while the water height and length remains the same.



(a) (b)

Figure 4.6: Comflow simulation snapshot indicating pressure for 1m wave height for a structure kept at 2.0m in x-direction at time 0.35s and 0.37s

Figure 4.6 is the snapshots of the Comflow simulation indicating the inertia dominated peak pressure for time 0.3 and 0.3750s. The pressure obtained is in the range of 10^4Pa. These snapshots are then compared with the pressure graph obtained from the monitor point placed 0.025m before the structure geometry as in figure 4.7.In 4.7, Pressure peak at 0.35s can be observed, it is the impact pressure peak obtained when the displaced water hits the structure. Comparing the pressure graph over a duration of 0.35s with the ComFLOW snapshots in figure 4.6a and 4.6b from the same simulation at two different time periods, accurate peak pressure is obtained. The monitor point picks up all the activity of dam break wave, thus the impact pressure can be shown earlier than anticipated from snapshots.

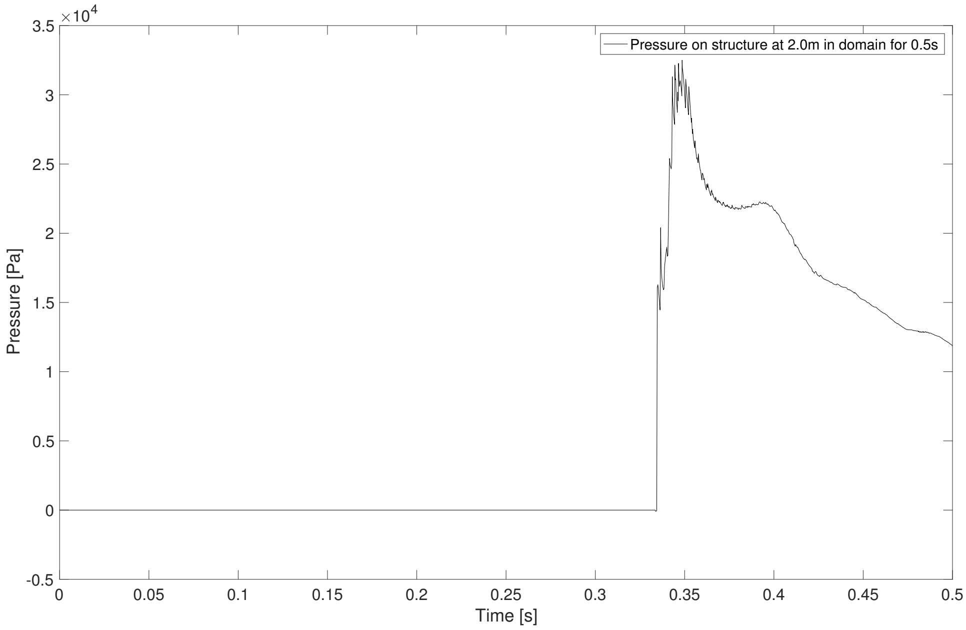


Figure 4.7: Simulation of 1m wave height with structure at 2.0m in x-direction for duration of 0.5s

To continue the study of pressure peak over a varied distance of structure, simulations with same input parameters are performed. However, the distance of a structure is varied. There are many pressure peaks at 0.35s in figure 4.7, thus an average of three pressure is selected as impact pressure for the study of pressure vs distance relation explained in the appendix. Figure 4.8 represents the pressure variation for 1m dam break wave over a distance of 1.5m to 2.8m in the ComFLOW domain.

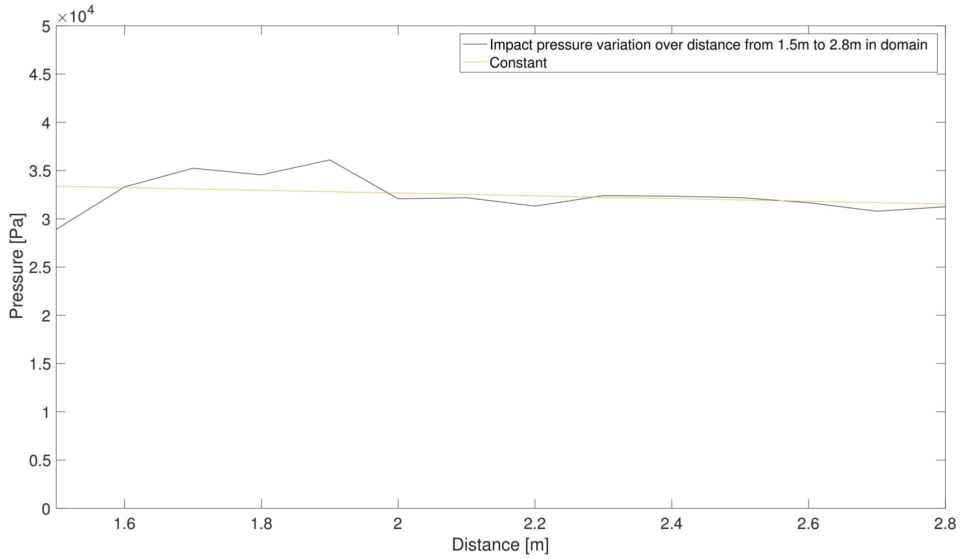


Figure 4.8: Variation of 1m dam break wave with structure at distance varied from 1.5 to 2.8m in ComFLOW domain

It is observed that there is a slight increase in impact pressure till 1.9m and then it drops at 2.0m but then appears to be constant irrespective of distance. There is slight fluctuation in impact pressure in and around 3.2x10^4Pa after 2.0m. Thus, it can be concluded that impact pressure remains constant over distance.

4.3. ISSUES IN COMFLOW

4.3.1. FORMATION OF WATER DROPLETS

All the parameters are kept constant as in the previous case. It is observed that ComFLOW is very sensitive to CFL number. For previous case, CFL_{min} and CFL_{max} is 0.15 and 0.4 respectively, However for this case it is changed to 0.3 and 0.8 respectively and results appear to be distant from what was obtained previously. Irregularities in ComFLOW is observed as the time duration of the simulation is increased. These irregularities in ComFLOW arises which causes the liquid in a domain to behave like small water droplets and fires them towards the structure. Every droplet then stipulates pressure peak onto the monitor point making it difficult to analyze real initial impact pressure. This phenomenon is represented in figure 4.9 where peaks larger than the impact pressure are obtained later in the simulation which is of the order 10^6Pa.

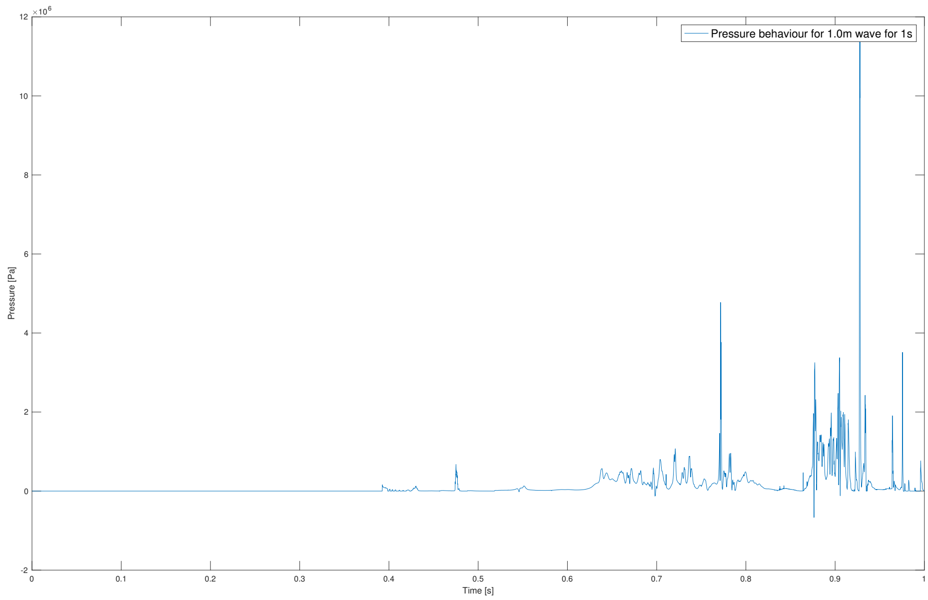


Figure 4.9: Simulation of 1m wave height with a structure at 2.7m in the x-direction for a duration of 1s

A simulation considered, the structure is at 2.7m in ComFLOW domain and regular dam break wave is simulated which has 1m height and length, similar to the previous case of distance variation. Only the CFL number is changed while every parameter remains the same. In figure 4.3.1, the formation of droplets all over the ComFLOW domain is shown which cause very high irregular peak pressures which cannot be trusted. This occurs due to a bug in ComFLOW.

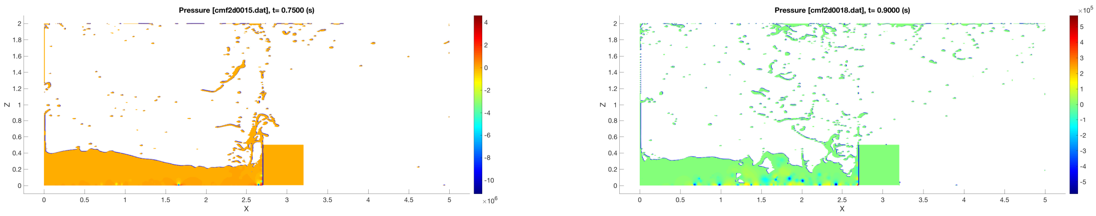
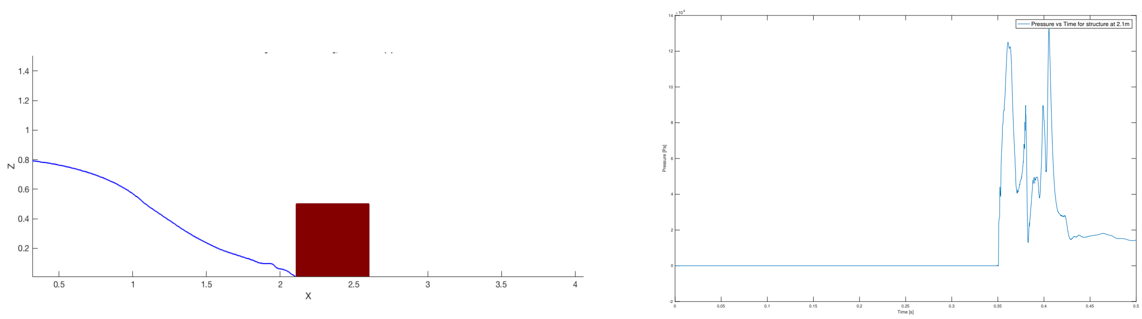


Figure 4.10: Water droplets in Comflow domain at 0.75s and 0.9s

4.3.2. DAM BREAK WAVE SHAPE

Pressure depends on incident wave shape as studied by Mokrani [24]. While performing the dam break simulations with systematic study in this research with higher CFL number i.e. 0.3 and 0.8, Wave shape appears to change with an increase in distance of the structure from dam break wave which resulted in a change in the trend of impact pressure peaks.
Case 1: The structure is placed 2.1m in the ComFLOW domain i.e. 1.1m from the water column which is 1m high. Wave shape is observed just before it hits the structure. The figure below represents the water column to a distance of structure in x-z plane. X axis shows the distance of a structure in the domain while the z-axis represents the wave height. Figure 4.11a indicates the wave shape just before initial impact i.e. 0.35s into dam break while figure 4.11b indicate the pressure generated from this wave respectively.

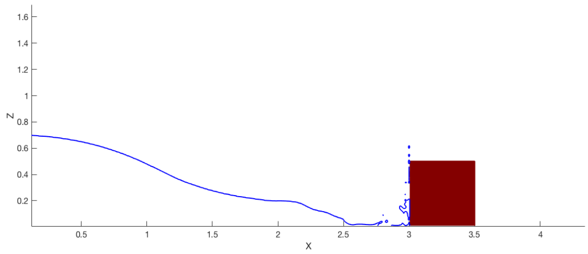


(a) Wave shape before initial impact

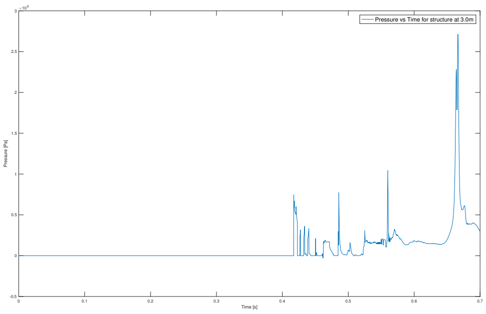
(b) Pressure resulting due to wave impact

Figure 4.11: Wave shape and respective pressure generated due to impact from the wave

Case 2: The structure is placed at 3.0m in the ComFLOW domain which is 2.0m from 1m high dam break water column. Wave shape and its impact are observed before the impact on the structure. The series of figure below represents the wave shape and the pressure generated by it respectively. The domain axis remains the same as in the previous case. It is observed that water droplets are formed and travel faster before the initial impact of liquid on the structure, also monitored from the monitor point placed in front of the structure. Figure 4.12b indicates the pressure generated by an incoming wave with its respective shape.



(a) Wave shape before initial impact



(b) Pressure resulting due to wave impact

Figure 4.12: Wave shape and respective pressure due to impact from the wave for a structure at 3.0m

4.4. CONCLUSION

Dam break waves were studied with variation in dam height and distance of the structure from the wave. From the variation of dam height, it is observed that impact pressure is almost linear with dam break height. For variation in distance of structure, it is observed that impact pressure is constant and independent of distance variation. It is also observed that ComFLOW is sensitive to CFL parameter. Although it is stable up to CFL=1, CFL of 0.4 is required for reliable and stable results.

5

Dam break waves on idealized deck

Simulations are performed for 2 dimensional dam-break waves on a structure which is idealized as a deck on an offshore vessel. The objective of these simulations is to be one step in between a dam break in an empty domain to simulating green water on a ship in waves. In the previous chapter, dam break waves were simulated on ComFLOW domain boundary conditions. In this chapter, a structure is designed which is idealized as a deck of vessel and dam break waves will be simulated on this deck to observe the changes and effects between the dam break waves on domain boundary conditions to deck structure.

5.1. ComFLOW domain for idealised deck

The Comflow domain is 5m long, 1m wide and 2m high. An idealized deck is designed in front of a structure which is a representation of vessel deck with topside structure. The dam height of 1m is generated on this deck. The grid resolution is $500 \cdot 1 \cdot 200$ which results in 0.01m grid cells in x and z direction. The time step is 0.01s and CFL_{min} and CFL_{max} are 0.15 and 0.4 respectively. Monitor point is placed in front of the structure which is represented by the red dot in figure 5.1. Simulation on idealized deck is generated to observe changes in pressure graphs which may arise due to the presence of deck instead of ComFLOW boundary.

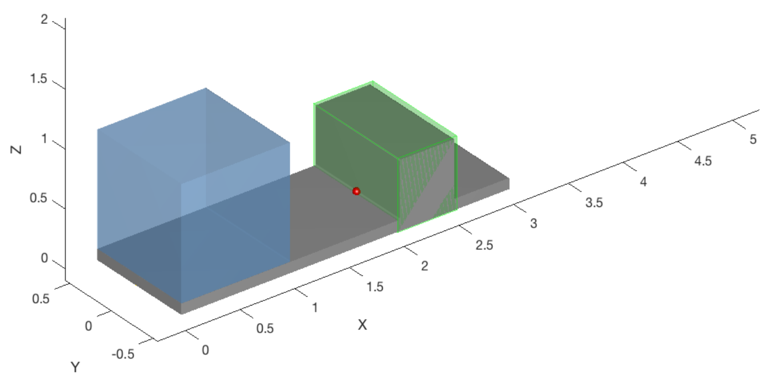


Figure 5.1: 3d isometric view of idealized deck in ComFLOW

A figure of the pressure can be observed in 5.2, the structure for this simulation is at 2.0m in x-direction or 1m from dam break wave. The dam break simulation is generated on the deck and the impact pressure is monitored by the monitor point before box structure.

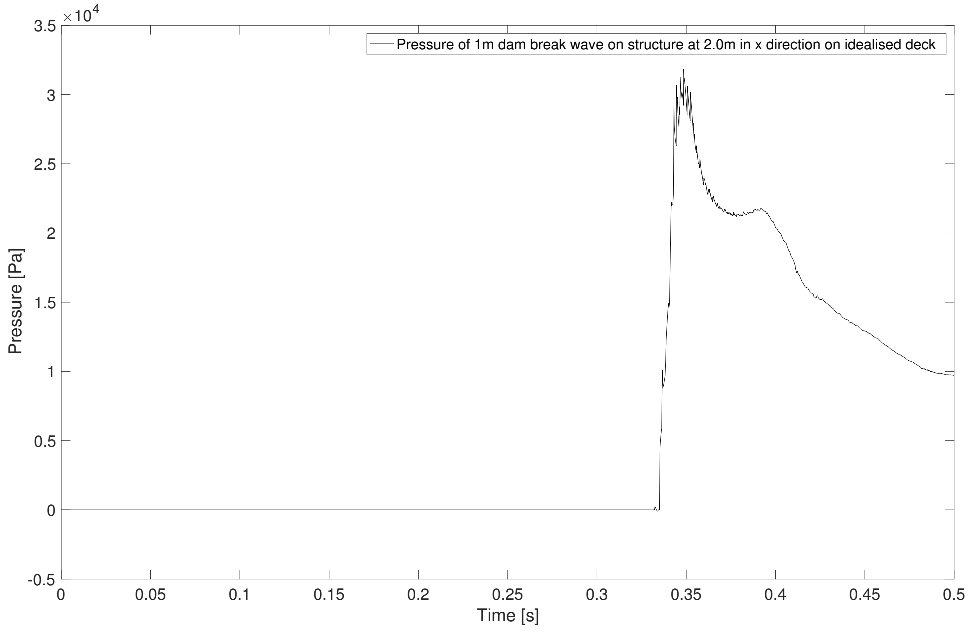


Figure 5.2: Pressure generated by dam break wave on a structure at 2m in the x direction

The same simulation has been generated previously with ComFLOW domain boundary conditions, thus it can be compared with this simulation involving ideal deck.

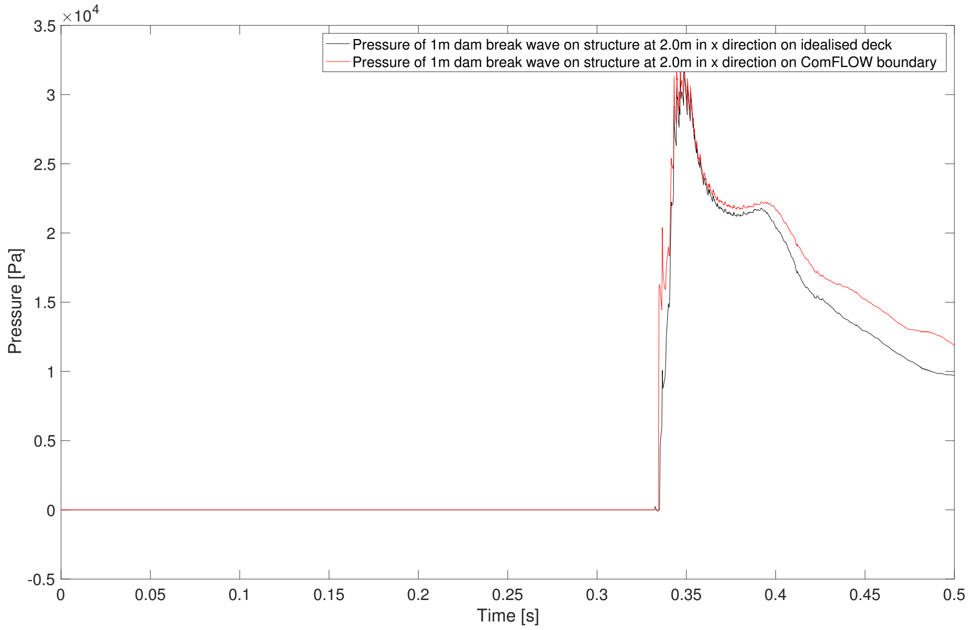
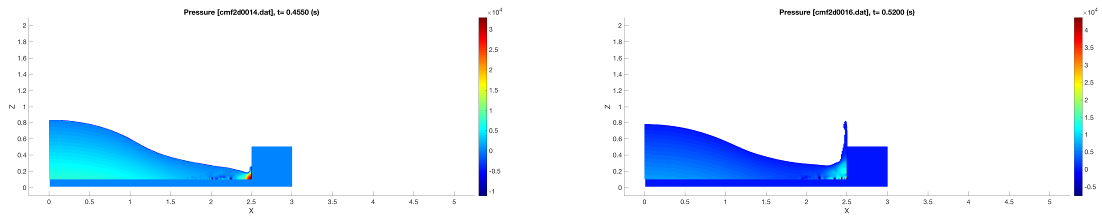


Figure 5.3: Comparison of two same simulations

From figure 5.3, it is observed that the pressure generated on an idealised deck in black line is slightly lower compared to one with boundary condition indicated in the red line. The impact peak is lower along with reduced pressure for stationary and stability phase.

5.2. VARIATION OF STRUCTURE ON DECK

Since there are slight changes observed with dam break simulations on idealised deck, it is desired to vary the distance of structure on deck and observe the trend of impact pressure generated. The figure represents the pressure generated by dam break wave on structure placed at 2.5m in x direction on idealised deck.



(a) (b)

Figure 5.4: Pressure exerted on the structure at 2.5m on idealised deck

The input parameters remain same, impact pressure is observed for different locations. The distance varied is from 1.5m to 2.8m in ComFLOW domain and the pressure is calculated for every variation in 0.1m while dam height remains same. The pressure is averaged to three impact peaks to get reasonable results explained in the appendix. The pressure to distance relationship can be observed in figure 5.5:

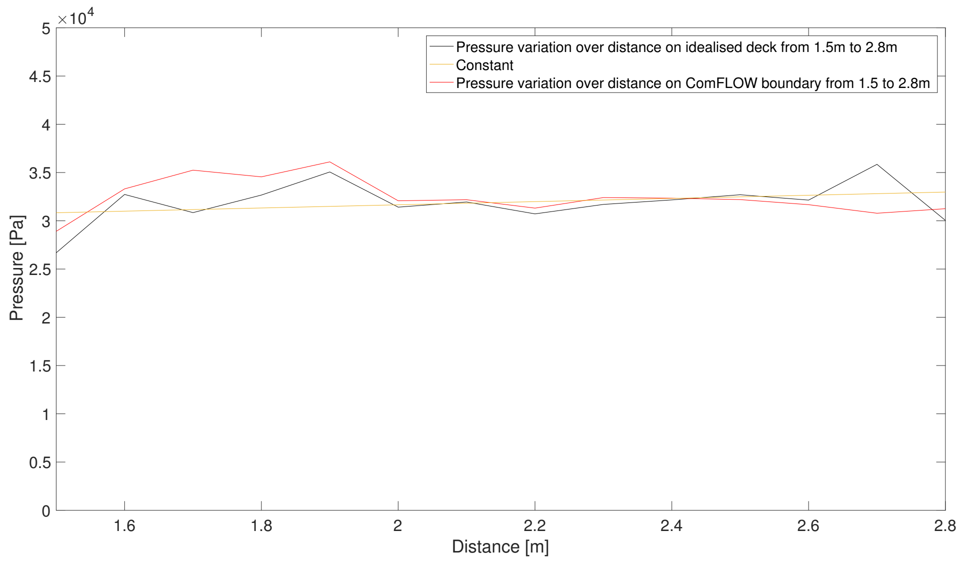


Figure 5.5: Pressure variation for a 1m wave on an idealised deck and ComFLOW boundary

It is observed that the pressure line for the idealised deck (black line) is lower than for ComFLOW boundary conditions(red line). Both lines, however, remain constant over distance. Thus, it can be concluded that pressure generated by dam break wave impact is independent of distance.

5.3. VARIATION OF DAM BREAK HEIGHT ON DECK

The main objective of this simulation is to simulate green water on the deck of a vessel using dam break model. The ComFLOW domain size is now reduced to $3.22\text{m} \cdot 1\text{m} \cdot 1\text{m}$. Dam heights are varied by 0.05m from 0.2 to 0.55m. In the domain, a box structure 0f $0.161\text{m} \cdot 0.403\text{m} \cdot 0.161\text{m}$ is placed at a distance of 2.3955m in the x-direction and on the top of an idealised deck which is 0.05m high and 1m wide. The dam break wave is simulated on top of this idealised deck. The structure on deck is fixed for all simulations to reduce the complexity of the problem. A 2 dimensional simulation is performed with a grid resolution of $236 \cdot 1 \cdot 68$ which offers 0.01m grid cells in x and z-direction. The grid resolution and size of grid cells are kept similar to previous simulations of dam break waves on ComFLOW boundary conditions. The duration of the simulation is 1s which is good enough to capture impact loading caused by dam wave.

The time step is 0.01s with CFL_{min} and CFL_{max} of 0.15 and 0.4 respectively. A monitor point is placed 0.021m above the surface on the structure to obtain pressure impact, similarly, three other pressure points are placed onto the structure at a distance of 0.04m apart in z-direction.

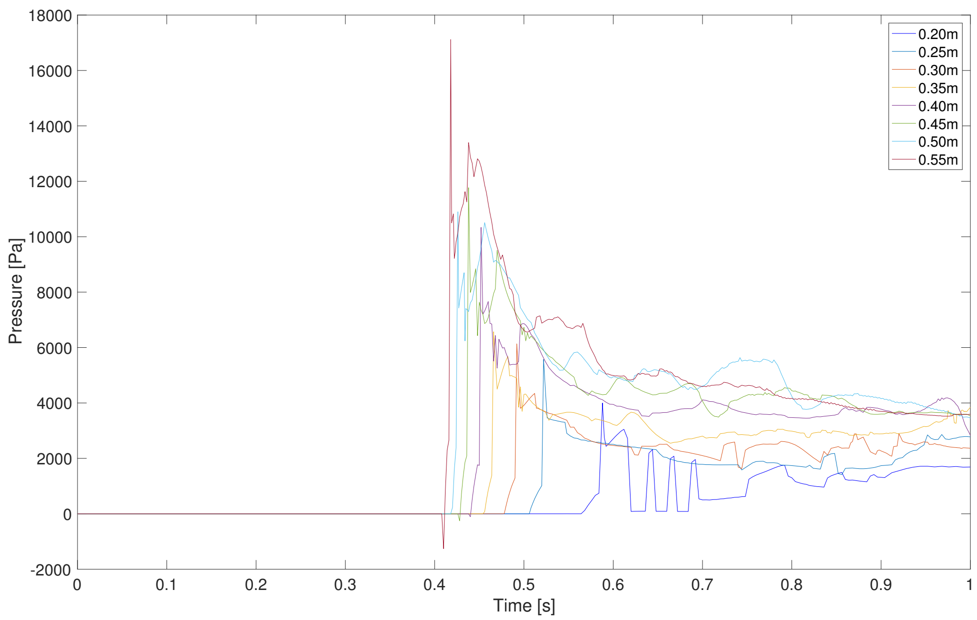


Figure 5.6: Dam break ComFLOW simulations on idealised deck indicating pressure exerted on the structure over duration of 1s for different dam heights

The figure in 5.6 represents the pressure behaviour in a time period of 1s for different dam heights. It is observed that for dam height of 0.55m, highest pressure is obtained which reduces with time as the dam height is decreased. In figure 5.7, it is observed that pressure increases linearly with increase in dam break height which is similar to the variation of dam height on ComFLOW boundary conditions.

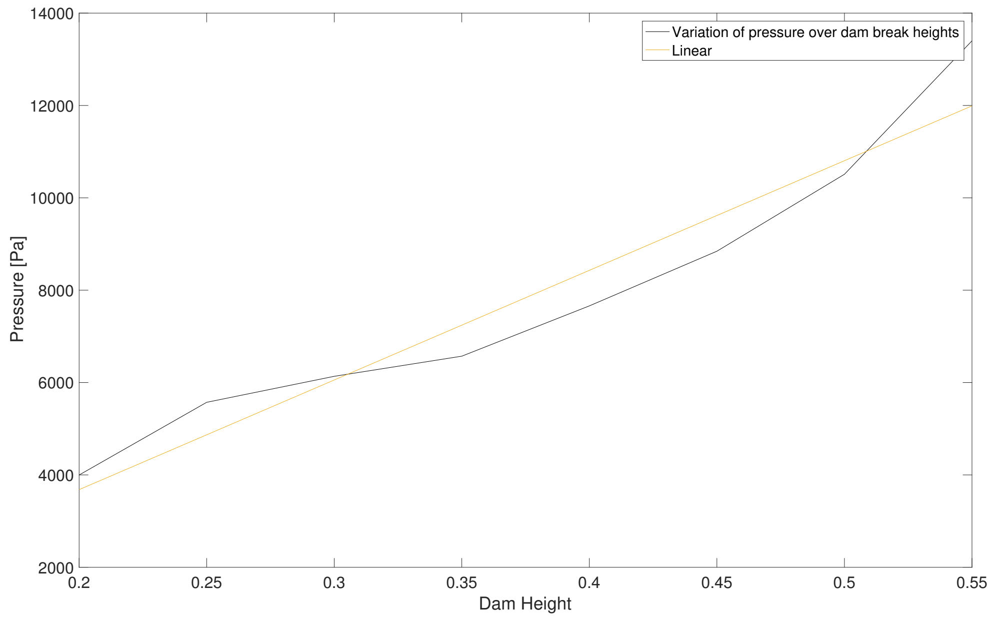


Figure 5.7: Pressure variation over different dam heights on idealised deck

In figure 5.8, The pressure variation over dam break heights on an idealised deck (black line) is compared with that to ComFLOW boundary conditions(red line). It is observed that pressure generated on the structure on an idealised deck is lower compared to that on boundary conditions. However, both lines show linear behaviour over dam heights.

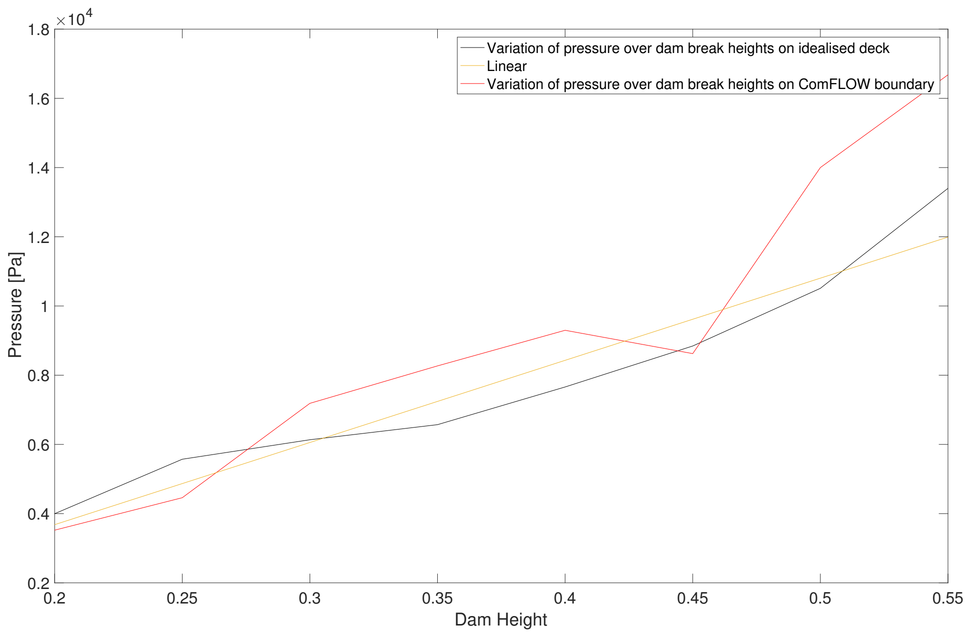


Figure 5.8: Comparison of pressure variation for simulations with idealised deck to ComFLOW boundary conditions

5.4. CONCLUSION

The dam breaks waves were simulated on an idealised deck to observe changes in pressure behaviour to previously generated dam-break waves on ComFLOW boundary conditions. It is observed that the pressure generated on idealised deck is lower to boundary condition simulations for both the variations of dam height and distance of the structure on deck. Since no slip boundary conditions are the default conditions at the domain boundaries in ComFLOW, there is a change in pressure on idealised deck.

6

Structure in Regular waves

Impulsive pressures generated from regular or irregular waves are a great concern for the integrity and safety of the structures threatening equipment, facilities, and personnel on deck in far offshore locations. Heavy loading exerted by wave impingement damages the structure on the deck of the vessels such as FPSOs (Floating production storage and offloading vessels). The knowledge of impact pressure caused by non-breaking waves has been developed and integrated into practical designs. However due to the complex nature involving Green water phenomenon which includes regular and irregular waves on the vessel are still not well understood.

6.1. Absorbing Boundary condition

The boundary conditions which does not influence the outgoing waves or generates numerical reflections affecting the waves inside simulation domain and at the same time can allow the regular and irregular waves to enter the domain needs to be considered. Absorbing boundary conditions are used for numerical simulation of regular Stokes wave and irregular JONSWAP spectrum waves in two dimensional ComFLOW domain which is an extension of Sommerfeld condition. The absorbing boundary conditions for 2D flow for potential Φ [26]:

$$\left(\frac{\partial}{\partial t} + c^{out}\frac{\partial}{\partial x}\right)\Phi^{out} = 0 \tag{6.1}$$

It is fully absorbing for the outgoing wave in the domain with phase velocity c^{out}. However (6.1) gives reflection for waves with components of other phase velocities $c(kh)$ arriving at the boundary at the end of a domain, where kh is dimensionless wave number. The dispersion relation can be obtained by the following approximation, It is, however, the poor form of approximation [26]:

$$c(kh) = \sqrt{gh}\sqrt{\frac{tanh(kh)}{kh}} \approx c^{out} \tag{6.2}$$

For the absorbing condition used in ComFLOW, dispersion equation is approximated by the rational function to obtain the superior approximation.

$$c(kh) \approx \sqrt{gh}\sqrt{\frac{a_0 + a_1(kh)^2}{1 + b_1(kh)^2}} \tag{6.3}$$

the coefficients here (a_0, a_1, b_1) are tuned for optimal approximation over the possible ranges of kh values. The reflection coefficient can be represented as:

$$R = \frac{c^{out} - c(kh)}{c^{out} + c(kh)} \tag{6.4}$$

The reflection of absorbing boundary condition is much lower over the wide ranges of kh values. Due to e^{-kz} behavior of Φ in z-direction, i.e. $k^2\Phi^{out} = \frac{\partial^2}{\partial z^2}\Phi^{out}$, k is replaced by second order derivatives along the boundary of the domain. This leads the Absorbing boundary condition to be:

$$\left[\left(1 + b_1 h^2 \frac{\partial^2}{\partial z^2}\right)\frac{\partial}{\partial t} + \sqrt{gh}\left(a_0 + a_1 h^2 \frac{\partial^2}{\partial z^2}\right)\frac{\partial}{\partial x}\right]\Phi^{out} = 0 \tag{6.5}$$

Absorbing boundary conditions can also be applied for the waves propagating at an angle α, in this case, head waves on the vessel bow are considered, used to reduce the numerical reflections from open boundaries.

6.2. VARIATION OF WAVE HEIGHT AND WAVE PERIOD FOR STRUCTURE IN REGULAR WAVES

The pontoon structure is placed in ComFLOW domain to study green water loading on deck. Regular stokes wave of 5^{th} order is generated in the domain. Nine simulations are generated with a variation of wave height and wave period respectively. The domain is 12m long, 1m wide and 1.6m high and a water depth of 0.6m. Absorbing boundary conditions are used at the end of a domain to prevent wave reflections within the domain. The grid cells used are 0.01m in x and z-direction i.e. i_{max} and k_{max}= 1000 and 130 respectively. The wave height and wave period for different simulations are provided in table 6.1: Three wave heights with three different wave period are generated on pontoon like structure to observe green water loading on the structure on the deck. Monitor points are assigned on the structure to capture impact pressure due to green water. Three different

Wave height and wave periods in ComFLOW	
Wave Height	0.2m
Wave Period	1.142, 1.342, 1.5
Wave Height	0.18m
Wave Period	1.142, 1.342, 1.5
Wave Height	0.16m
Wave Period	1.142, 1.342, 1.5
Time duration	30s
Time step	0.01s
CFL Number (Min - Max)	0.25 - 0.4

Table 6.1: Wave input Parameters

wave periods of 1.142 , 1.342 and 1.5s are generated for wave heights of 0.2 , 0.18 and 0.16m respectively. The distance of the structure on deck remains constant for these variations. The CFL_{min} and CFL_{max} is kept 0.25 and 0.4 respectively. The time duration is kept 30s which is long enough to observe various green water events but the first significant green water is considered for comparison.

6.2.1. WAVE PERIOD: 1.142S

This case indicates the pressure generated by the wave with a wave period of 1.142s for three different wave heights. Figure 6.1 represents the pressure generated on deck. The simulation is generated for 30s however, only first significant green water is of importance which occurs at 8s into the simulation. Thus, the impact peaks are refined in time for better observation.

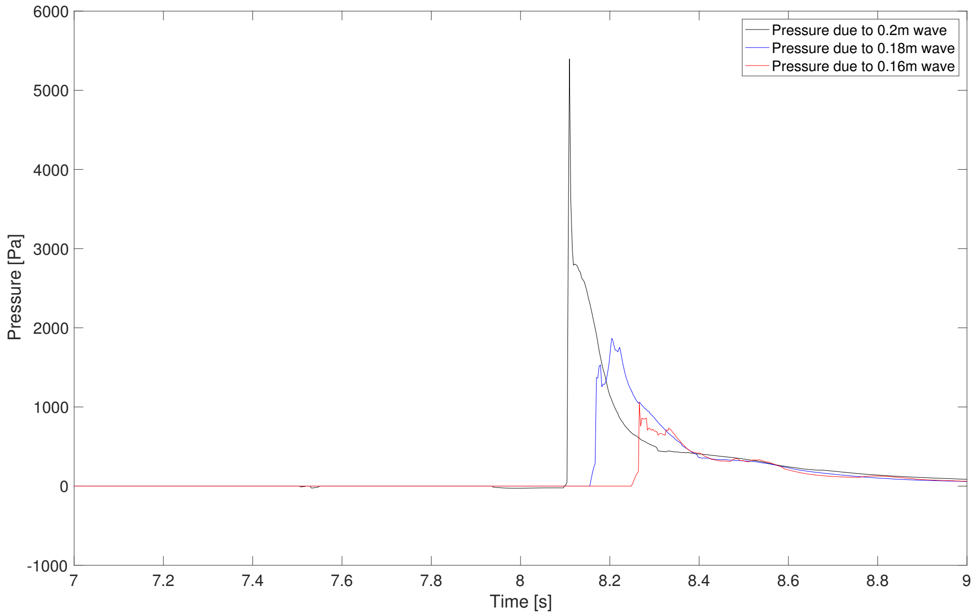


Figure 6.1: Impact pressure for wave period of 1.142s for three waves

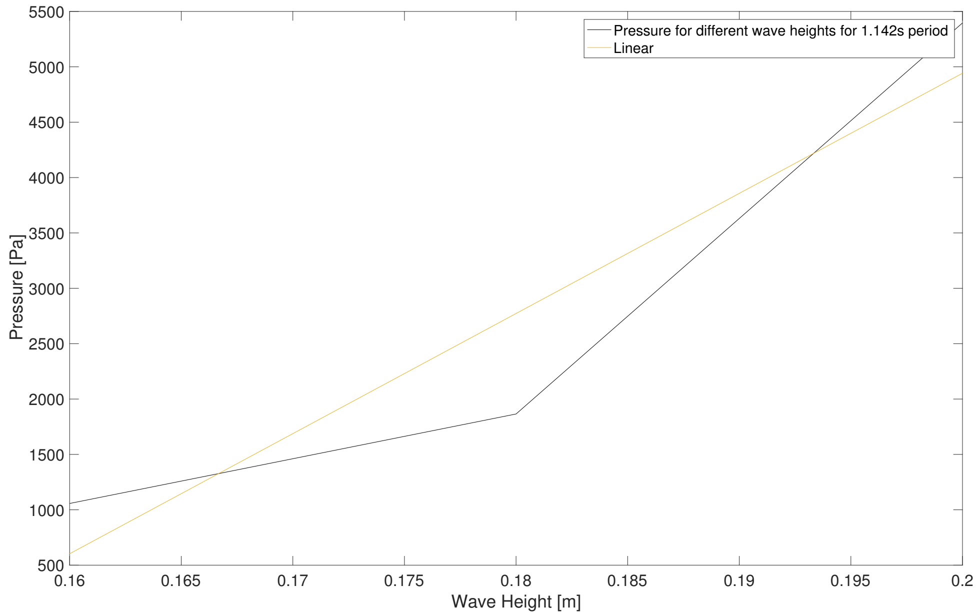


Figure 6.2: Impact pressure with wave height relation

In figure 6.1, the black pressure line indicates the pressure generated by a 0.2m wave, while blue and red line represents the pressure for 0.18m and 0.16m wave height respectively. It is observed that pressure generated by 0.2m wave height is largest compared to two other waves for 1.142s wave period.

Figure 6.2 represents the wave height and impact pressure relation, it is observed that the pressure generated on the structure on a deck is linear which is similar to observed cases with dam break simulations. With linear increase of wave height, increase of pressure is observed which is similar to cases involving dam break study.

6.2.2. WAVE PERIOD: 1.342S

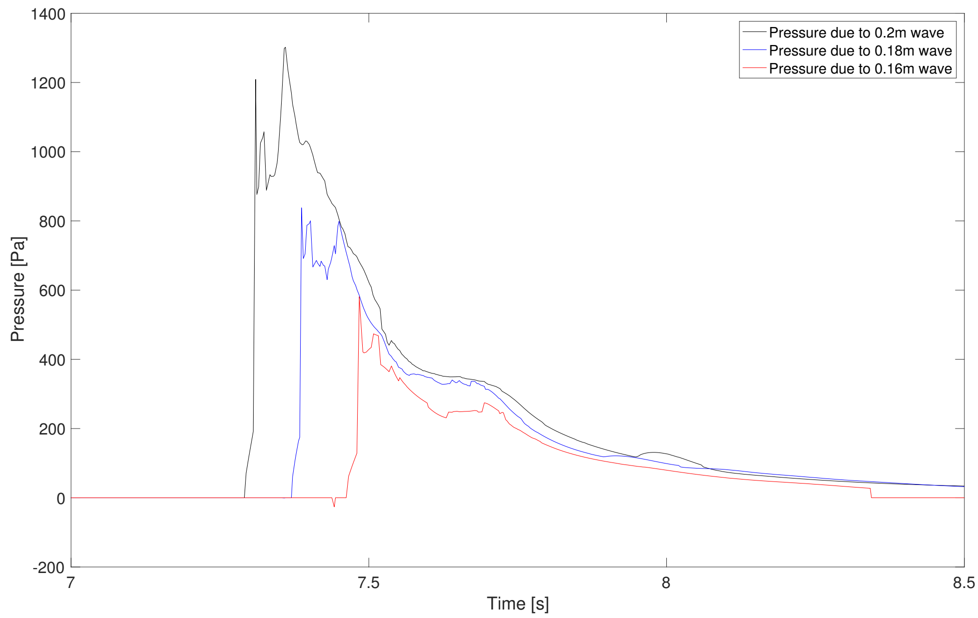


Figure 6.3: Impact pressure for wave period of 1.342s for three waves

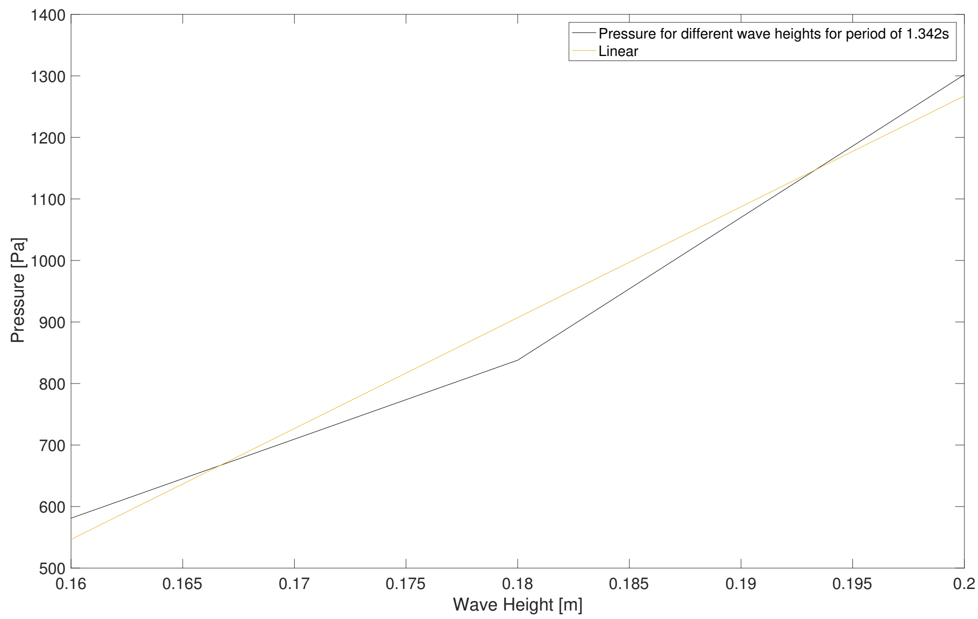


Figure 6.4: Impact pressure with wave height relation for wave period of 1.342s

Three wave heights are compared which are kept similar to previous case, wave period is kept constant i.e. 1.342s. The impact peaks are compared to different wave heights. Only the first green water loading is shown in figure 6.3 which occurs from 7 to 8.5s into the simulation. The black pressure line indicates the pressure generated by a 0.2m wave, while blue and red line represents pressure for 0.18m and 0.16m wave height respectively for same wave period. It is observed that pressure generated by a 0.2m wave is high compared to other two wave heights which are similar to results observed in the previous case.
Figure 6.4 represents the impact pressure relation for different wave heights. It is observed that the pressure is linear with linear increase in wave height which is similar to wave period of 1.142s.

6.2.3. WAVE PERIOD: 1.5S

Three wave heights are compared, wave period is kept constant i.e. 1,5s. The impact peaks are compared to different wave heights.

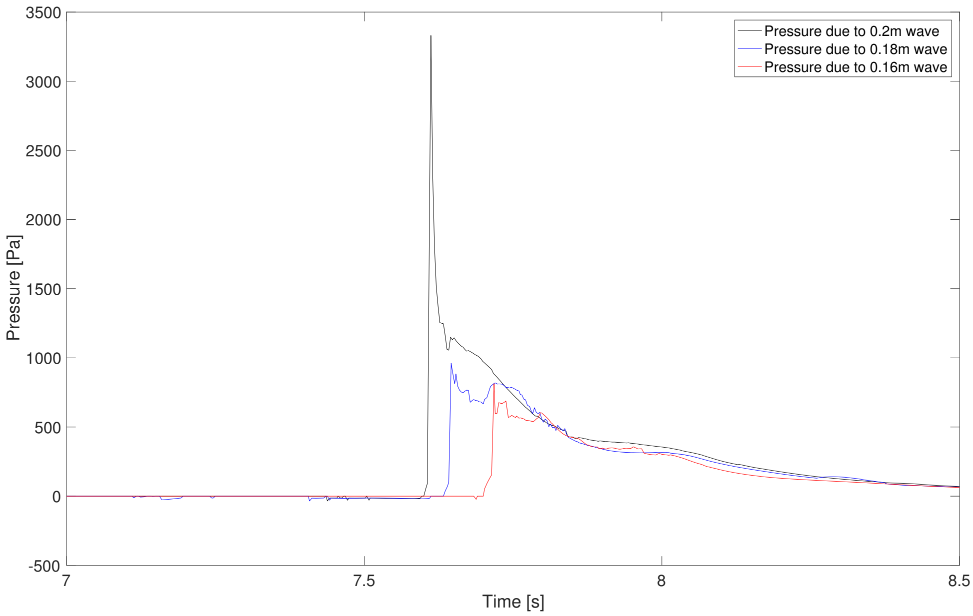


Figure 6.5: Impact pressure for wave period of 1.5s for three waves

Three wave heights are compared, wave period is kept constant i.e. 1,5s. The impact peaks are compared to different wave heights. In Figure 6.5The black pressure line indicates the pressure generated by a 0.2m wave, while blue and red line represents pressure for 0.18m and 0.16m wave height respectively for same wave period. It is observed that pressure generated by a 0.2m wave is highest compared to other two wave heights.
Figure 6.6 indicates the impact pressure relation for different wave heights. It is observed that the impact pressure is linear with linear increase of wave height. This is similar to all compared cases for regular waves and dam break simulations.

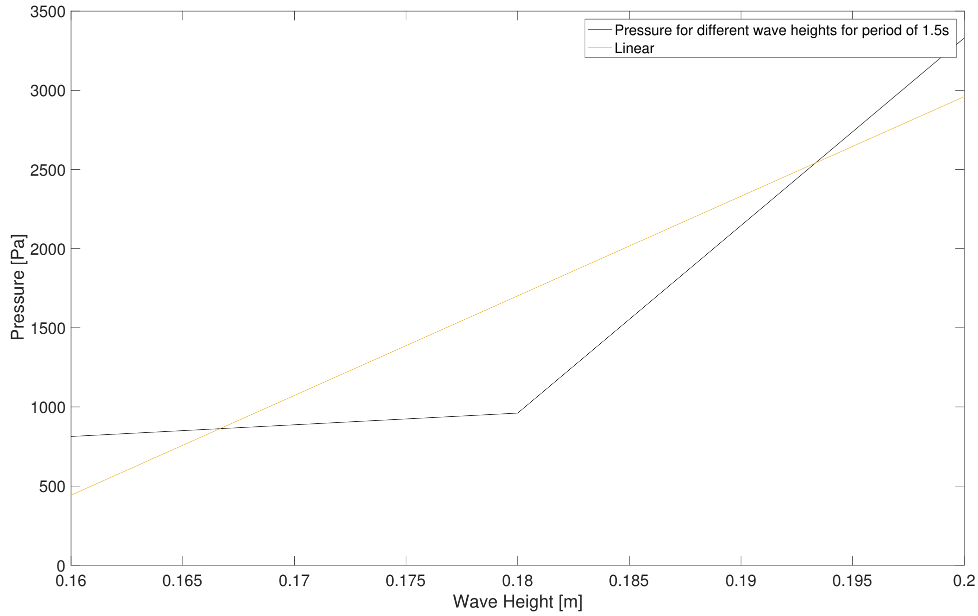
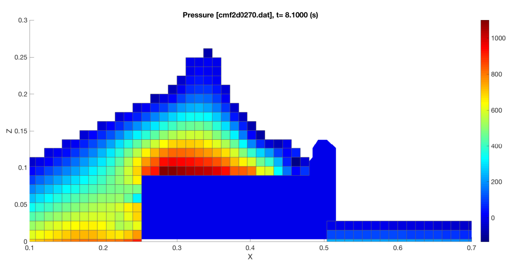


Figure 6.6: Impact pressure with wave height relation for wave period of 1.5s

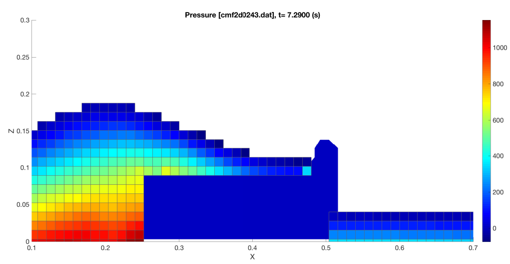
6.3. RESULTS

6.3.1. RELATION OF WAVE PERIOD TO WATER ON DECK

It is observed that the impact pressure due to 0.2m regular wave generates the maximum pressure on the structure on deck irrespective of wave periods. It is also observed that for the lowest wave period of 1.142s, maximum pressure is generated for all cases of wave height. The pressure is related to the water on deck, series of the figure represent the pressure for water on deck in ComFLOW.



(a) Water on deck for 1.142s wave period wave



(b) Water on deck for 1.342s wave period wave

Figure 6.7: Snapshots representing the pressure due to the amount of water on deck

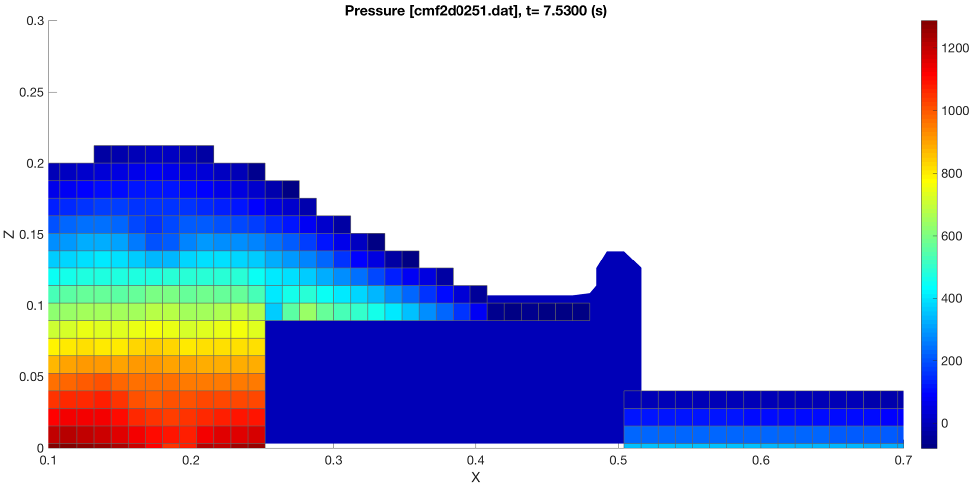
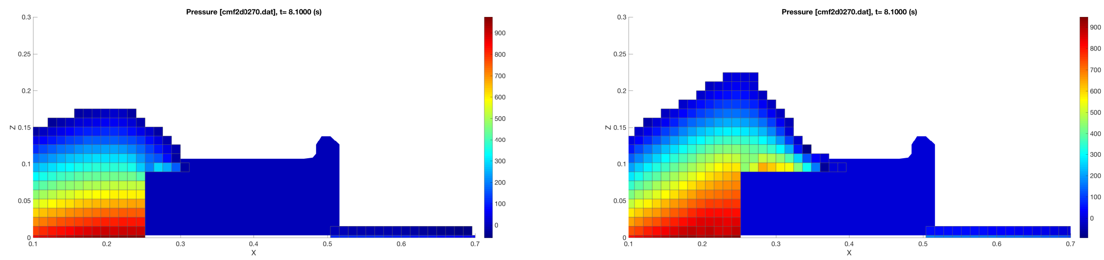


Figure 6.8: Water on deck for wave period of 1.5s

From the series of figures, it is observed that the amount of water on deck for 0.2m wave height of wave period 1.142s is the highest. Thus the pressure generated by the green water is highest compared to other simulations with increasing wave period represented in 6.7b and 6.8.

6.3.2. RELATION OF WAVE HEIGHT TO WATER ON DECK

It is observed that wave height plays an important role in the impact pressure on deck. Relation of wave height to water on deck is studied to observe if a linear change in wave height leads to a linear change in height of the water on deck. Series of ComFLOW snapshots represent the water on deck for three different wave heights i.e. 0.2, 0.18 and 0.16m for a same wave period of 1.142s.



(a) Water on deck for wave height of 0.16m

(b) Water on deck for wave height of 0.18m

Figure 6.9: Snapshots representing the pressure due to the amount of water on deck

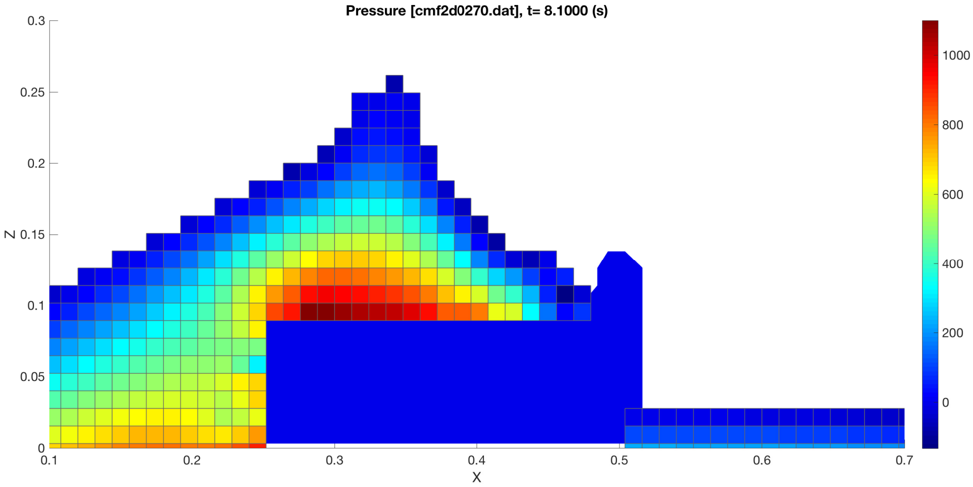


Figure 6.10: Water on deck for wave height of 0.2m

The snapshots for different wave heights are shown for time: 8.1s in the simulation. From the series of snapshots in figure 6.9a,6.9b and 6.10. It is observed that linear increase in the water height leads to a linear change of height of the water on deck. Similar results are observed with different waves of the different wave period. However, to confirm the observation from the snapshots, the height of water on deck (H_0) is plotted with the wave height in figure 6.11. Thus, it can be concluded that linear increase of wave height results to linear increase of water on deck.

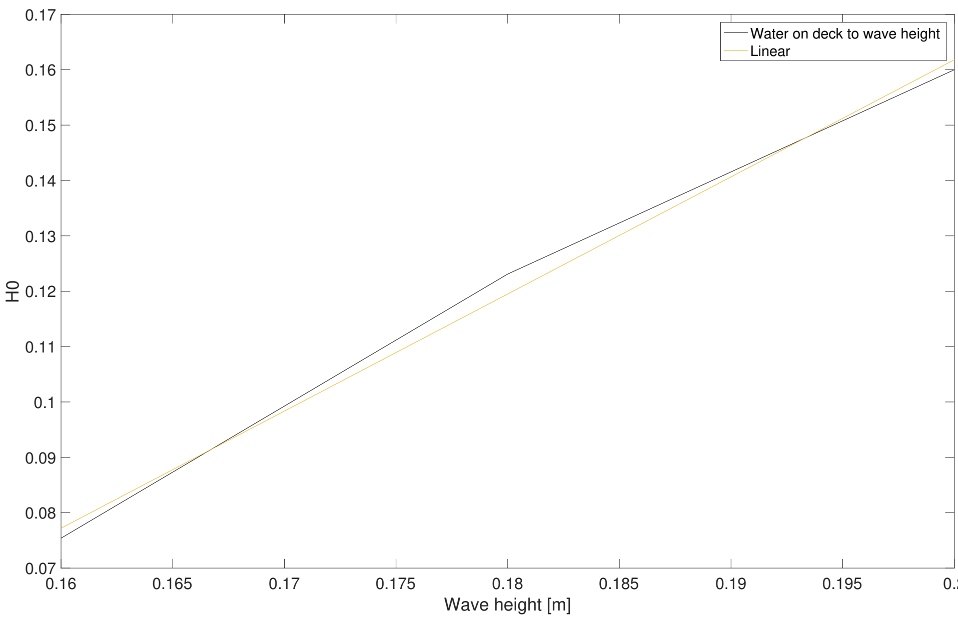


Figure 6.11: Water on deck for wave height of 0.2m

6.4. COMPARISON WITH DAM BREAK

After conducting Systematic variation of regular wave height, it is observed that there is a linear increase in the pressure on the structure on a deck which is similar to cases conducted in dam break simulations. However, impact pressure on the structure is primary concern in this research.Thus, there is a need to compare two simulations and observe the impact pressure and analyse if dam break model is a complete model to study green water loading on deck.

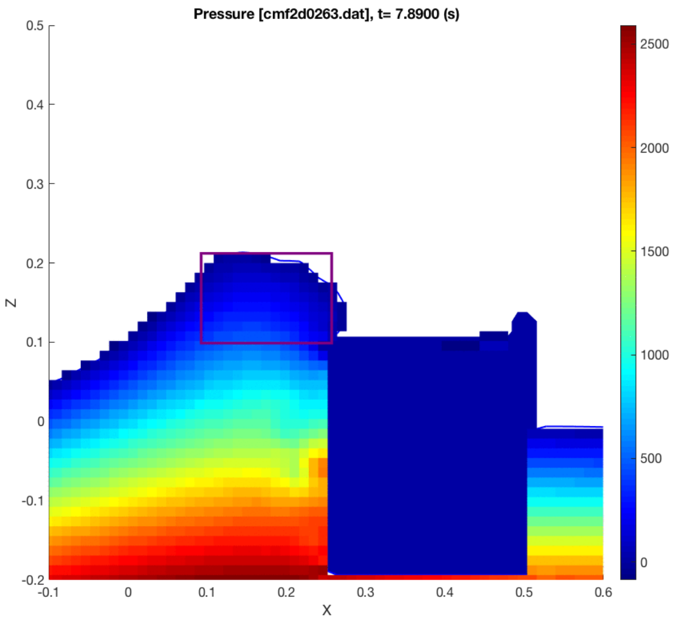
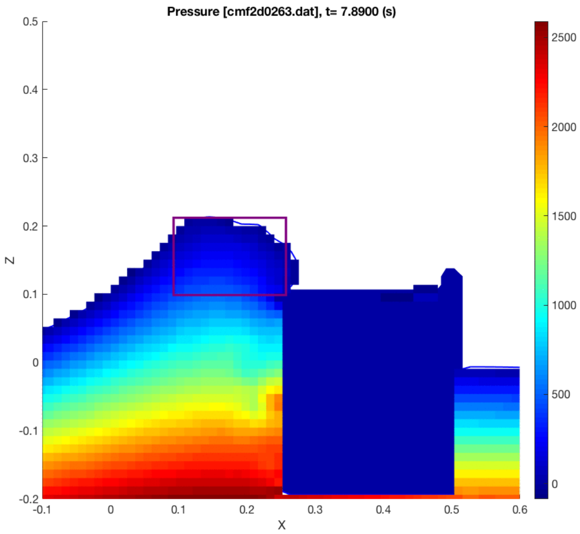


Figure 6.12: Wave height exceedance from bow

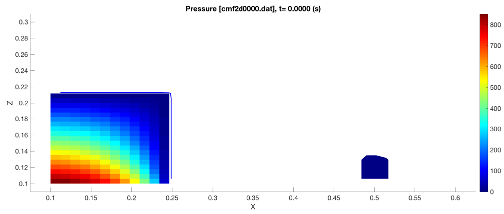
Figure 6.12 represents the water height exceedance which results in water on deck from the bow, generated from regular waves in ComFLOW domain, this water height exceedance is marked in front of the vessel in figure 6.12. This marked area is compared and generated as dam break wave and impact pressure is monitored on the structure on deck. The distance of sub structure on deck is kept the same as the case for regular waves. The marked area is calculated from this case and the length and height of water column are generated in dam break simulation. Simulation in regular waves is generated in ComFLOW for wave heights of 0.2m, 0.18m, and 0.16m with structure in the domain. First significant green water event is considered for comparison with dam break simulation. The amount of water which exceeds the bow height of the vessel is generated in ComFLOW for dam break simulation marked in figure 6.12.

6.4.1. 0.2M REGULAR WAVE COMPARISON WITH DAM BREAK SIMULATION

The amount of water exceeded for 0.2m regular wave height from the bow of the vessel is generated for comparison with dam break simulation. The marked area in figure 6.13a is generated as dam break wave in figure 6.13b. The domain for this dam break simulation is 0.5m long and 0.2m high. The distance of the structure is 0.478m in x-direction which is similar to a distance of structure on deck for regular waves. The water column obtained from the indicated area is 0.144m long and 0.1108m high. This dam break simulation is generated to obtain the impact pressure for comparison at the same monitor point at the height of 0.01m on the structure.



(a) Water exceedance for 0.2m regular wave



(b) Dam break scenario for 0.2m wave case

Figure 6.13: Regular wave scenario generated as dam break simulation

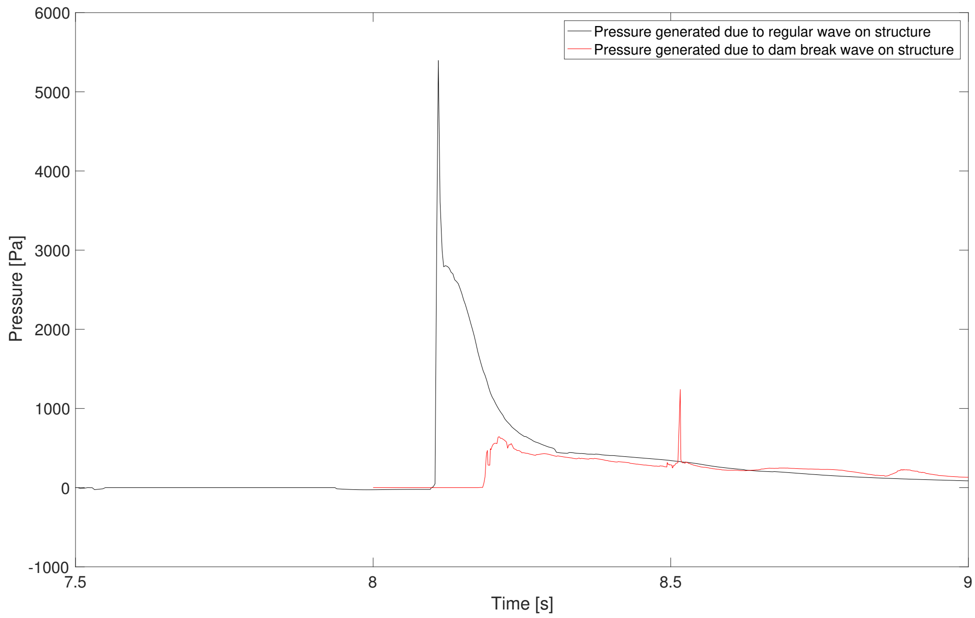
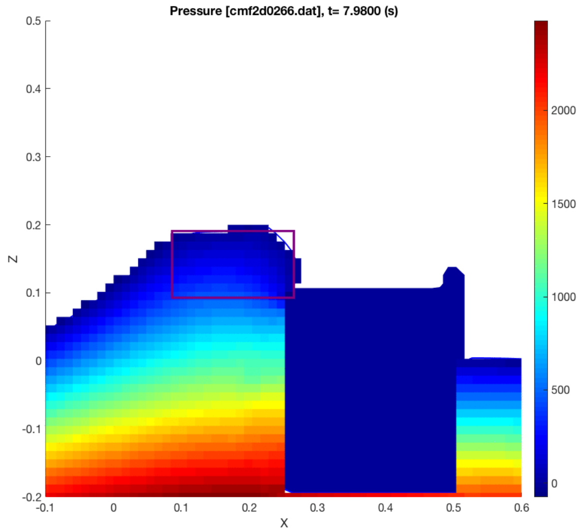


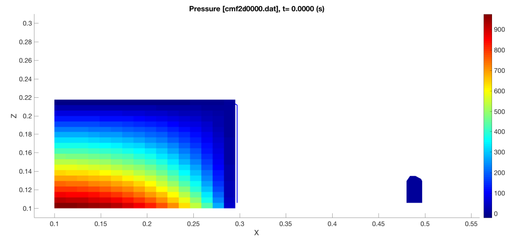
Figure 6.14: Comparison of regular wave impact with impact due to dam break wave

Figure 6.14, represents the impact pressure at the monitor point on the structure. The black pressure line indicates the pressure generated on the structure from the regular waves simulation while red line indicates the pressure generated from dam break simulation.The time for dam break simulation is adjusted according to regular wave simulation for accurate comparison of impact pressure. It is observed that there is a huge variation in the impact pressure from the two simulations. The pressure generated on the structure from regular waves is very high compared to pressure generated from dam break simulations.

6.4.2. 0.18M REGULAR WAVE COMPARISON WITH DAM BREAK SIMULATION



(a) Water exceedance for 0.18m regular wave



(b) Dam break scenario for 0.18m wave case

Figure 6.15: Regular wave scenario generated as dam break simulation

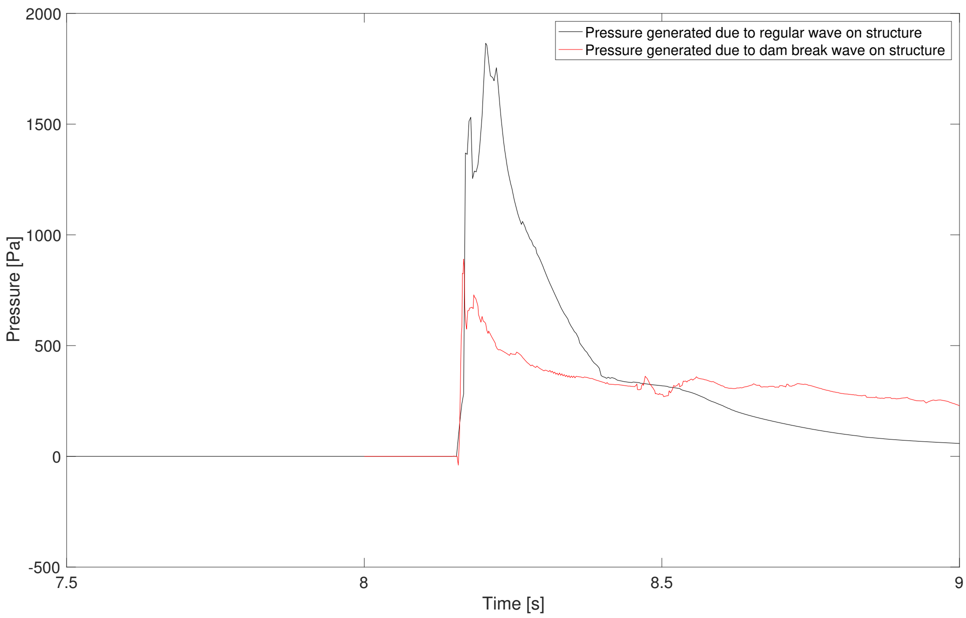


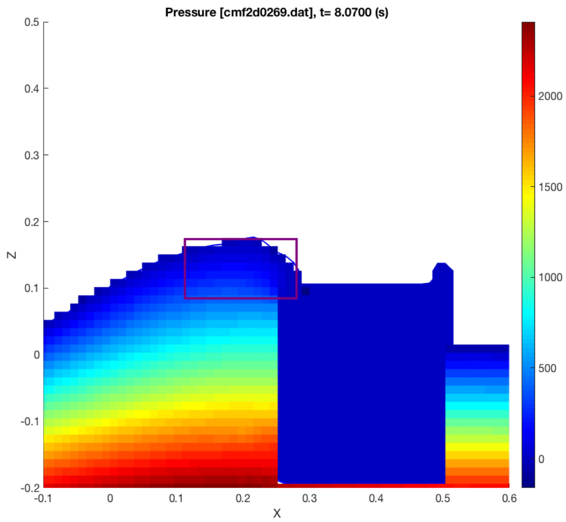
Figure 6.16: Comparison of regular wave impact with impact due to dam break wave

The amount of water exceeded for 0.18m regular wave height from the bow of the vessel is generated for comparison with dam break simulation. The marked area in figure 6.15a is generated as dam break wave in figure 6.15b. The domain for this dam break simulation is 0.5m long and 0.2m high. The distance of the structure is 0.478m in x-direction which is similar to a distance of structure on deck for regular waves. The water column obtained from the indicated area is 0.192m long and

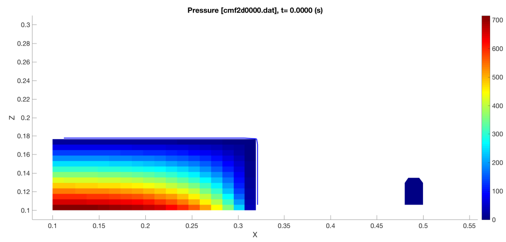
0.1123m high. This dam break simulation is generated to obtain the impact pressure for comparison at the same monitor point at the height of 0.01m on the structure.
Figure 6.16, represents the impact pressure at the monitor point on the structure. The black pressure line indicates the pressure generated on the structure from the regular waves while red line indicates the pressure generated from dam break simulation. The time for dam break simulation is adjusted according to regular wave simulation for accurate comparison of impact pressure. It is observed that there is a huge variation in pressure from the two simulations. The pressure generated on the structure from regular waves is high compared to pressure generated from dam break simulations.

6.4.3. 0.16M REGULAR WAVE COMPARISON WITH DAM BREAK SIMULATION

The amount of water exceeded for 0.16m regular wave height from the bow of the vessel is generated for comparison with dam break simulation. The marked area in figure 6.17a is generated as dam break wave in figure 6.17b. The domain for this dam break simulation is 0.5m long and 0.2m high. The distance of the structure is 0.478m in x-direction which is similar to a distance of structure on deck for regular waves. The water column obtained from the indicated area is 0.216m long and 0.0754m high. This dam break simulation is generated to obtain the impact pressure for comparison at the same monitor point at the height of 0.01m on the structure.



(a) Water exceedance for 0.16m regular wave



(b) Dam break scenario for 0.16m wave case

Figure 6.17: Regular wave scenario generated as dam break simulation

The figure 6.18 represents the impact pressure at the monitor point on the structure. The black pressure line indicates the pressure generated on the structure from the regular waves while red line indicates the pressure generated from dam break simulation. It is observed that there is a huge variation in pressure from the two simulations. The pressure generated on the structure from regular waves is high compared to pressure generated from dam break simulations. The time for dam break simulation is adjusted according to regular wave simulation for accurate comparison of impact pressure.

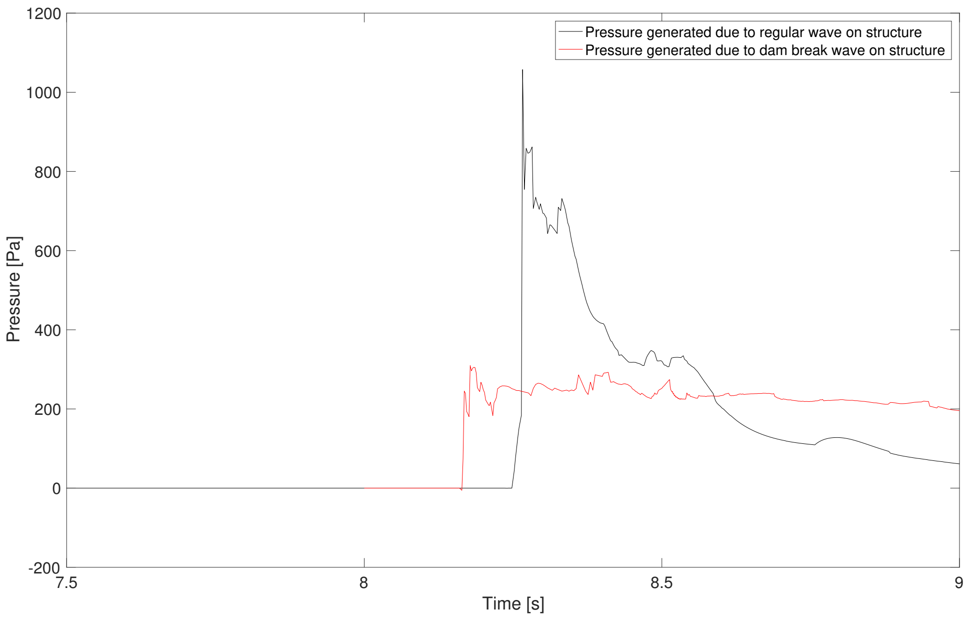


Figure 6.18: Comparison of regular wave impact with impact due to dam break wave

The results for all three comparisons are similar, the pressure generated from regular waves is higher for all cases compared to dam break simulations. It is due to initial velocity distribution which causes the wave impact to be higher and is present for regular waves while absent for dam break simulations in ComFLOW. It is observed that dam break model is not a complete model to study green water loading on deck but shows similarity with the theory related to wave height variation.

6.5. DISTANCE VARIATION OF STRUCTURE IN REGULAR WAVE

The pontoon structure is placed in the domain to observe the green water events. Regular Stokes wave of 5^{th} order is generated in the domain using ComFLOW. The duration of the simulation is 180s, such that one green water event is observed in one simulation. The structure is idealized as the vessel in an offshore location. The Substructure is placed on top of the vessel to obtain the pressure values generated by the green water on deck, the distance of substructure is varied to obtain the pressure relationship. The input parameters for numerical simulation can be observed in Table 6.2.

The Comflow domain, in this case, is increased to 100m long, 30m wide and 30m in height. The water depth is kept to be 15m, however, regular waves of 2.5m high are generated keeping in mind realistic offshore conditions. The grid resolution value is $200 \cdot 1 \cdot 200$ for the domain since it is a 2 dimensional scenario. A monitor point is placed in front of the substructure to obtain the pressure and velocity of the green water. Absorbing boundary conditions are used to minimize the numerical reflection in the domain as explained earlier in section 6.1. The regular waves are generated head on to the bow of vessel keeping zero angle of incidence.

Important Wave Parameters in ComFLOW	
Wave Period	6 s
Wave Height	2.5 m
Water Depth	15 m
Position of crest	1 m
Phase offset	0.1160 rad
Wave number	0.1160
Wave velocity	9.024 m/s
Time duration	180 s
Time step	0.1 s
CFL Number (Min - Max)	0.3 - 0.7

Table 6.2: ComFLOW Input Parameters

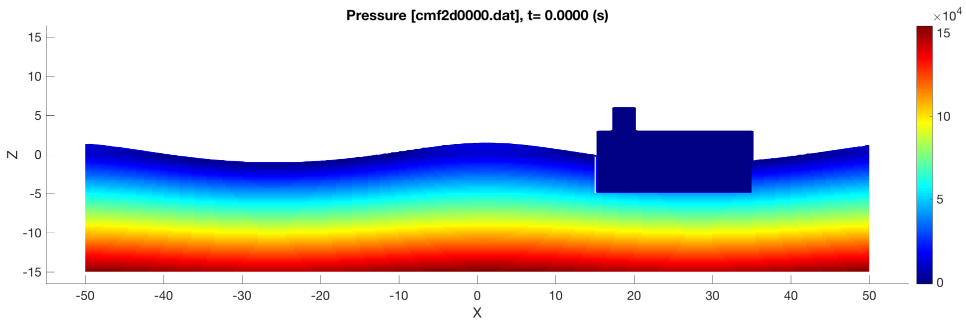
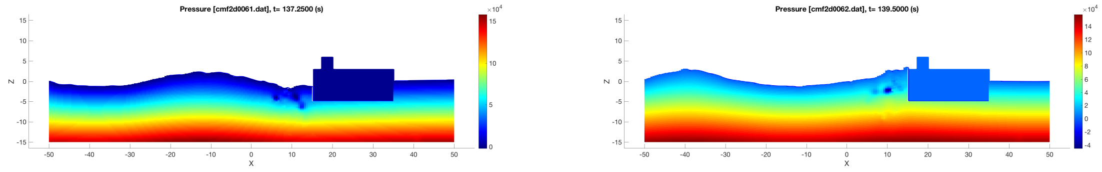


Figure 6.19: Idealized vessel in regular wave at t=0s

The figure 6.19 represents idealised vessel in the ComFLOW domain at a distance of +15m in the x-direction, however, the substructure is placed initially at 2m from the bow of the vessel. This distance is varied and pressure relation is calculated to perform the systematic variation. The draft height of the vessel is 5m. A thin blue line between the vessel and wave represents the wave-structure interaction. The series figure 6.20 indicates the green water event in the numerical simulation which occurs at t=130.5s when the wave hits the deck.



(a) (b)

Figure 6.20: Comflow simulation snapshot for pressure indicating green water event in the simulation

The figure 6.21 shows pressure exerted by the green water on the deck of a vessel on the substructure. Highest peak can be observed at t=140.9s, however other small peaks later in time are due to the fact that water still remains on the deck. It is captured by the monitor point located 0.25m in front of the substructure to observe maximum accuracy in pressure with respect to grid refinement for this simulation.

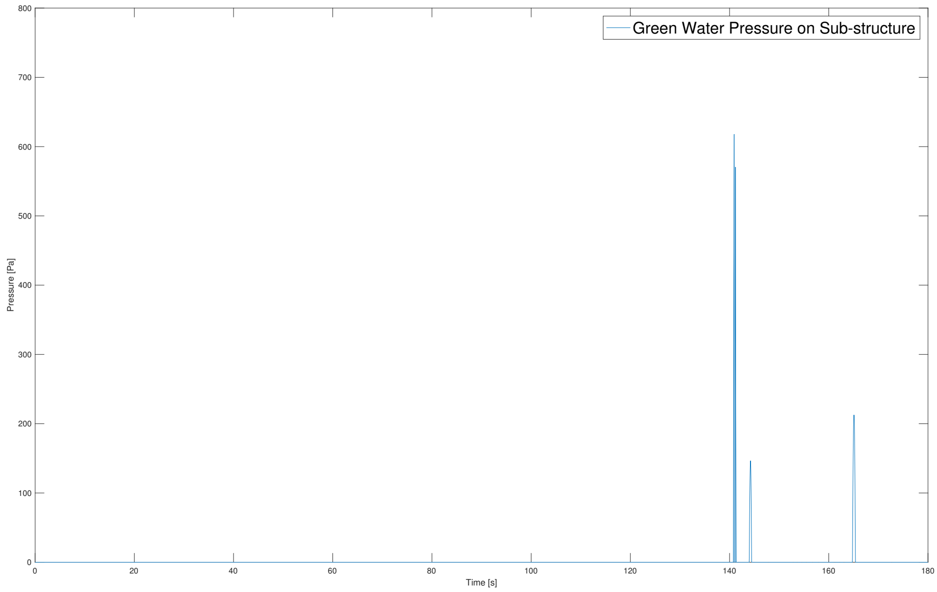


Figure 6.21: Green water pressure on substructure

The influence of wind on the green water phenomenon is null. Therefore the wind is not simulated in the numerical simulations. The green water occurrence in this scenario can be briefly described as a sequence of events which has been described by Buchner[16] :

1. Wave motion causes water above freeboard of the pontoon.
2. The flow of water on to the deck.
3. A shallow water wave flows over the deck.
4. The water hits the structure on deck.

6.5.1. DISTANCE VARIATION OF SUBSTRUCTURE ON DECK

Input parameters remain the same, a distance of the substructure on deck is varied from 2m to 10m on the idealized vessel pontoon. Same regular waves are generated and the monitor point is shifted in the x-direction with respect to the placement of substructure on deck to capture pressure values. The obtained values are plotted against varied distance in x-direction and relationship between distance and pressure is obtained.

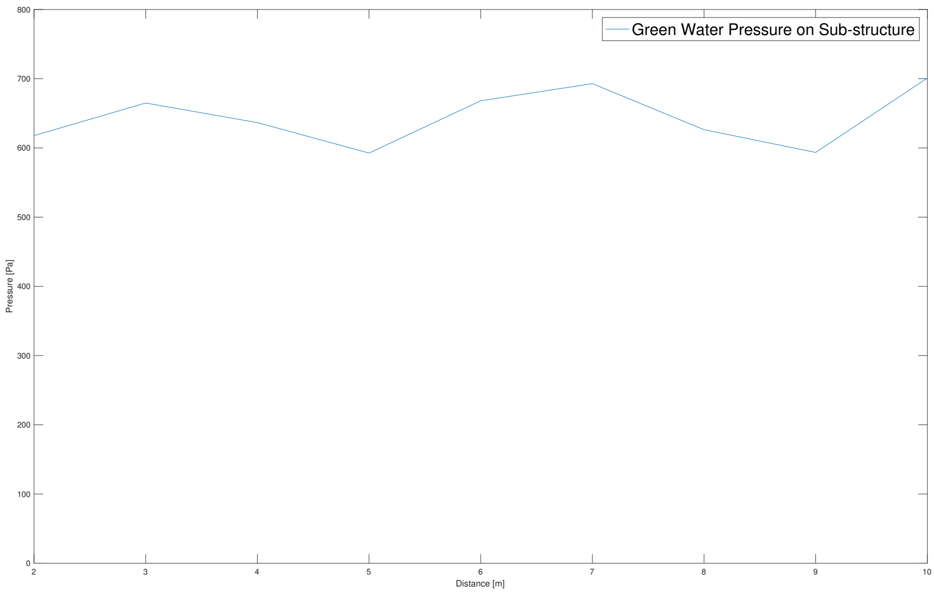


Figure 6.22: Distance vs Pressure

The figure 6.22 explains the behavior of generated pressure by green water events over a variation of distance on a deck of an idealized vessel. The constant behavior of impact pressure is observed which is similar to results in dam break wave cases.

6.6. CONCLUSION

Study of structure in regular waves is performed to validate if dam break model is an appropriate model for studying green water loading on the structure. Variation of time period concludes that waves with lesser time period have more impact on structures while waves with higher wave heights have a higher impact. The impact pressure is linear with linear increase of wave height for all cases conducted. The water on deck is also observed to increase with an increase in wave height. It is also observed that loading due to regular waves when compared with dam break waves have vast difference in pressure impact on the structure. The variation of distance of the substructure on deck gives similar results to that of dam break results i.e. constant behaviour of impact pressure with an increase in distance. With these observations, it can be concluded that dam break model is not a complete model for studying green water loading for regular waves since the pressures variation is quite large when compared with dam break simulations. However, with variation in wave height and distance of the structure on deck similar results are observed.

7

STRUCTURE IN IRREGULAR WAVES

Irregular waves in the domain are generated which have different parameters compared to regular ones. To generate such waves, user defined parameter needs to be provided for initial liquid configuration in ComFLOW. These user waves in ComFLOW is defined as:

$$\eta(x,t) = \sum_{i=1}^{n} a(i)(cos(\omega(i)t) - k(i)xcos\beta(i) - k(i)ysin\beta(i) + \phi(i)) \tag{7.1}$$

Here,

ω = Natural frequency of the wave

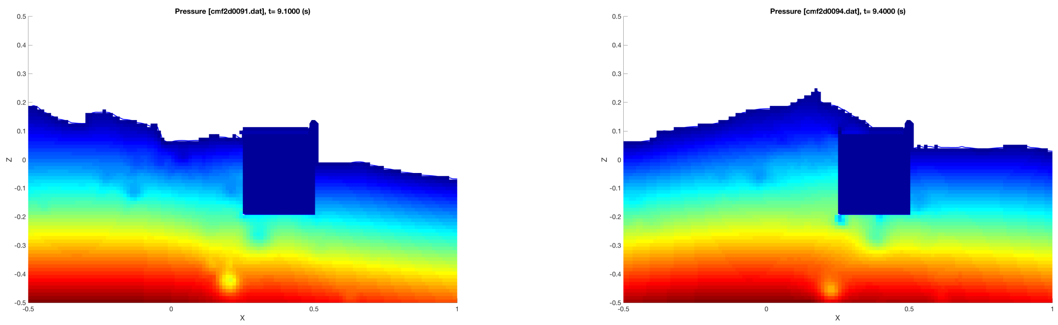
ϕ = Phase of the wave

β = Angle of incidence of the wave

k = wave number

a = wave amplitude

Irregular waves of different amplitudes are generated along with their respective wave number k and phase ϕ. An angle of incidence β is zero to keep waves head on to the bow of the pontoon structure. The wave period and water depth are varied for two kinds of systematic variation and with same absorbing boundary conditions similar to the simulations with regular waves. It follows the Jonswap spectrum for irregular waves in far offshore location.



(a) (b)

Figure 7.1: Comflow simulation snapshot for pressure indicating green water event for irregular wave

7.1. VARIATION OF WAVE HEIGHT AND WAVE PERIOD FOR STRUCTURE IN IRREGULAR WAVES

The pontoon structure is placed in ComFLOW domain to study green water loading on deck caused by irregular waves. Nine simulations are generated with a variation of wave height and wave period respectively. The domain is 12m long, 1m wide and 1.6m high and a water depth of 0.6m. Absorbing boundary conditions are used at the end of a domain to prevent wave reflections within the domain. The grid cells used are 0.01m in x and z-direction i.e. i_{max} and k_{max}= 1000 and 130 respectively. The wave height and wave period for different simulations are provided in table 7.1. Three wave heights with three different time period are generated on pontoon like structure to observe green water loading on the structure on the deck. Monitor points and force box are assigned on the structure to capture impact pressure due to green water. Wave period is selected such that no wave breaking is observed in the simulations.

Wave height and wave periods in ComFLOW	
Significant Wave Height	0.222m
Wave Period	2.0, 2.2, 2.4
Significant Wave Height	0.259m
Wave Period	2.0, 2.2, 2.4
Significant Wave Height	0.296m
Wave Period	2.0, 2.2, 2.4
Time duration	10s
Time step	0.01s
CFL Number (Min - Max)	0.15 - 0.4

Table 7.1: Irregular wave input parameters

Three different wave periods of 2.0, 2.2 and 2.4s are generated for wave heights of 0.222, 0.259 and 0.296m. The distance of the structure on deck remains constant for these variations. The CFL_{min} and CFL_{max} is kept 0.25 and 0.4 respectively. The time duration is kept 30s which is long enough to observe various green water events but the first significant green water is considered for comparison.

7.2. WAVE PERIOD: 2.0S

This case indicates the pressure generated by the wave with time period of 2.0s for three different significant wave heights of 0.222m, 0.259m and 0.296m. Figure 7.2 represents the pressure generated on deck. The simulation is generated for 15s however, only first significant green water is of importance which occurs at 8s into the simulation. Thus, the impact peaks are refined in time for better observation.

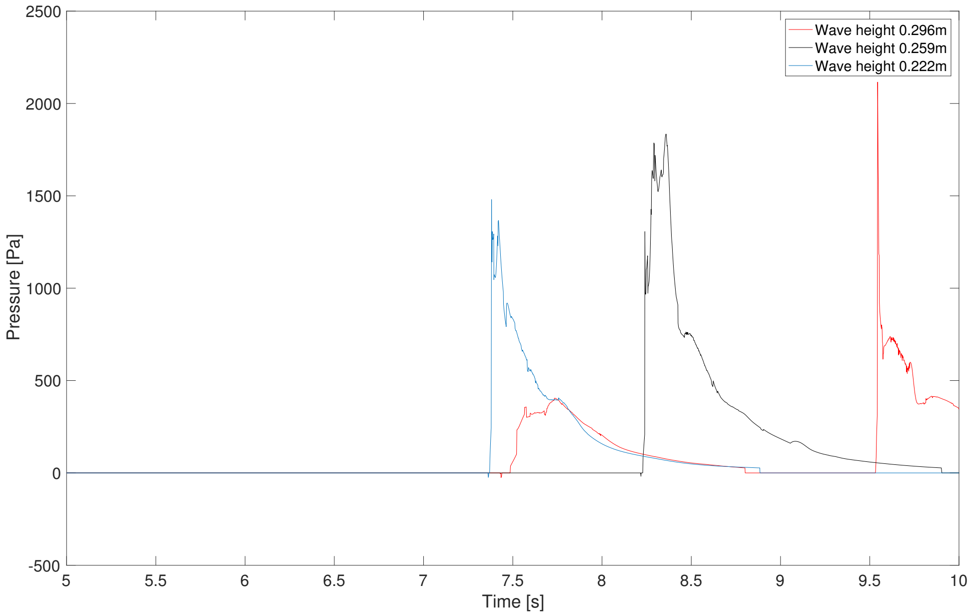


Figure 7.2: Impact pressure for wave period of 2.0s for three waves

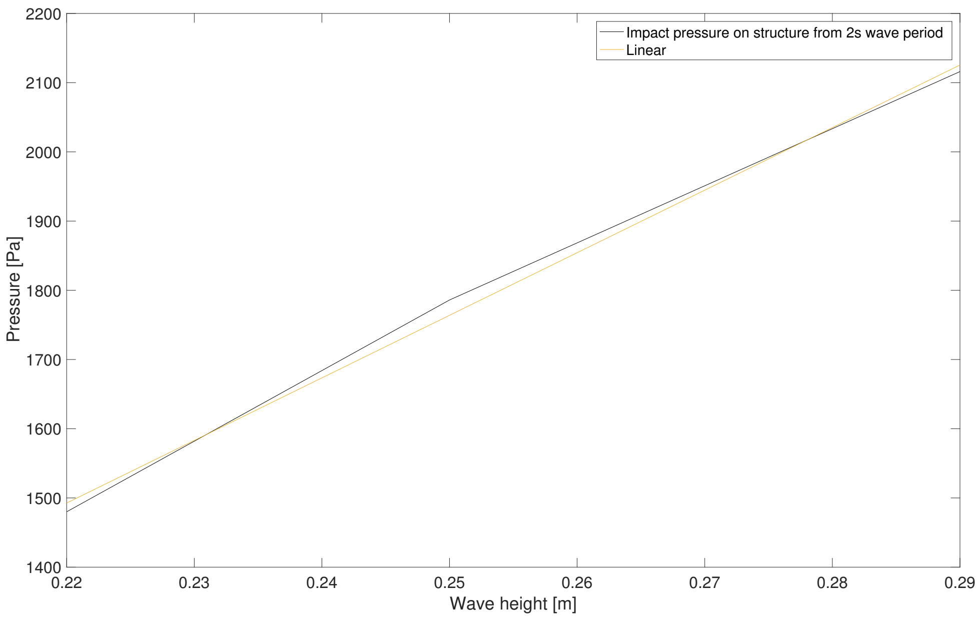


Figure 7.3: Linear impact pressure for wave period of 2s for three wave heights

In figure 7.2, the blue pressure line indicates the pressure generated by 0.222m wave, while the black and red line represents the pressure for 0.259m and 0.296m significant wave heights respectively. It is observed that the pressure generated by a 0.296m wave is largest compared to two other waves for 2.0s wave period. The figure 7.3 indicates the pressure over wave height, a linear increase in impact pressure is observed with the linear increase of significant wave height with constant wave period.

7.3. WAVE PERIOD: 2.2S

Three wave heights are compared which are kept same compared to the previous case, wave period is kept constant i.e. 2.2s. The impact peaks are compared to different wave heights. Only the first green water loading is shown in figure 7.4 which occurs from 5 to 10s into the simulation for all three waves. The blue pressure line indicates the pressure generated by 0.222m wave, while the black and red line represents the pressure for 0.259m and 0.296m wave height respectively for same wave period. It is observed that pressure generated by a 0.296m wave is highest compared to other two wave heights which are similar to results observed in the previous case.

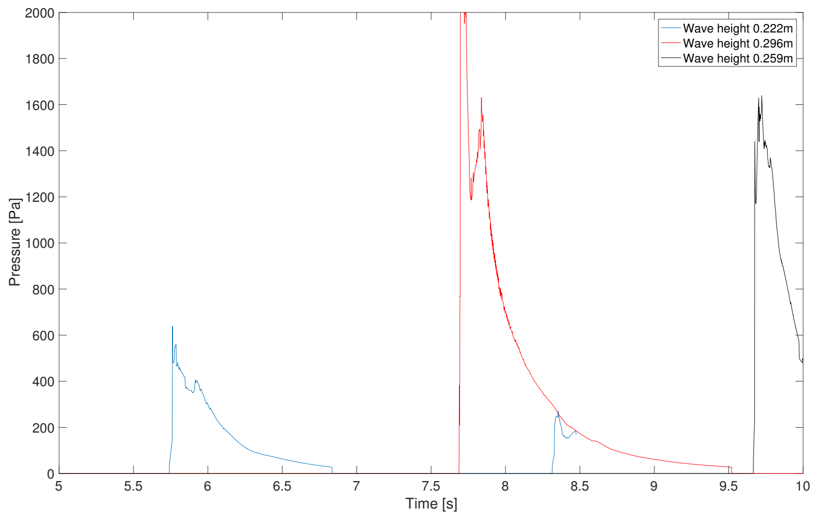


Figure 7.4: Impact pressure for wave period of 2.2s for three waves

Figure 7.5 represents the linear behaviour of impact pressure for three wave heights with wave period of 2.2s. It is similar to case with 2s wave period and with simulations involving regular waves.

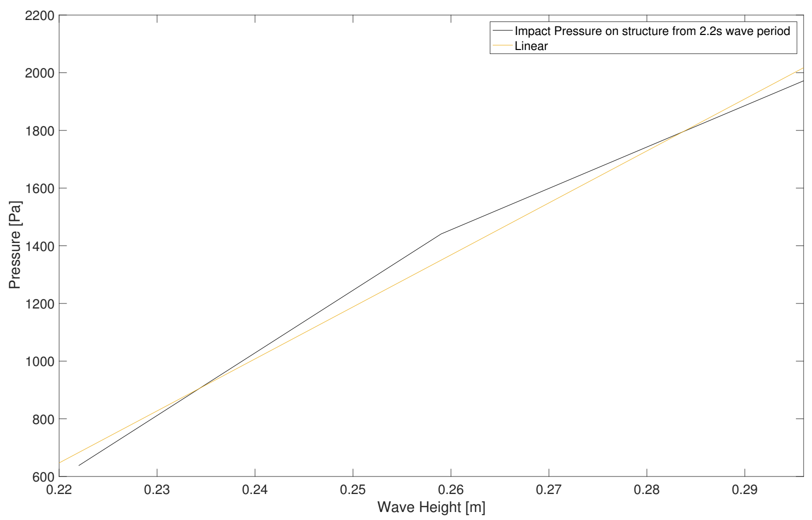


Figure 7.5: Linear behaviour of impact pressure of three wave heights for 2.2s wave period

7.4. WAVE PERIOD: 2.4S

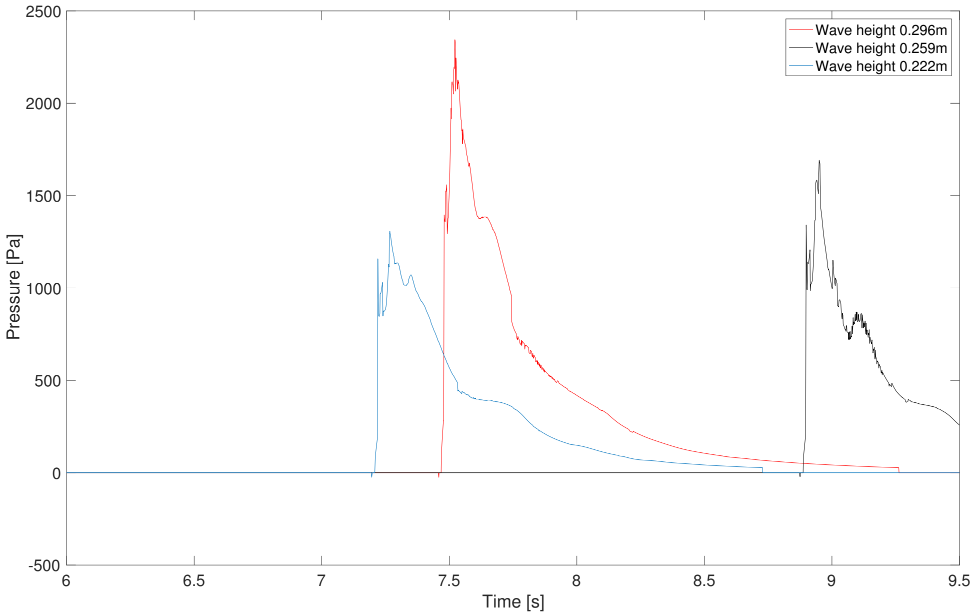


Figure 7.6: Impact pressure for wave period of 1.5s for three waves

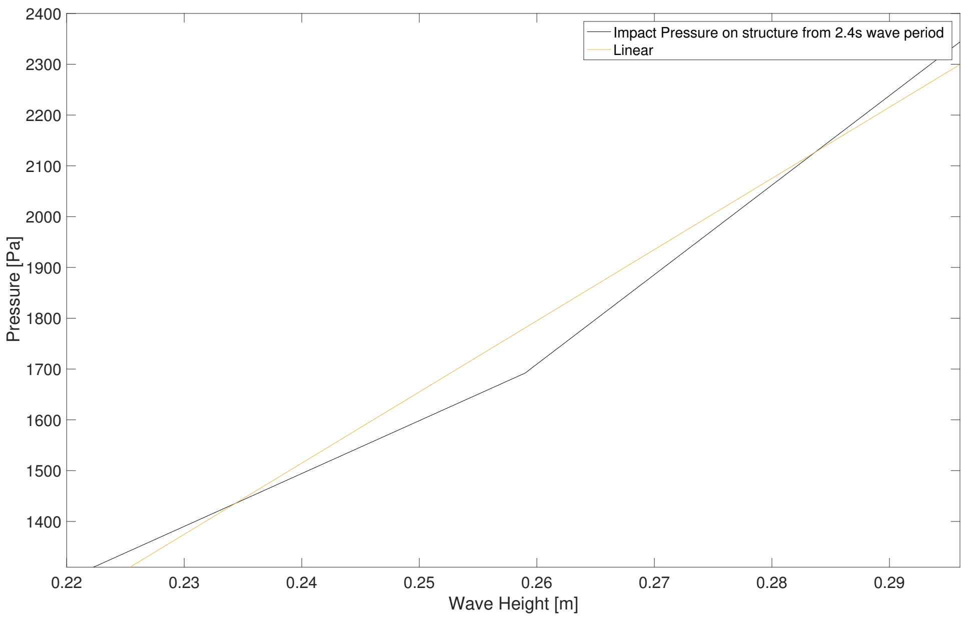


Figure 7.7: Linear impact pressure for wave period of 2.4s for three wave height

Three wave heights are compared, wave period is kept constant i.e. 2.4s. The impact pressure peaks are compared to different wave heights. The blue pressure line indicates the pressure generated by 0.222m wave, while the black and red line represents the pressure for 0.259m and 0.296m wave height respectively for same wave period. It is observed that the pressure generated by a 0.296m wave is highest compared to other two wave heights. It is observed that there is a linear increase in pressure with increase in irregular wave height shown in figure 7.7:

It can be concluded that the impact pressure generated for the least wave period (2.0s) is highest compared to two different compared wave periods. It is a similar result observed from simulations in regular waves. It is also concluded that the linear increase in significant wave height results in the linear increase in impact pressure on the structure on deck.

7.5. VARIATION IN DISTANCE OF STRUCTURE ON DECK

The Comflow domain, in this case, is increased to 100m long, 30m wide and 30m in height, it is similar to distance variation for regular waves. The water depth is kept to be 15m, however, irregular waves of 2.0m high are generated to observe realistic offshore conditions. The grid size is 0.01m for x and z-direction for the domain since it is a 2-dimensional scenario. A monitor point is placed in front of the substructure to obtain the pressure of the green water loading. Absorbing boundary conditions are used to minimize the numerical reflection in the domain. The irregular waves are generated head on to the bow of vessel keeping zero angle of incidence. Parameters used to generate irregular waves in a domain are described in table 7.2.

Important Irregular Wave Parameters in ComFLOW	
Peak Period	6 s
Wave Height	2.0 m
Water Depth	15 m
Wave Number	0.1184
Wave length	53.073 m
Wave Velocity	8.846 m/s
Position of crest	1 m
Phase offset	0.1184 rad
Time duration	400 s
Time step	0.1 s
CFL Number (Min - Max)	0.3 - 0.7
Type of Boundary condition	Absorbing BC

Table 7.2: ComFLOW Input Parameters for generation of irregular waves

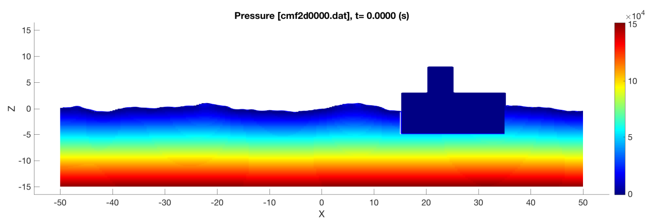
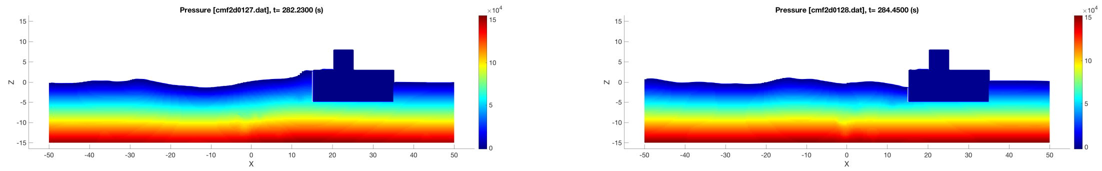


Figure 7.8: Idealized vessel in irregular wave at t=0s

Series of a figure in 7.9 represents the green water event at 282s which has pressure impact on the substructure kept on the deck which is monitored by placing the monitor point 0.25m in front.



(a) (b)

Figure 7.9: Comflow simulation snapshot for pressure indicating green water event for irregular waves in the simulation

Figure 7.10 represents pressure due to green water event in the simulation for substructure kept 5m in the x-direction from the bow of the pontoon. We observe, for a duration of 400s one significant green water event hits the structure which is represented by the peak pressure in figure 7.10.

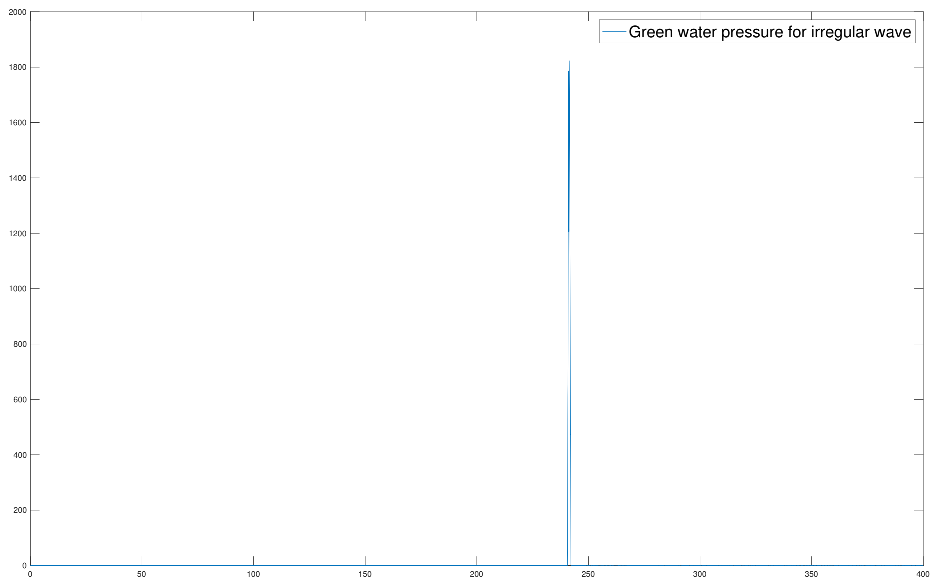


Figure 7.10: Idealized vessel in irregular wave at t=0s

7.5.1. DISTANCE VARIATION OF SUBSTRUCTURE ON DECK

Similar to a variation of the substructure for regular waves, the same procedure is performed for irregular waves. The sub-structure on the pontoon is varied in the x-direction from 2m to 10m. Green water event is monitored for the duration of simulations performed. Pressure is observed acting on the substructure along with its respective velocities.

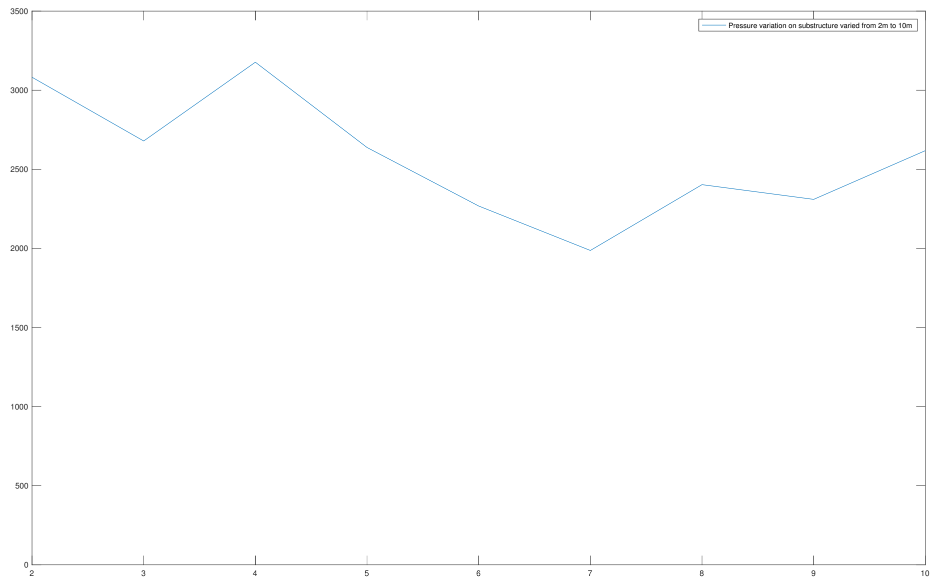


Figure 7.11: Impact Pressure with distance variation on deck for irregular waves

Figure 7.11 indicates pressure for a varied distance on deck. It is observed that impact pressure is independent of the distance of the structure which is similar to cases with regular waves and Dam-break simulations.

7.6. CONCLUSION

Conclusions can be made from this chapter for structure in irregular waves. One or more significant green water event is observed for cases of irregular waves. It is observed that the impact pressure is linear with linear increase of wave height which is similar to dam break simulations and simulations with regular waves. Impact pressure is independent of the distance of a structure in regular waves i.e. the impact pressure over a distance variation is constant.

8

Conclusions and Recommendations

Numerical simulations for methodology and validation have been conducted. Conclusion and recommendations are presented in this chapter from the results obtained in the previous chapters. The main aim of the conclusion is to answer the thesis objective while recommendations are provided to give further insights into this research. The main objective of this research was to study Offshore green water and how it impacts the structure kept on deck.

8.1. Conclusions

- **Grid Dependence:** The main objective of numerical solution is to provide most accurate results when the grid is refined.The software used, ComFLOW does not provide grid independence. The accuracy of the simulation is not improved with refined grid size. In point of fact, many uncertainties arise in simulations and very high peak pressure and force values arise which can not be trusted. Water droplets are observed in the domain, which causes these uncertainties.
- **Verification of ComFLOW:** ComFLOW is verified for simulations in 2-dimensions with kleefsman's 3-dimensional simulations. It was concluded that ComFLOW offers similar results for 3-dimensional and 2 dimensional simulations.
- **Grid Selection:** Grid cell size of 0.01m for the dam break simulations, regular and irregular waves is selected for accuracy. This grid size offers better comparative and stable values to other grid sizes and follows the same trend of peak, stationary and stability phase as observed by Kleefsman in her numerical simulations.
- **Sensitivity of CFL_{min} and CFL_{max}:** It is observed that ComFLOW is sensitive to CFL_{min} and CFL_{max} numbers. Accurate and stable results are obtained when these numbers are kept below 0.5 rather than 1.
- **Variation of distance in Dam break simulations:** It is concluded that the impact pressure on the structure due to dam break wave is independent of the distance of the structure. Constant pressure behaviour is observed which is similar to that observed by Buchner.
- **Variation of Dam-break height:** It is concluded that impact pressure due to a linear variation of dam break height shows a linear increase in impact pressure.
- **Variation of wave height in Regular wave:** It is concluded that there is a linear increase in impact pressure on the structure on a deck with a linear increase of wave height. It is observed that the wave with least wave period has maximum impact pressure on a structure.

- **Distance variation of the structure on deck in regular wave:** Constant impact pressure is observed with variation in the distance on a deck which is similar to dam break simulations.
- **Comparison of regular waves with dam break simulation:** It is observed that there is a vast difference in impact pressure from the two compared simulations. Thus, dam break model is not an ideal and complete model for studying green water phenomenon in regular waves since the pressure upon impact from a wave is not accurate.
- **Variation of wave height in Irregular wave:** It is concluded that there is a linear increase in impact pressure with an increase in wave height, Similar to cases with dam break and regular wave simulation.
- **Distance variation of the structure on deck in irregular wave:** It is concluded that impact pressure on a structure with the variation in distance has a constant behaviour over distance.

8.2. RECOMMENDATIONS FOR FUTURE RESEARCH

- Experiments should be conducted for dam break waves, variation of distance can be done to confirm the relationship between wave height and impact pressure.
- Numerical simulations should be conducted in the 3-Dimensional domain which is regulated in a 2-Dimensional domain in this research due to computational time constraints.
- Ship motions should be considered while performing numerical simulation and experiment for structure in regular and irregular waves.
- Initial velocity distribution should be induced in dam break simulations for accurate impact pressure calculations.

Bibliography

[1] K. Kleefsman, G. Fekken, A. Veldman, B. Iwanowski, and B. Buchner, *A volume-of-fluid based simulation method for wave impact problems,* Journal of computational physics **206**, 363 (2005).

[2] *Marin green water loading,* http://www.marin.nl/web/Research-Topics/Loads-responses/Green-water-loads.htm (2014).

[3] S. Tan, *Observations made on board of dutch merchant ships,* Int. Shipbuilding Progress **16**, 259 (1969).

[4] *Discovery series deadliest catch,* https://www.discovery.com/tv-shows/deadliest-catch/ (2005).

[5] W. Morris, J. Millar, and B. Buchner, *Green water susceptibility of north sea fpso/fsus,* in *Proceedings of the 15th Conference on Floating Production Systems (FPS), London* (2000).

[6] G. Ersdal, A. Kvitrud, *et al., Green water on norwegian production ships,* in *The Tenth International Offshore and Polar Engineering Conference* (International Society of Offshore and Polar Engineers, 2000).

[7] D. Enright, R. Fedkiw, J. Ferziger, and I. Mitchell, *A hybrid particle level set method for improved interface capturing,* Journal of Computational physics **183**, 83 (2002).

[8] F. H. Harlow and J. E. Welch, *Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface,* The physics of fluids **8**, 2182 (1965).

[9] C. W. Hirt and B. D. Nichols, *Volume of fluid (vof) method for the dynamics of free boundaries,* Journal of computational physics **39**, 201 (1981).

[10] S. Osher and J. A. Sethian, *Fronts propagating with curvature-dependent speed: algorithms based on hamilton-jacobi formulations,* Journal of computational physics **79**, 12 (1988).

[11] H.-C. Chen *et al., Time-domain simulation of nonlinear wave impact loads on fixed offshore platform and decks,* International Journal of Offshore and Polar Engineering **20** (2010).

[12] J. Gerrits and A. Veldman, *Dynamics of liquid-filled spacecraft,* Journal of Engineering Mathematics **45**, 21 (2003).

[13] G. E. Loots, *Fluid-structure interaction in hemodynamics,* (2003).

[14] R. Newton, *Wetness related to freeboard and flare,* Transaction RINA **102**, 49 (1960).

[15] O. Faltinsen, M. Greco, and M. Landrini, *Green water loading on a fpso,* Transactions-American Society of Mechanical Engineers Journal of Offshore Mechanics and Arctic Engineering **124**, 97 (2002).

[16] B. Buchner *et al., The impact of green water on fpso design,* in *Offshore technology conference* (Offshore Technology Conference, 1995).

[17] I. Watanabe, *Remarks on influence of bow flare form on deck wetness,* in *Proceedings* (1990) pp. 167–172.

[18] M. Greco, *A two-dimensional study of green-water loading,* Ph.D. thesis, Fakultet for ingeniørvitenskap og teknologi (2001).

[19] M. Barcellona, M. Landrini, M. Greco, and O. Faltinsen, *An experimental investigation on bow water shipping,* Journal of ship research **47**, 327 (2003).

[20] *Marin comflow project plan,* http://www.marin.nl/web/JIPs-Networks/Archive-JIPs-public/Comflow-3.htm (2009).

[21] L. Skjelbreia and J. Hendrickson, *Fifth order gravity wave theory,* Coastal Engineering Proceedings **1**, 10 (1960).

[22] I. Orlanski, *A simple boundary condition for unbounded hyperbolic flows,* Journal of computational physics **21**, 251 (1976).

[23] P. R. Wellens, *Wave simulation in truncated domains for offshore applications,* (2012).

[24] C. Mokrani and S. Abadie, *Conditions for peak pressure stability in vof simulations of dam break flow impact,* Journal of Fluids and Structures **62**, 86 (2016).

[25] W.-L. Chuang, K.-A. Chang, and R. Mercier, *Impact pressure and void fraction due to plunging breaking wave impact on a 2d tlp structure,* Experiments in Fluids **58**, 68 (2017).

[26] R. Luppes, B. Düz, H. van der Heiden, P. van der Plas, and A. Veldman, *Numerical simulations of two–phase flow with comflow: past and recent developments,* in *Proceedings of the ECCOMAS 2012 Conference* (2012) pp. 1–16.

[27] D. F. Silva, P. T. Esperança, and A. L. Coutinho, *Green water loads on fpsos exposed to beam and quartering seas, part ii: Cfd simulations,* Ocean Engineering **140**, 434 (2017).

A

Systematic variation

A.1. Pressure and Force figures

Pressure and force graphs with their respective grid mesh size are represented in the figures below. It shows the behavior of incoming wave on the structure for every grid.

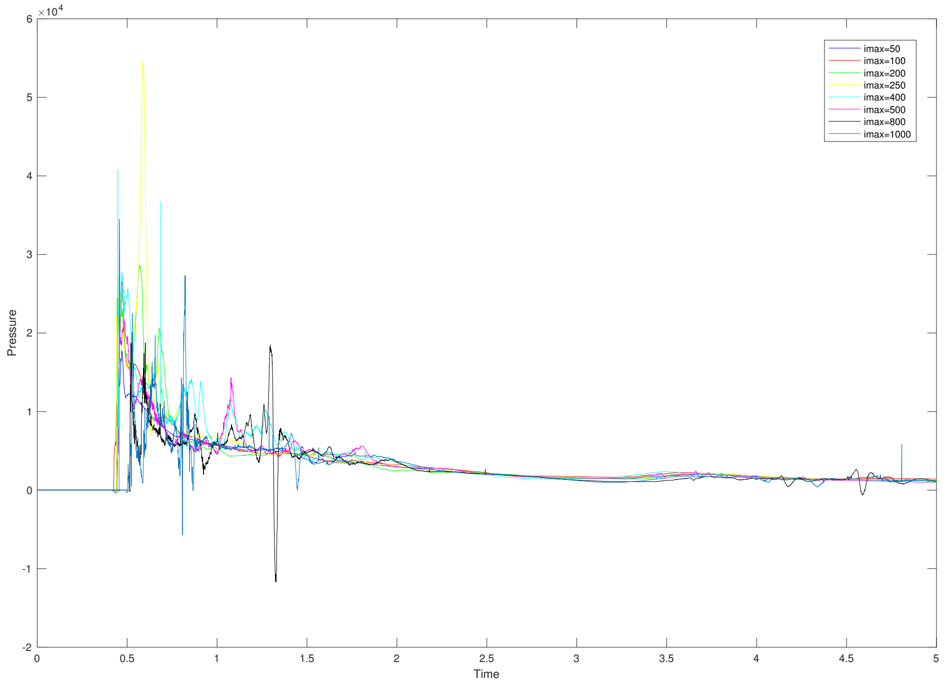


Figure A.1: Pressure values for different grid sizes

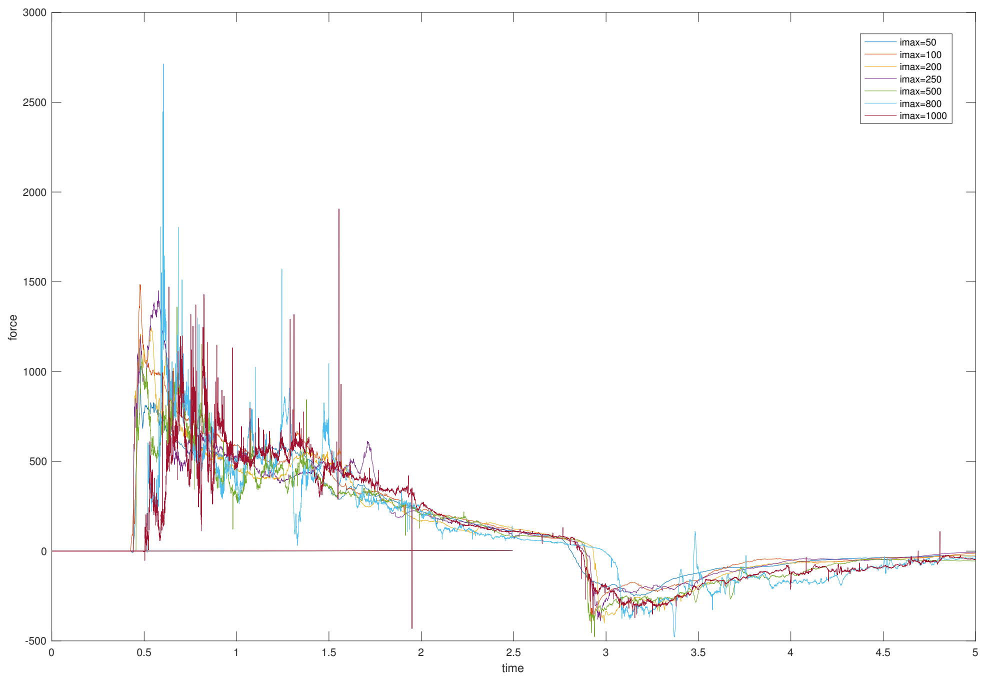


Figure A.2: Force values for different grid sizes

A.2. VARIATION OF DISTANCE FOR CFL_{max}=0.8

This table A.1 indicates the impact pressure peak obtained from every simulation performed in ComFLOW. The pressure values are obtained from the monitor point graph in ComFLOW. It is refined in time with comparison to snapshots to obtain three values which are averaged to obtain the final pressure value.

SYSTEMATIC VARIATION FOR DAM BREAK									
1.0m Wave Height					1.1m Wave Height				
Distance [m]	Pressure1	Pressure2	Pressure3	Average Pressure [Pa]	Distance [m]	Pressure1	Pressure2	Pressure3	Average Pressure [Pa]
1,5	28320	26740	28190	27750	1,5	27000	25790	24200	25663,33333
1,6	32770	31780	31020	31856,66667	1,6	34280	29340	32980	32200
1,7	33660	31640	29290	31530	1,7	34820	34690	32060	33856,66667
1,8	36080	33890	30310	33426,66667	1,8	33910	32910	39230	35350
1,9	35020	33940	30870	33276,66667	1,9	34930	32400	30860	32730
2	38890	37510	37110	37836,66667	2	43050	37510	33420	37993,33333
2,1	89730	84850	85170	86583,33333	2,1	47150	43490	39670	43436,66667
2,2	86940	82840	65830	78536,66667	2,2	73060	68790	45300	62383,33333
2,3	117900	116500	114700	116366,6667	2,3	98680	88970	84750	90800
2,4	53300	52500	53050	52950	2,4	146300	169400	165200	160300
2,5	90770	89380	87410	89186,66667	2,5	72330	47550	46790	55556,66667
2,6	90290	85960	88360	88203,33333	2,6	47000	46670	46180	46616,66667
2,7	58060	57520	58810	58130	2,7	51290	49720	48280	49763,33333
2,8	46110	45670	45320	45700	2,8	61400	60760	59950	60703,33333
2,9	43130	42770	42160	42686,66667	2,9	42760	41990	41230	41993,33333
3	30800	30840	30800	30813,33333	3	39880	39860	39770	39836,66667

Table A.1: Pressure calculations for dam break waves of 1 and 1.1m height

The graphs below represent the pressures obtained for the simulation for 1m input wave at all the locations while performing the systematic study, the ComFLOW shows reasonable pressure behaviour when the structure is at a distance of 1m from the wave, after changing the variable, uncertainties in obtaining the pressure is seen.

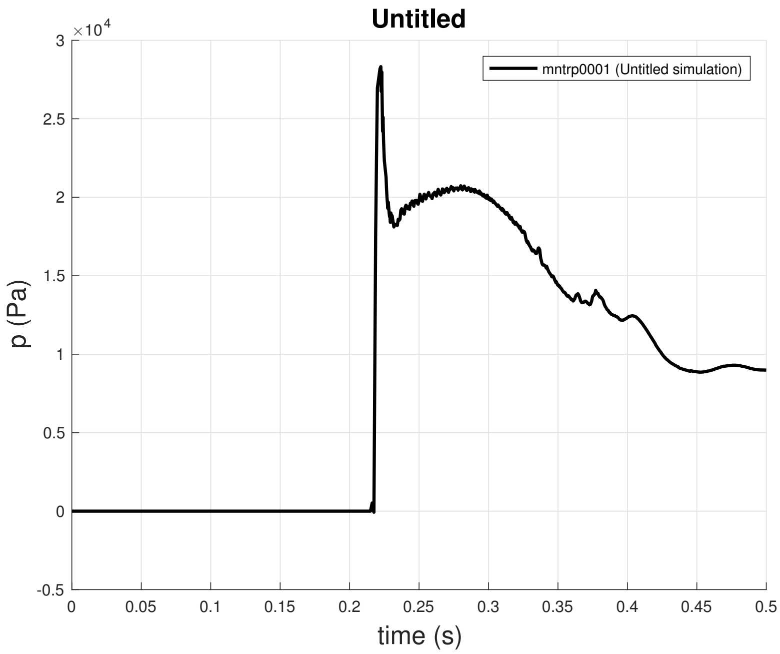


Figure A.3: Pressure at 1.5m distance for 1m wave in time

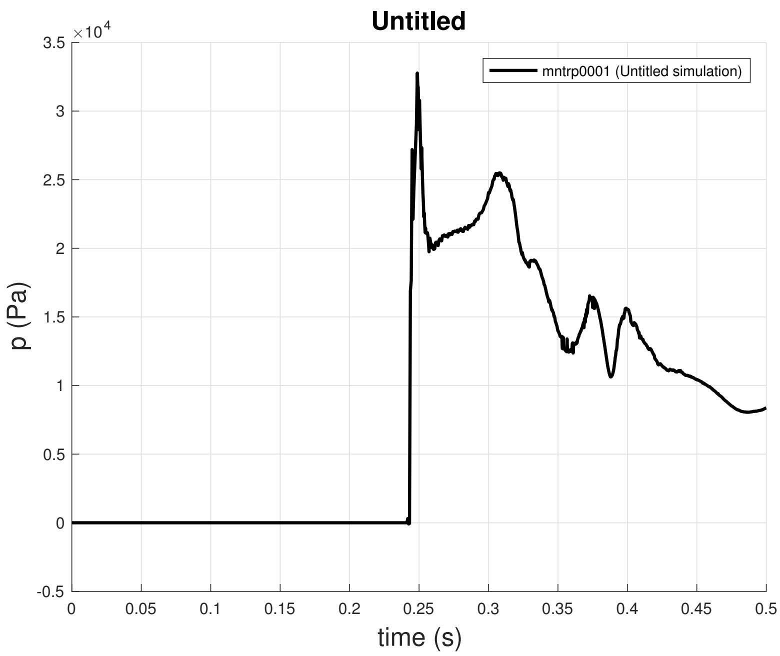


Figure A.4: Pressure at 1.6m distance for 1m wave in time

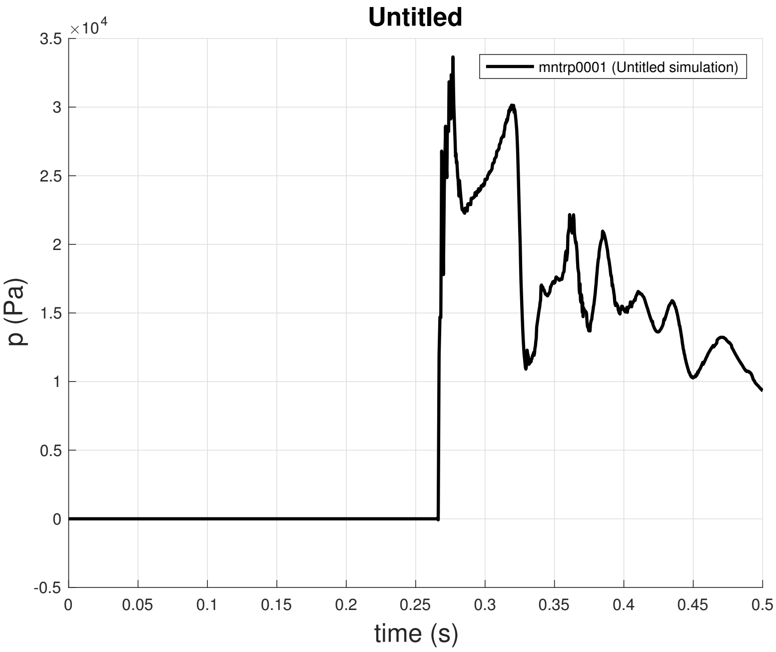


Figure A.5: Pressure at 1.7m distance for 1m wave in time

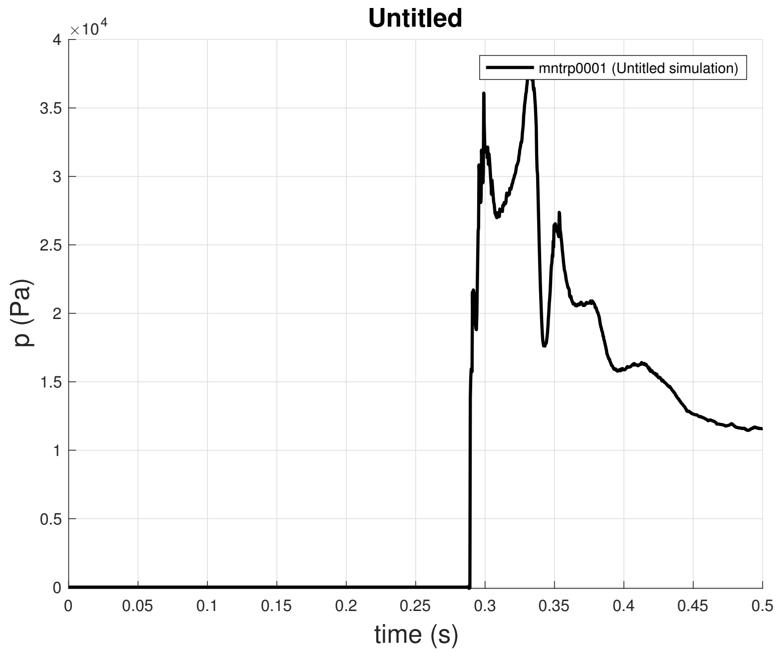


Figure A.6: Pressure at 1.8m distance for 1m wave in time

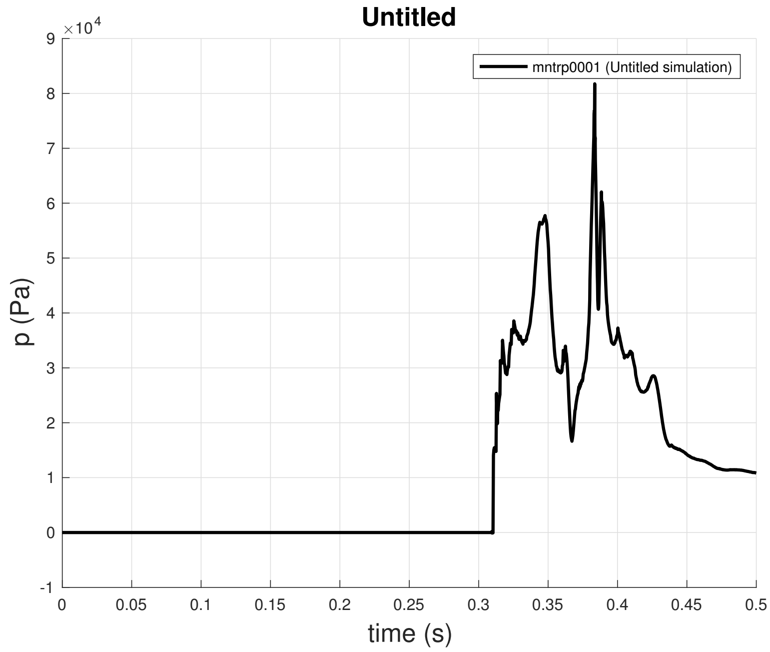


Figure A.7: Pressure at 1.9m distance for 1m wave in time

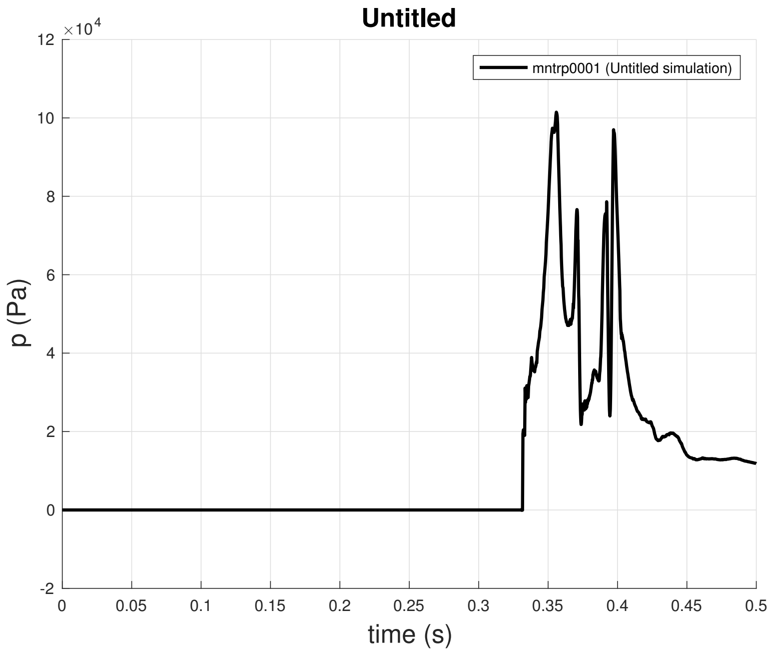


Figure A.8: Pressure at 2m distance for 1m wave in time

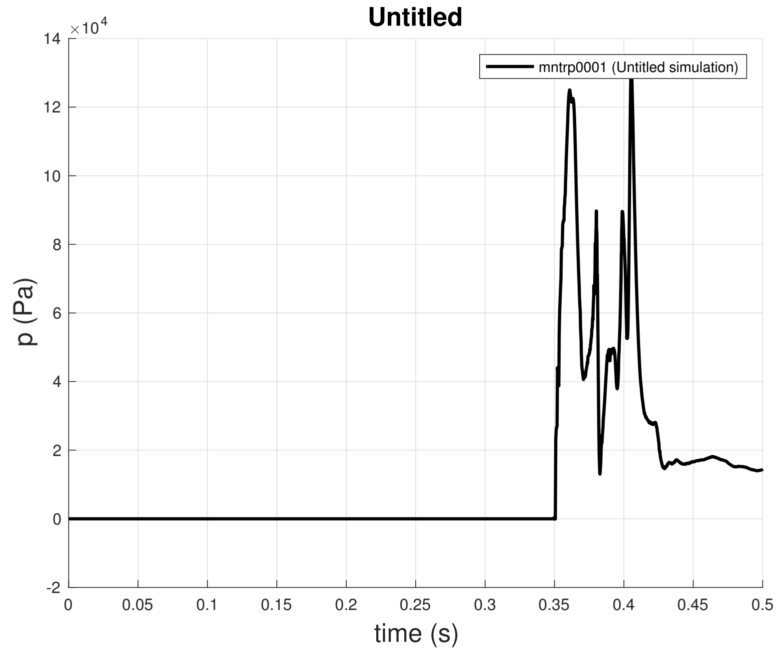


Figure A.9: Pressure at 2.1m distance for 1m wave in time

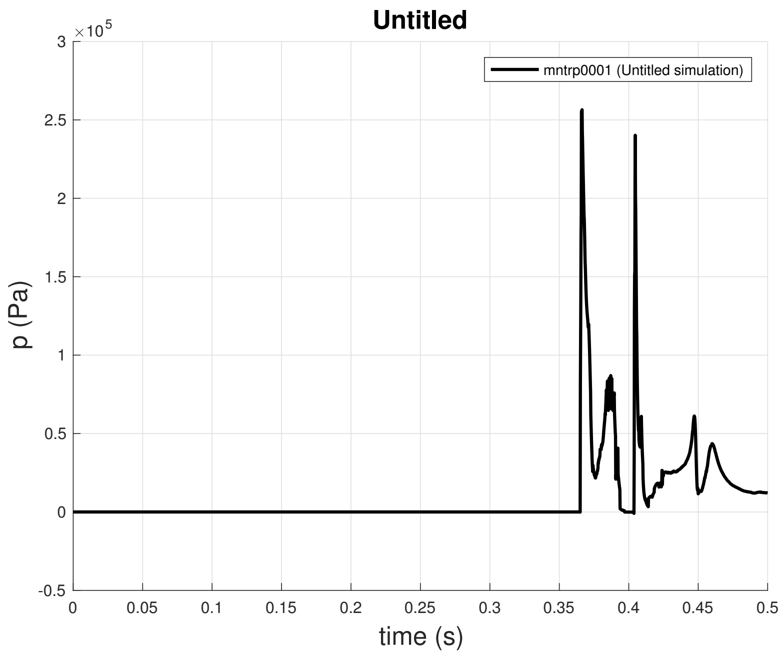


Figure A.10: Pressure at 2.2m distance for 1m wave in time

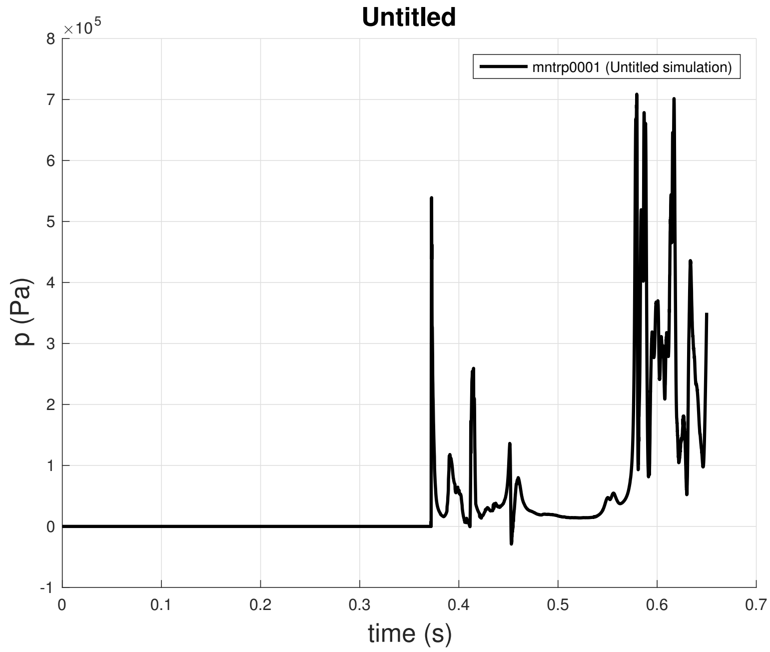


Figure A.11: Pressure at 2.3m distance for 1m wave in time

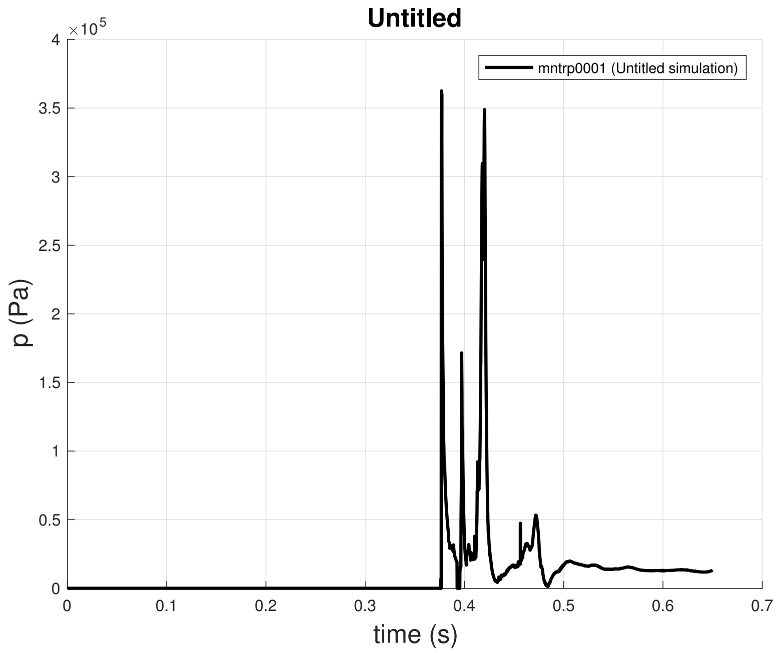


Figure A.12: Pressure at 2.4m distance for 1m wave in time

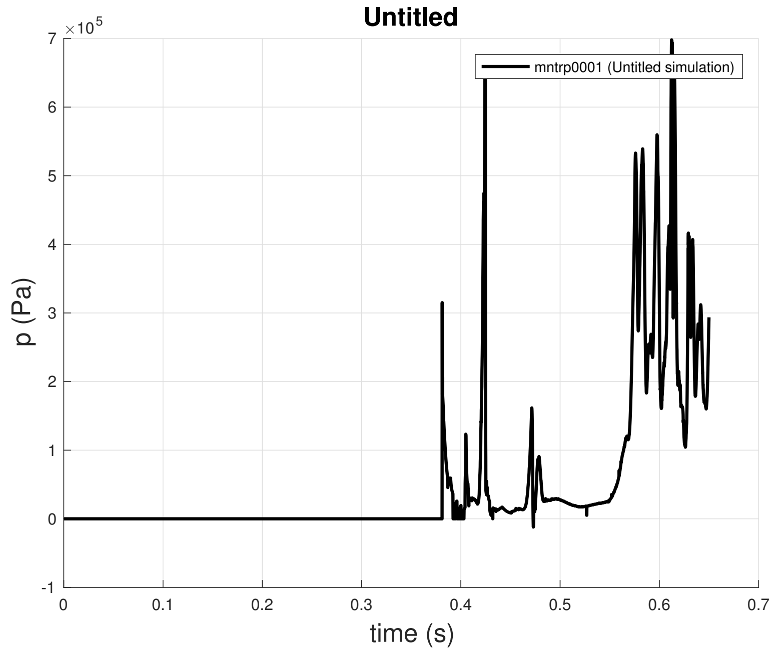


Figure A.13: Pressure at 2.5m distance for 1m wave in time

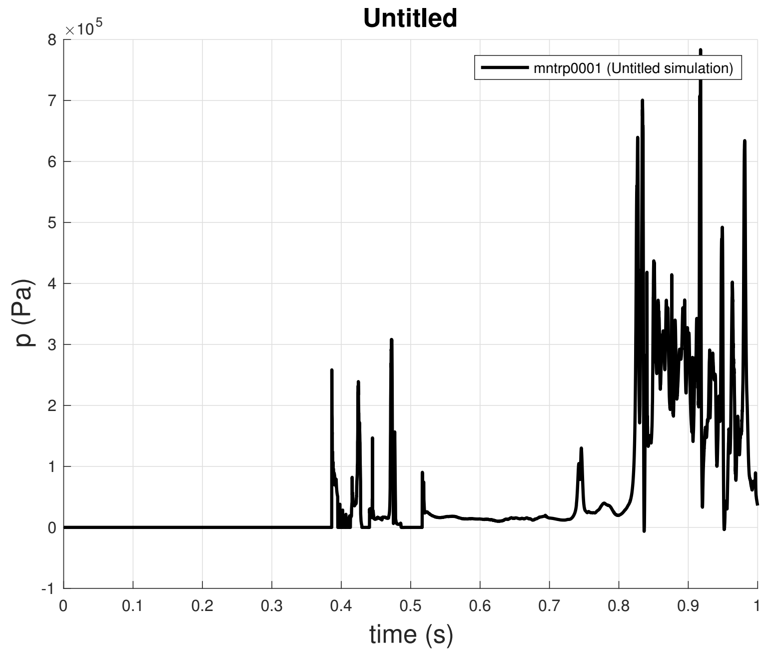


Figure A.14: Pressure at 2.6m distance for 1m wave in time

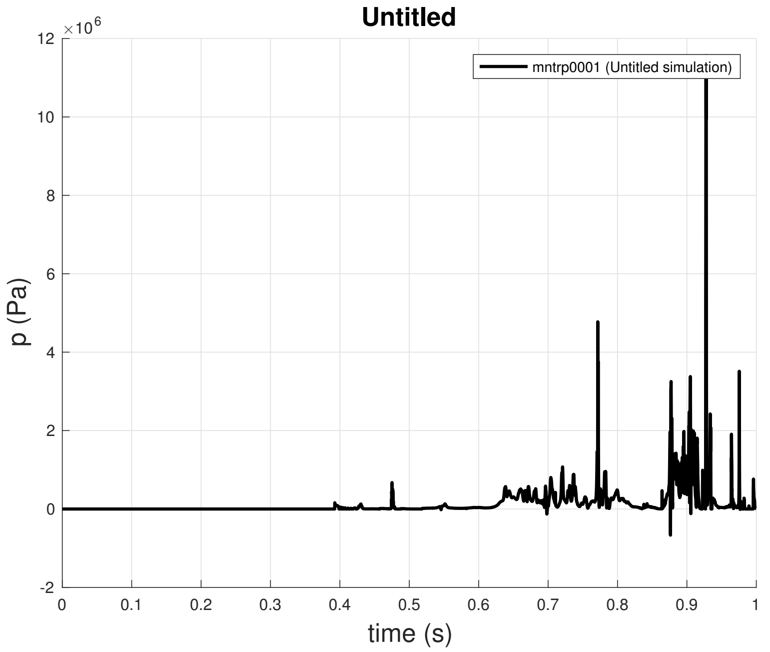


Figure A.15: Pressure at 2.7m distance for 1m wave in time

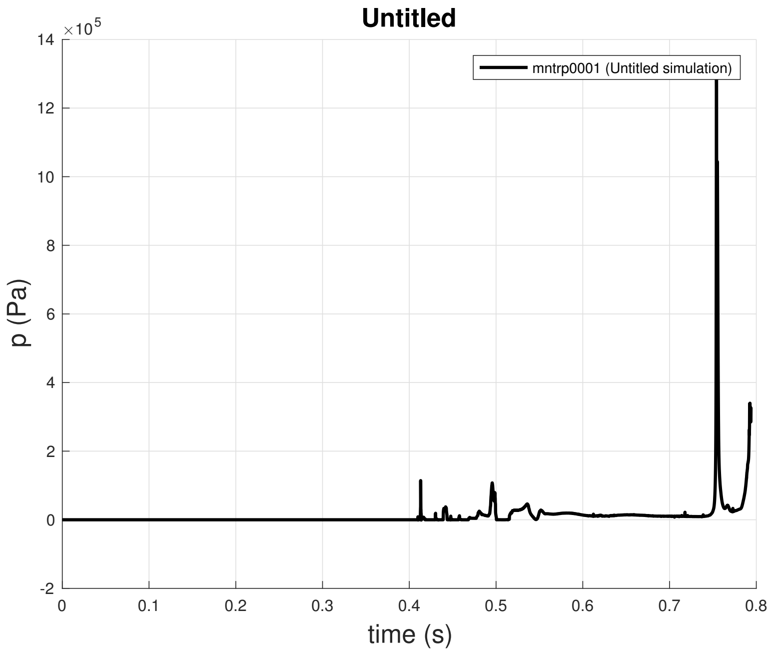


Figure A.16: Pressure at 2.8m distance for 1m wave in time

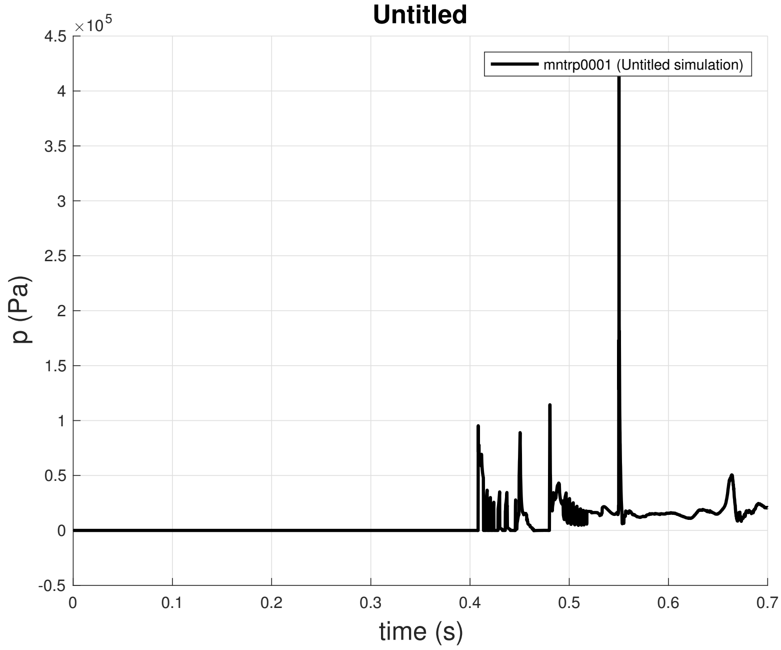


Figure A.17: Pressure at 2.9m distance for 1m wave in time

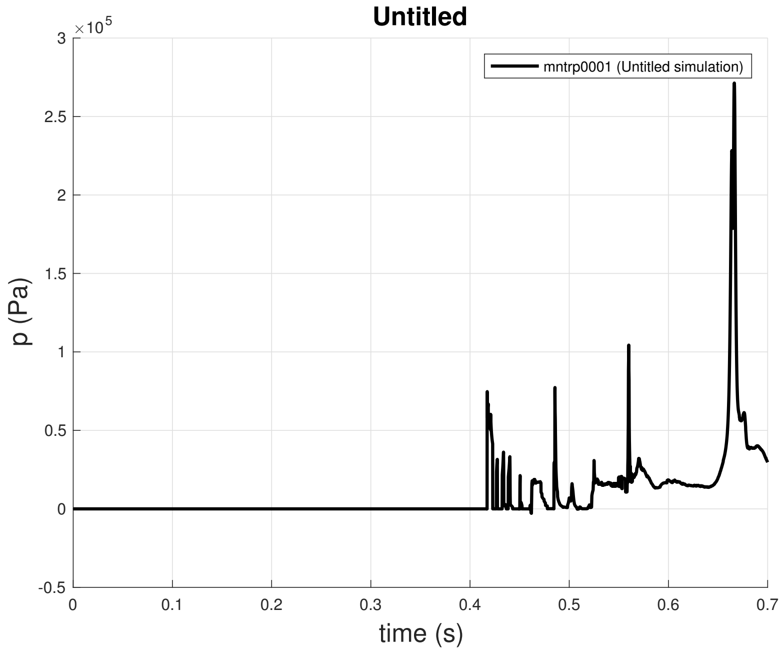


Figure A.18: Pressure at 3m distance for 1m wave in time

Similarly, The simulations for 1.1m wave height is performed for systematic variation calculation for dam break. These pressure graphs are presented below.

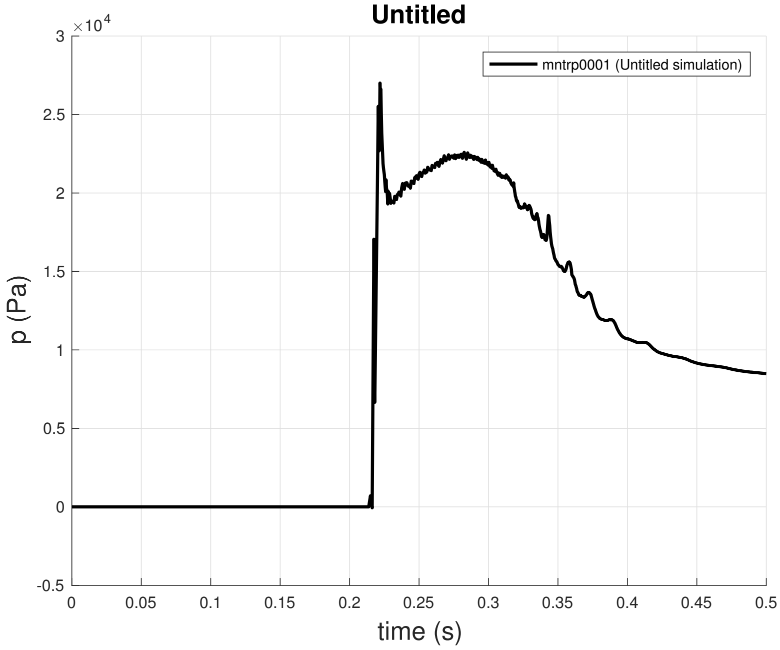


Figure A.19: Pressure at 1.5m distance for 1.1m wave in time

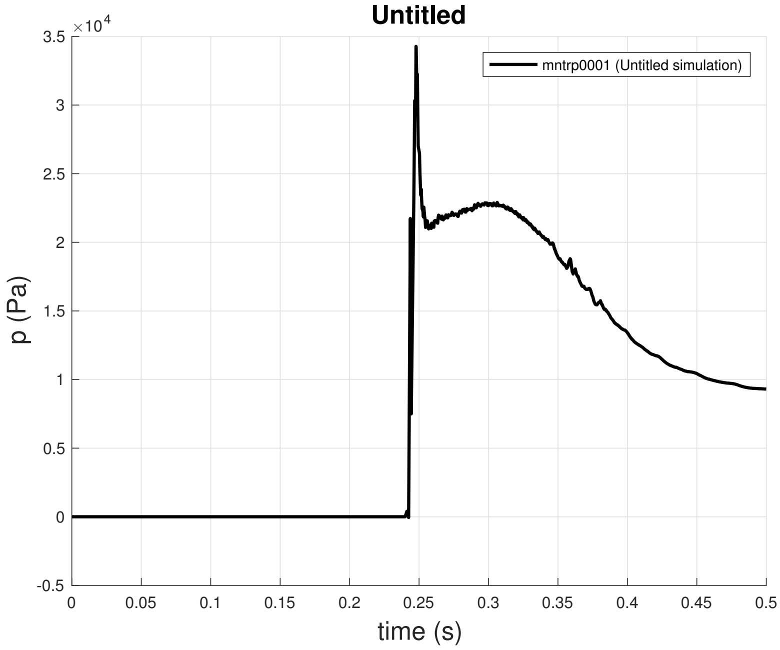


Figure A.20: Pressure at 1.6m distance for 1.1m wave in time

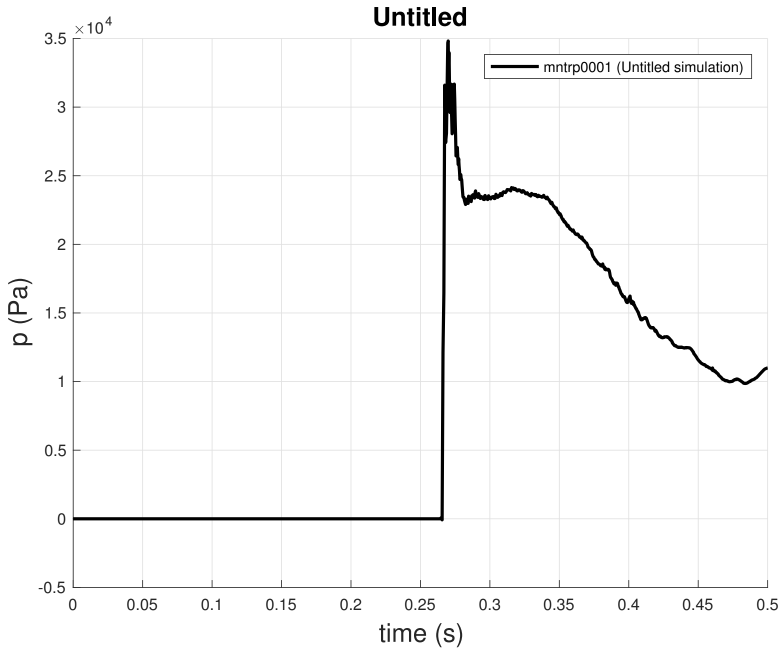


Figure A.21: Pressure at 1.7m distance for 1.1m wave in time

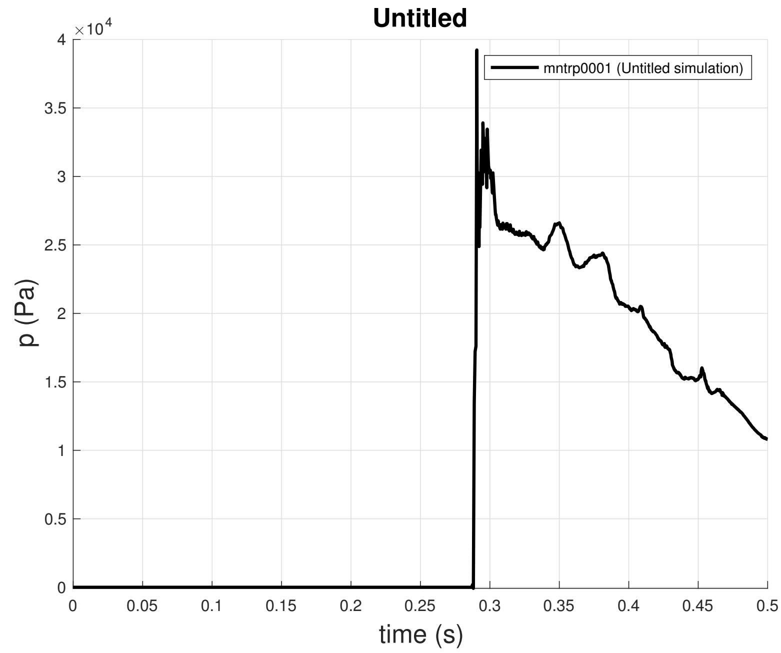


Figure A.22: Pressure at 1.8m distance for 1.1m wave in time

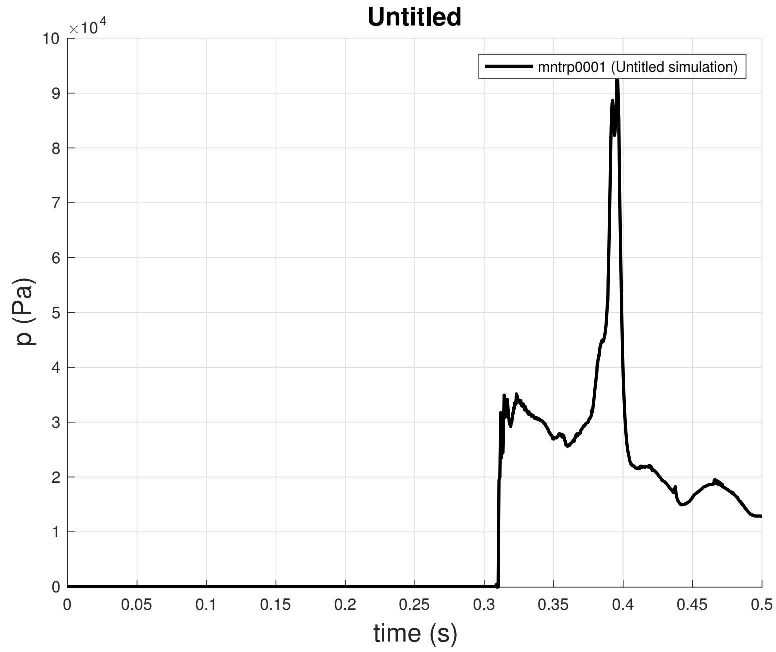


Figure A.23: Pressure at 1.9m distance for 1.1m wave in time

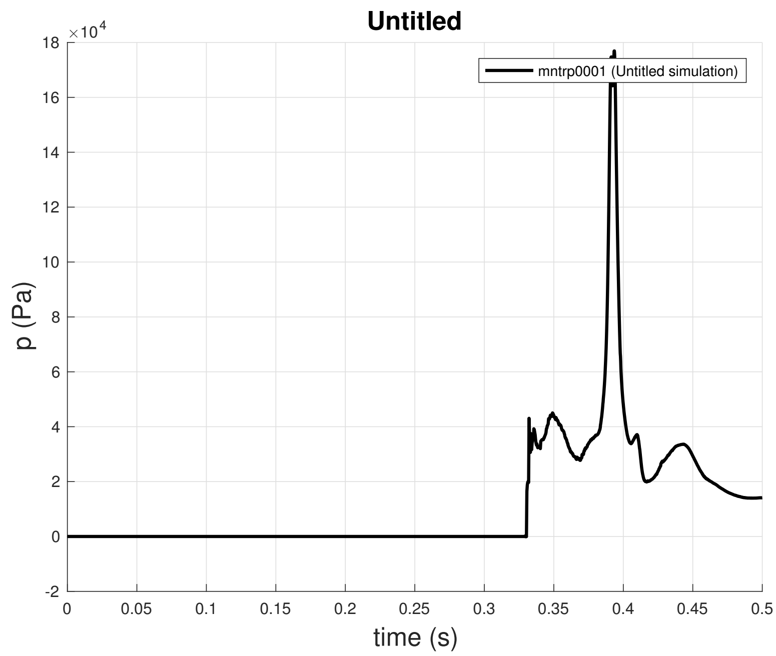


Figure A.24: Pressure at 2m distance for 1.1m wave in time

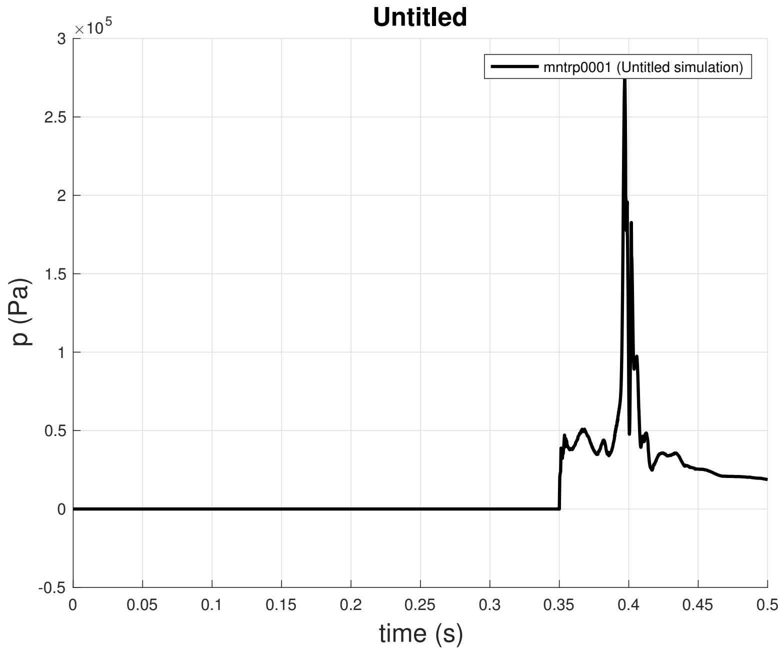


Figure A.25: Pressure at 2.1m distance for 1.1m wave in time

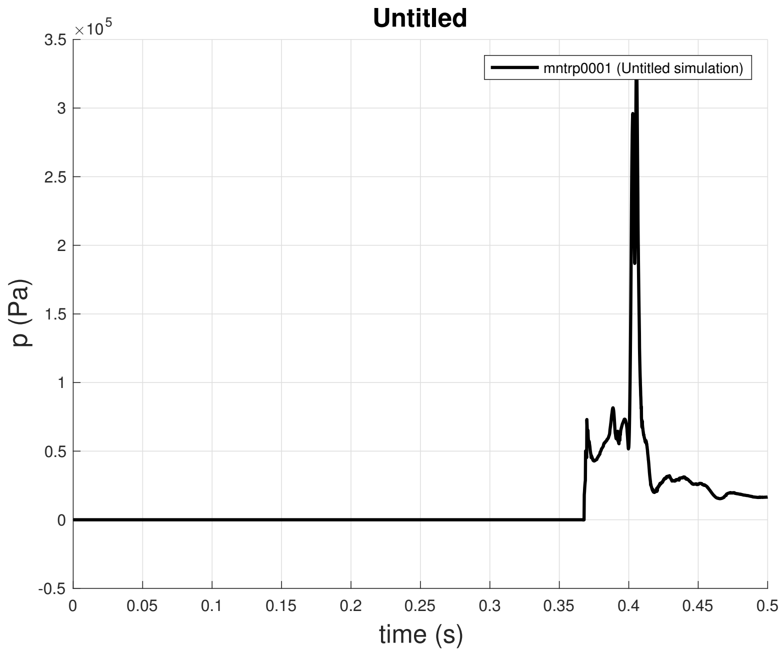


Figure A.26: Pressure at 2.2m distance for 1.1m wave in time

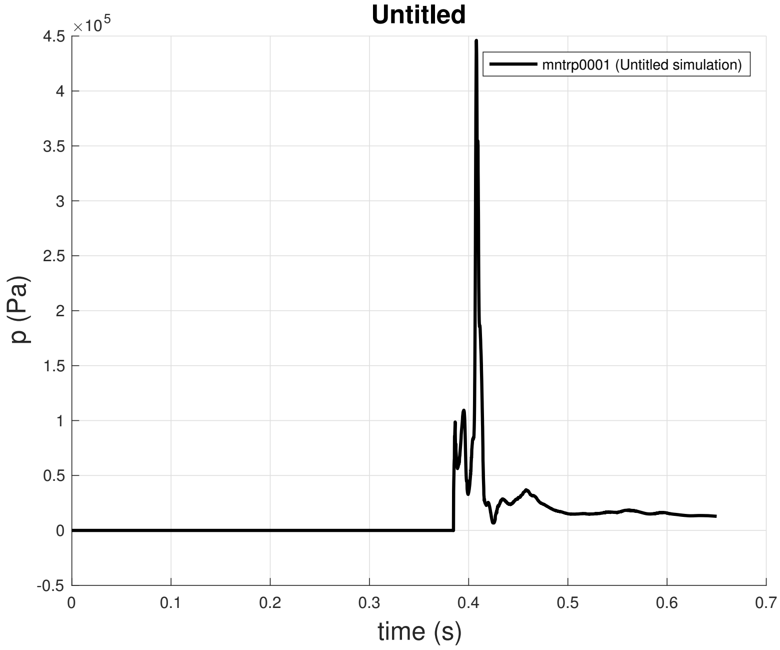


Figure A.27: Pressure at 2.3m distance for 1.1m wave in time

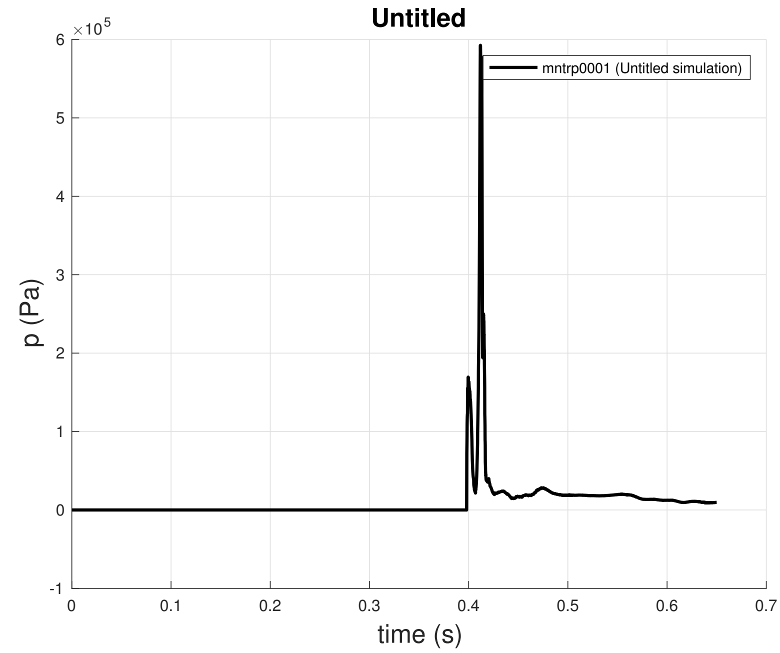


Figure A.28: Pressure at 2.4m distance for 1.1m wave in time

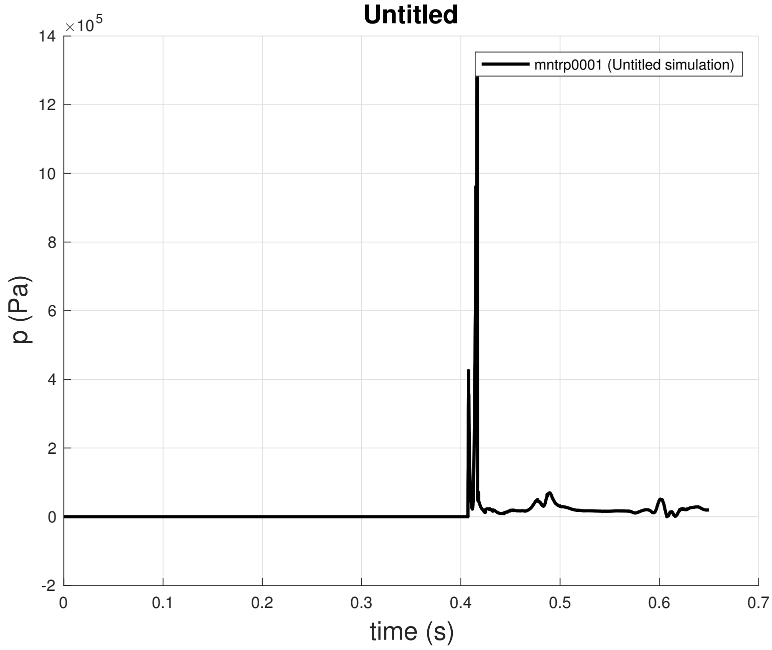


Figure A.29: Pressure at 2.5m distance for 1.1m wave in time

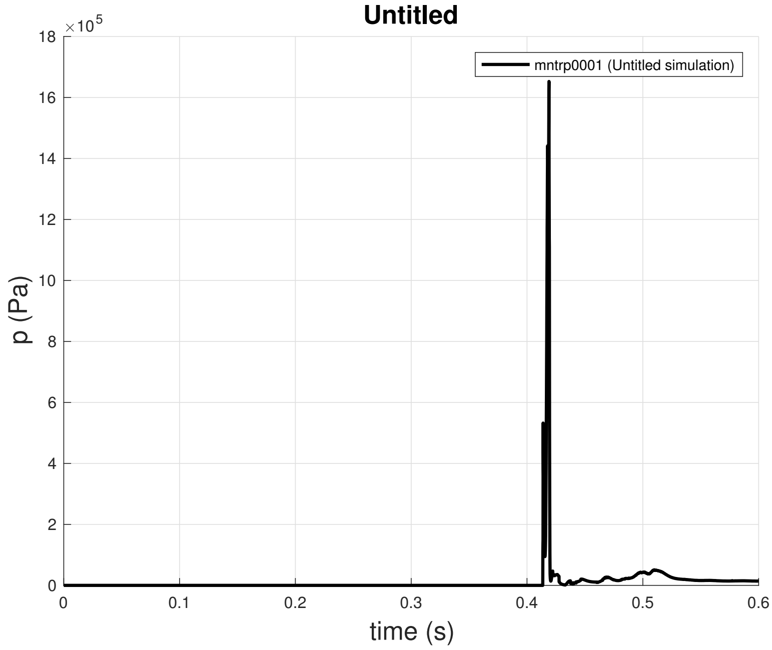


Figure A.30: Pressure at 2.6m distance for 1.1m wave in time

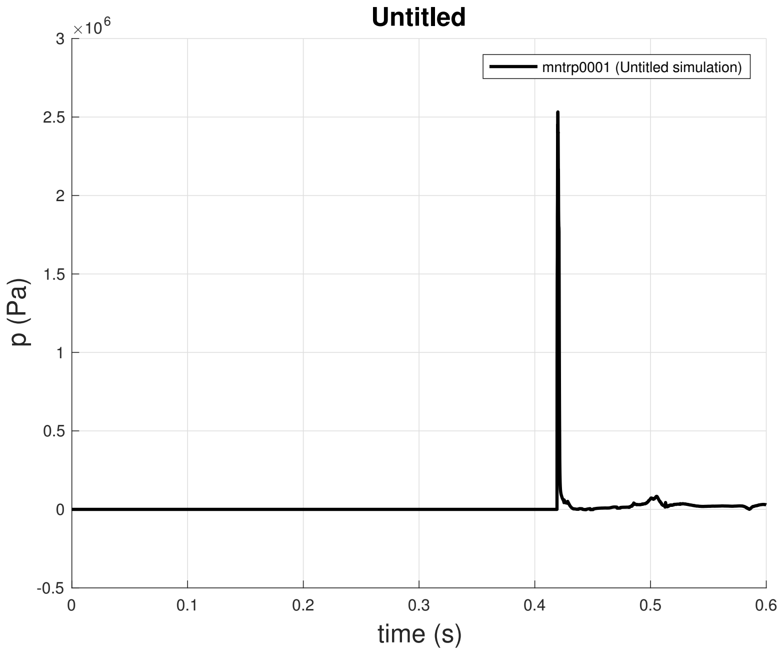


Figure A.31: Pressure at 2.7m distance for 1.1m wave in time

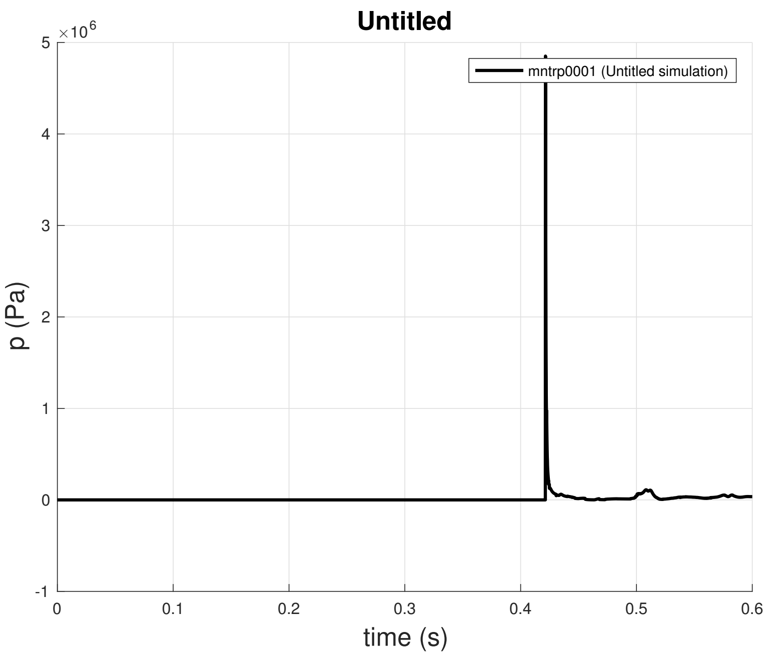


Figure A.32: Pressure at 2.8m distance for 1.1m wave in time

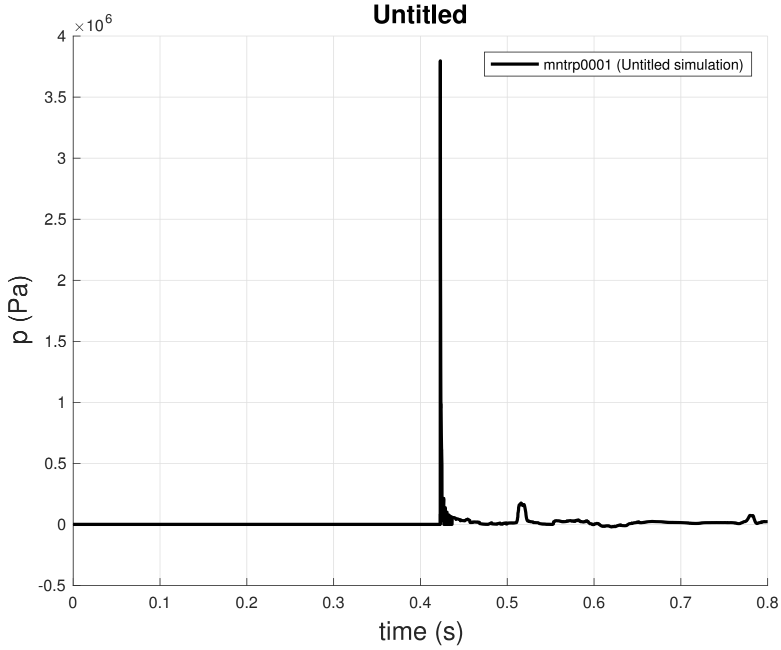


Figure A.33: Pressure at 2.9m distance for 1.1m wave in time

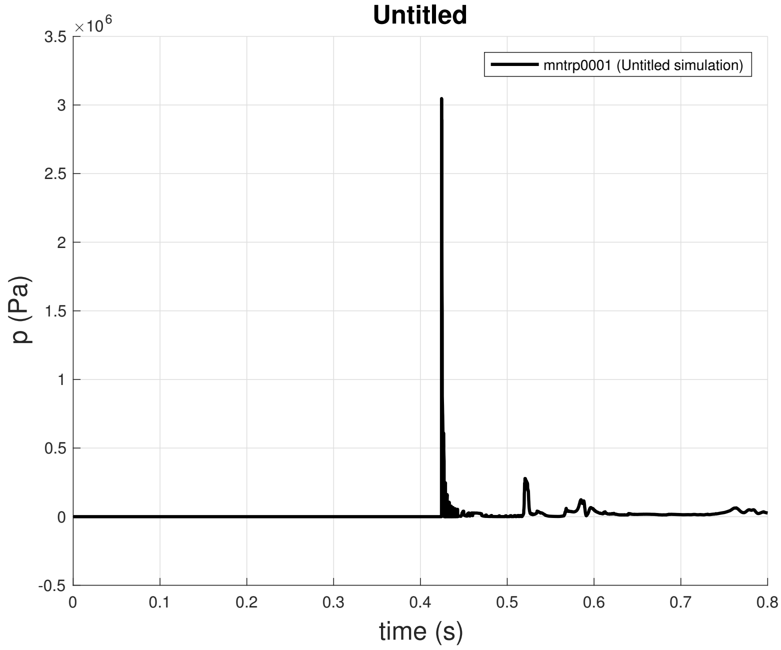


Figure A.34: Pressure at 3m distance for 1.1m wave in time

B

DAM BREAK WAVES ON IDEALISED DECK

B.1. CELL LABELING CASES ON IDEALIZED DECK WITH HIGHER CFL NUMBER

A basic concept of Cell labeling has been discussed in Introduction under section 2.4. It will be examined in detail here for dam break waves and effect of cell labels on fluid-structure interaction will also be observed. For simulations of fluid flow in ComFLOW, the whole domain is covered with the Cartesian grid with variables. The velocity components are defined on cell boundaries while pressure are designated in cell center. For our complex structure, cells of different character are used. This difference of character is induced by introducing edge and volume apertures. The fluid cells are labeled as empty cells (E), surface cells (S) and fluid cells (F). Based on Geometry, the cells are labeled as a boundary (B) or an exterior (X) cell. Navier-Stokes equations are used to discretise in space and time. For time discretisation, the Forward Euler method or Adams-Bashforth method can be used while central discretisation and first or second order upwind discretisation can opt for spatial discrerisation. The cells are 0.01m long with the same width. F cells represent the fluid cells in an incoming wave, S represents the surface cells while E represents the Empty cells and B serve as the body cells. A thin blue line represents the wave shape of the incoming wave while the structure on deck is represented by body cells on the right and an incoming wave is represented by surface and fluid cells in this Dam break ComFLOW domain.
The location of Surface and fluid cells before fluid impact on the structure is important. It is detected that the position of Fluid cell plays a vital role in pressure generated by this fluid onto the structure. The F cell should be present below surface S-cells as in figure B.1. This is the optimized shape for wave impact calculations in ComFLOW. The results from this shape can be trusted and are accurate.

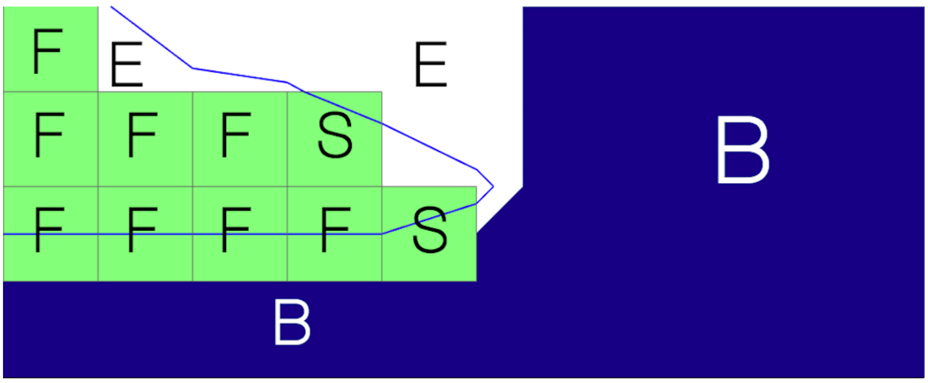
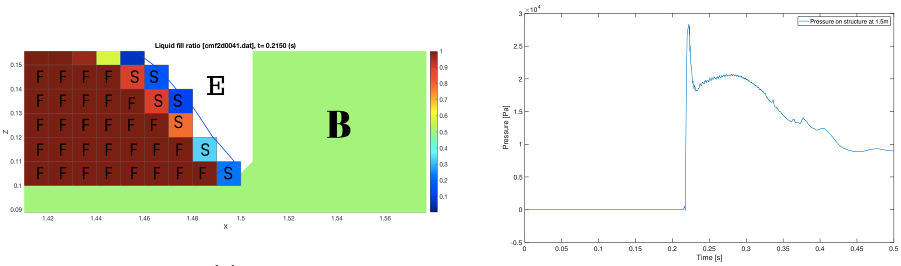


Figure B.1: The ideal cells scenario

In figure B.1, green cells on the left indicates the fluid cells and surface cells of incoming dam break wave. B embodies the boundary cells of structure on deck. E represents the air or empty cells. This configuration of wave cells in the ComFLOW domain is ideal and provides accurate results for pressure calculation generated. This configuration of a wave is only valid when the structure is close to the wave for $500 \cdot 1 \cdot 200$ grid resolution. When the distance of this structure is varied after 2.1m in the domain, wave shape changes and cause irregularities in Comflow which will be discussed in detail further.

Four scenarios are contested for the cell labels and their pressure generated on the structure monitored by the monitor point.

1. Case 1: The structure is placed at 1.5m in x-direction inside the ComFLOW domain i.e. 0.5m from the incoming wave. The water column is 1m high and wide. The whole domain is designed as a vessel deck. The dam breaking simulation is performed on the deck for a realistic interaction of deck structure and fluid.

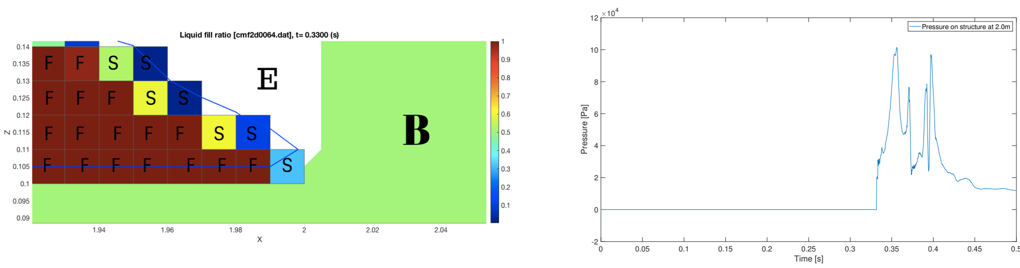


(a) (b)

Figure B.2: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 1.5m over a duration of 0.5s on idealised deck

For Dam breaking simulation for a structure at 1.5m, ideal SSF cells are observed which results in excellent pressure figure observed by Kleefsman [1] in her numerical simulations. Pressure peak, stationary and stability phase can be observed over the duration of 0.5s. This configuration of wave cells in B.2a results in ideal accurate pressure for observations.

2. Case 2: The structure is placed at 2.0m in x-direction inside ComFLOW domain i.e. 1.0m from the incoming wave. The water column height is same compared to the previous case. The simulation is performed on a realistic interaction of deck structure and fluid.

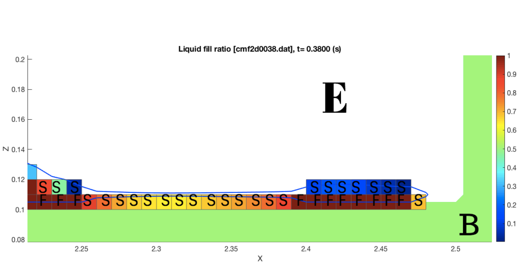


(a) (b)

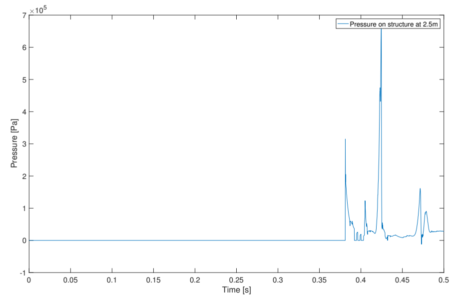
Figure B.3: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 2.0m over a duration of 0.5s on idealised deck

For dam break simulation on an idealised deck with a structure at 2.0m, ideal SSF cell pattern is observed which results in pressure figure B.3b. Pressure peak is observed due to initial impact at 0.3350s. The initial peak is considered while later peaks have to be ignored which is due to higher volume of fluid slamming on the structure. This configuration of wave cells results in pressure which can be calculated, however, cannot be trusted.

3. Case 3: The structure is placed at 2.5m in x-direction inside ComFLOW domain i.e. 1.5m from incoming dam break wave. The water column remains same as previous cases. The simulation is performed on idealised deck realistic interaction of deck structure and fluid.



(a)

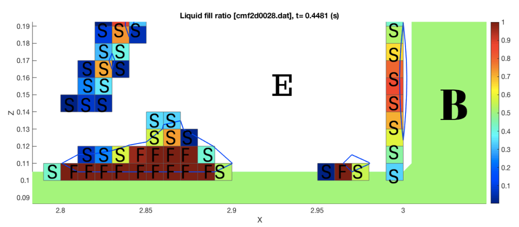


(b)

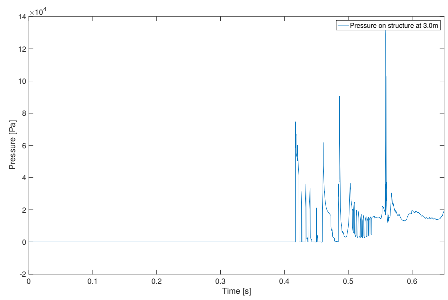
Figure B.4: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 2.5m over a duration of 0.5s on idealised deck

For dam break simulation on an idealised deck with a structure at 2.5m, irregularities are seen in ComFLOW. These irregularities are the peculiar pressure peak at 0.39s in simulation, which arises due to the formation of water stream consisting of just surface cells prior to actual fluid flow. The velocity of this stream is observed to be higher than the actual fluid, which results in peculiar high peaks. The cell pattern observed in figure B.4a represents both droplet and the fluid flow before an impact on the structure and is not the ideal SSF pattern observed in previous cases.

4. Case 4: The structure is placed at 3.0m in x-direction i.e. 2.0m from the incoming wave. The water column remains same as for previous three cases. The simulation is performed on idealised deck for the realistic interaction of deck structure and fluid.



(a)



(b)

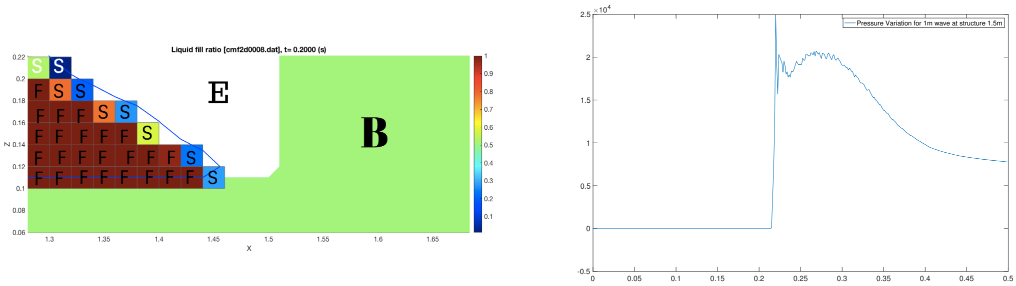
Figure B.5: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 3.0m over a duration of 0.8s on idealised deck

For simulation, in this case, water droplets are observed all over again. These droplets have an impact on the structure prior to original impact. However later in the simulation, water droplets are formed all over the domain which is fired towards the monitor point which again causes peculiar peak observed in figure B.5b. The cell label pattern in B.5a consists of mostly surface cells of droplets which makes it difficult in calculating the impact pressure.

B.2. CELL LABELING CASES WITH REDUCED GRID RESOLUTION

The same cases discussed previously will be simulated again in reduced gird resolution, the grid size for these cases remain as $250 \cdot 1 \cdot 100$ which is a fraction of previously used cases. It is simulated to observe if irregularities still exist when grid size is reduced and cells are sizable compared to previous cases.

1. Case 1: The structure is placed at 1.5m in x-direction inside ComFLOW domain. i.e. 0.5m from incoming wave column which is 1m high and wide. The whole domain is designed as vessel deck.



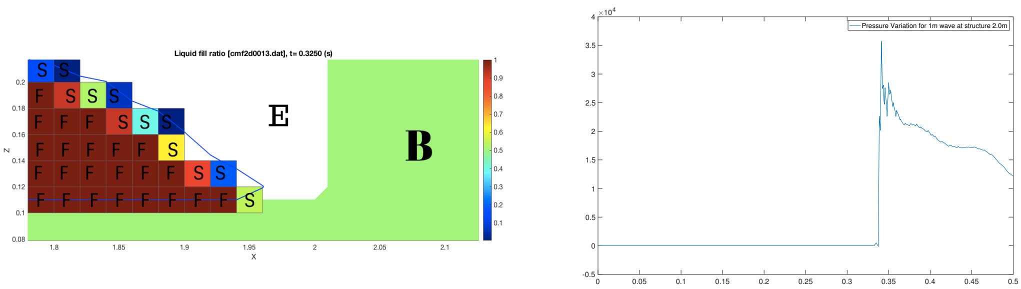
(a)

(b)

Figure B.6: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 1.5m over a duration of 0.5s on idealised deck with lower gird resolution

For Dam breaking simulation for a structure at 1.5m, ideal SSF cells are observed which results in excellent pressure figure observed by Kleefsman [1] in her numerical simulations. Pressure peak, stationary and stability phase can be observed over the duration of 0.5s. This configuration of wave cells in B.6a results in ideal accurate pressure for observations. They are similar to the previous case of same parameters observed in **??**

2. Case 2: The structure is placed at 2.0m in x-direction inside ComFLOW domain. i.e. 1.0m from incoming wave column which is 1m high and wide. The whole domain is designed as vessel deck for ideal interaction with incoming water.



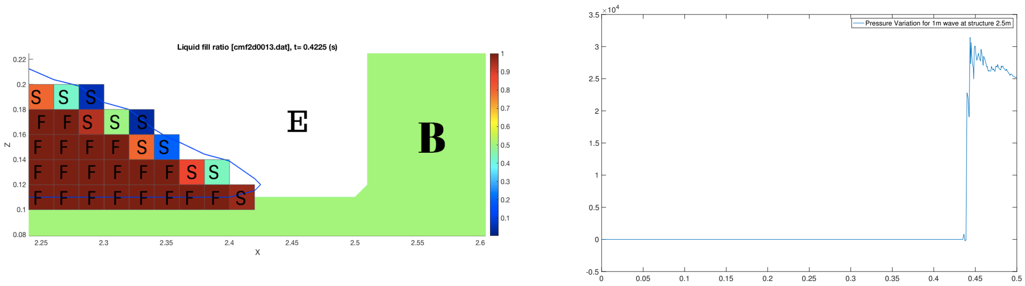
(a)

(b)

Figure B.7: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 2.0m over a duration of 0.5s on idealised deck with lower gird resolution

For dam break simulation on idealised deck with structure at 2.0m, ideal SSF cell pattern is observed which results in pressure figure B.7b. Pressure peak is observed due to initial impact at 0.3350s. This pressure behavior is totally different than observed in previous case in figure B.3. Higher peaks later in the simulation are observed in previous case while they are absent in this simulation. Irregularities present with refined grid size are non-existent in this case.

3. Case 3: The structure is placed at 2.5m in x-direction inside ComFLOW domain. i.e. 1.5m from incoming wave column which is 1m high and wide. The whole domain is designed as vessel deck for ideal interaction with incoming water.



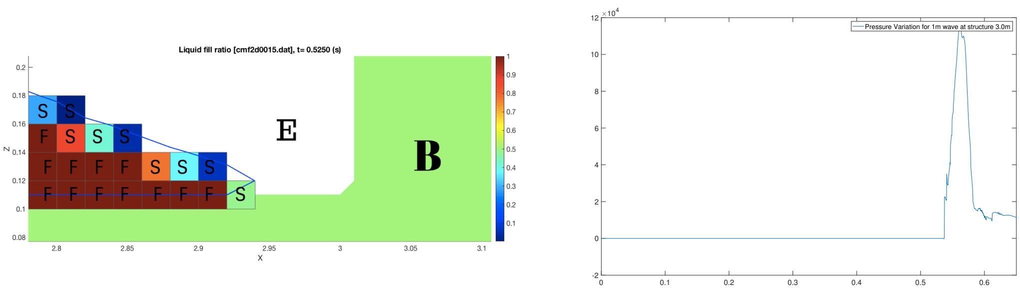
(a)

(b)

Figure B.8: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 2.5m over a duration of 0.5s on idealised deck with lower gird resolution

For dam break simulation on idealised deck with a structure at 2.5m, ideal SSF cell pattern is observed which results in pressure figure B.8b. Pressure peak is observed due to initial impact at 0.4550s. This pressure behavior is totally different than observed in the previous case in figure B.4b. Higher peaks later in the simulation are observed in the previous case while they are absent in this simulation. Water droplets are totally missing which caused irregularities in the previous case. However, high peak just after initial impact is observed which makes this result unreliable.

4. Case 4: The structure is placed at 3.0m in x-direction inside ComFLOW domain. i.e. 2.0m from incoming wave column which is 1m high and wide. The whole domain is designed as vessel deck for ideal interaction with incoming water.



(a)

(b)

Figure B.9: Dam Break Comflow simulation indicating cell labels and Pressure exerted on the structure at 3.0m over a duration of 0.65s on idealised deck with lower gird resolution

For dam break simulation on a idealised deck with structure at 3.0m, more Surface cells are observed which results in the pressure figure B.9b. Less Fluid cells are present in the incoming wave. Pressure peak is observed due to initial impact at 0.550s. This pressure behavior is totally different than observed in the previous case in figure B.5b. Huge peak just after impact is observed in this case which makes this result unreliable.

B.2.1. COMPARISON OF PRESSURE BEHAVIOR OVER TWO DIFFERENT CELL RANGES

In this section, pressure behavior of two different scenarios is discussed. The scenario one is with a grid resolution of $500 \cdot 1 \cdot 200$ which results in 0.01m cell size while the second scenario is with a grid resolution of $250 \cdot 1 \cdot 100$ which results in 0.02m cell size. The second scenario is reduced to half size of cells compared to first one. All other input parameters remain the same.

1. Case 1: The structure is placed at 1.5m in x-direction inside the ComFLOW domain i.e. 0.5m from the incoming wave. The water column is 1m high and wide. Pressure behavior over two simulations is monitored.

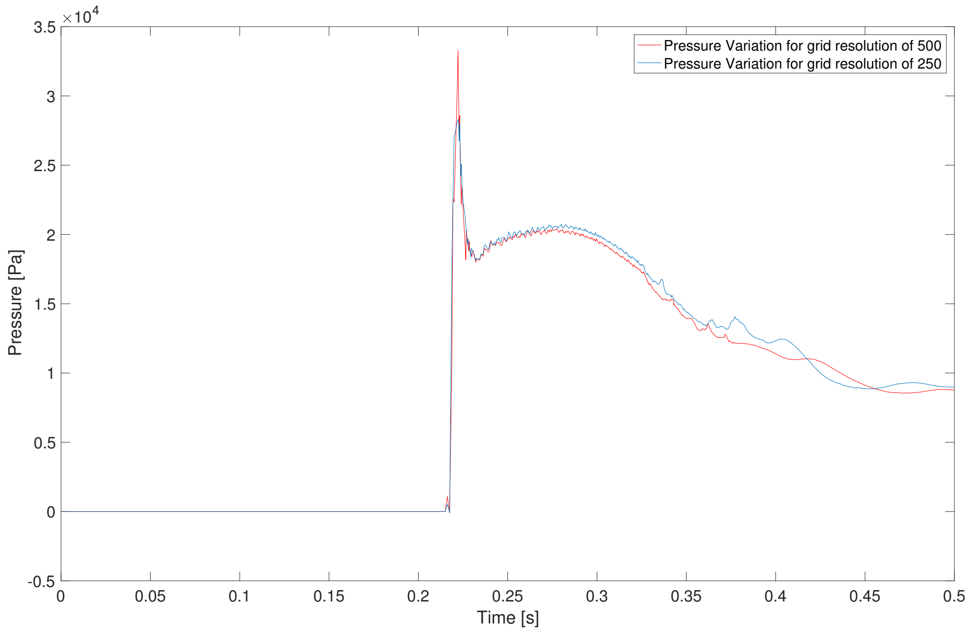


Figure B.10: Pressure behavior over two different grid sizes on structure kept at 1.5m

The course of two pressure for different grid size is almost the same. Higher initial impact peak is observed for grid resolution of 500 while behavior later in the simulation is almost equivalent to each other. Grid resolution of 250 offers more stability later in time.

2. Case 2: The structure is placed at 2.0m in x-direction inside the ComFLOW domain i.e. 1.0m from the incoming wave. The water column is 1m high and wide. Pressure behavior over two simulations is monitored.

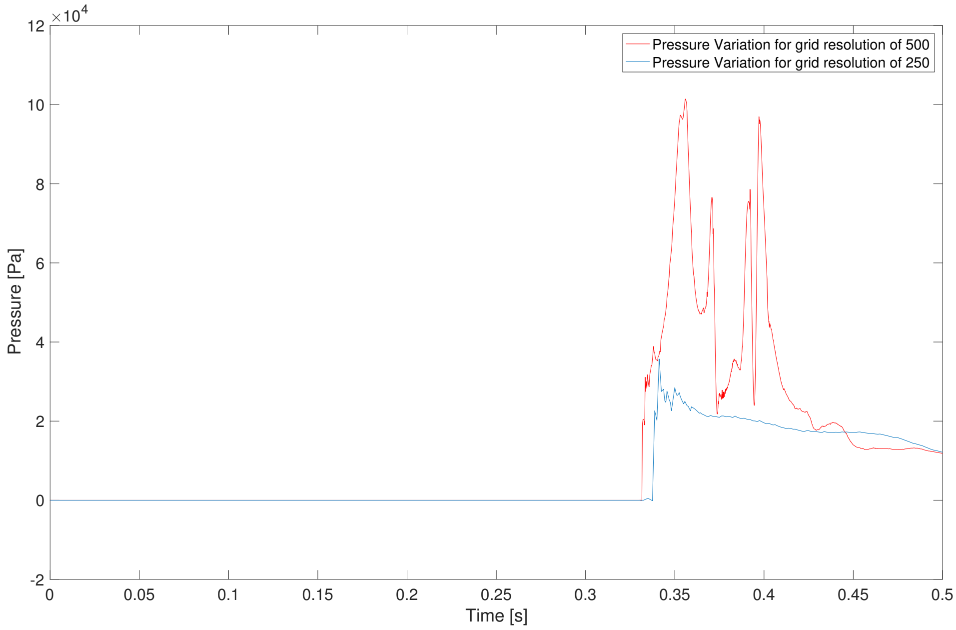


Figure B.11: Pressure behavior over two different grid sizes on structure kept at 2.0m

The course of two pressure for different grid size is totally divergent. The initial impact peak for both the cases is almost the same however later in simulation, high-pressure peaks for grid resolution 500 is observed which is irregular compared to a blue pressure line (grid resolution 250). The blue pressure line offers stable and accurate behavior compared to the red pressure line.

3. Case 3:

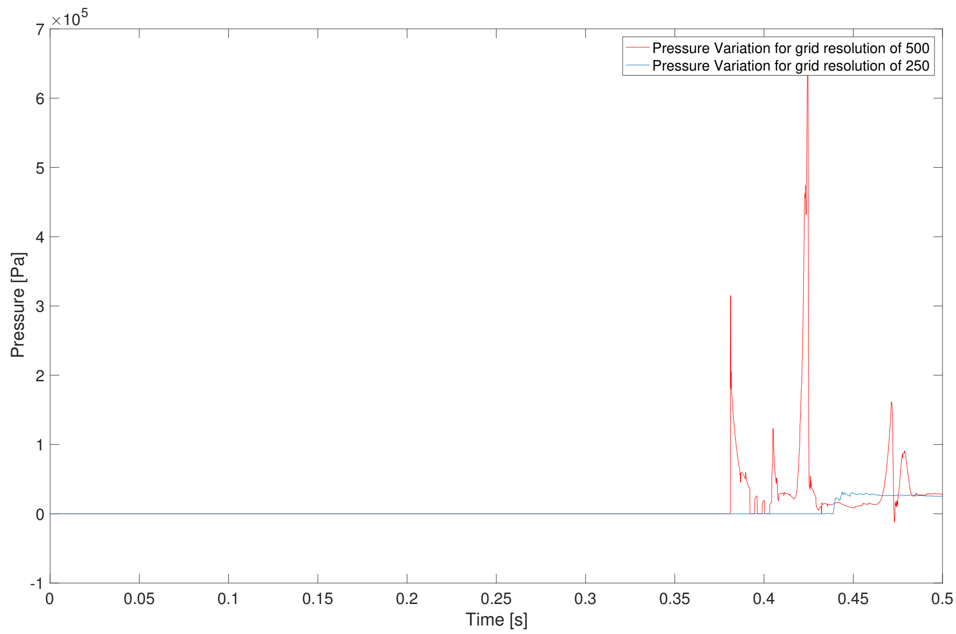


Figure B.12: Pressure behavior over two different grid sizes on structure kept at 2.5m

The structure is placed at 2.5m in x-direction inside the ComFLOW domain i.e. 1.5m from the incoming wave. The water column is 1m high and wide. Pressure behavior over two simulations is monitored. The course of two pressure for different grid size is totally divergent. The initial impact of water on a structure is different, unusual high peaks are observed for grid resolution of 500. However, the pressure variation for grid 250 is also irregular, the peak rises but refuses to decline and continues to stabilize at high pressure. This observation in both the results is un-trustworthy. The red line is highly unstable compared to blue line because of the formation of water droplets.

4. Case 4: The structure is placed at 3.0m in x-direction inside the ComFLOW domain i.e. 2.0m from the incoming wave. The water column is 1m high and wide. Pressure behavior over two simulations is monitored.

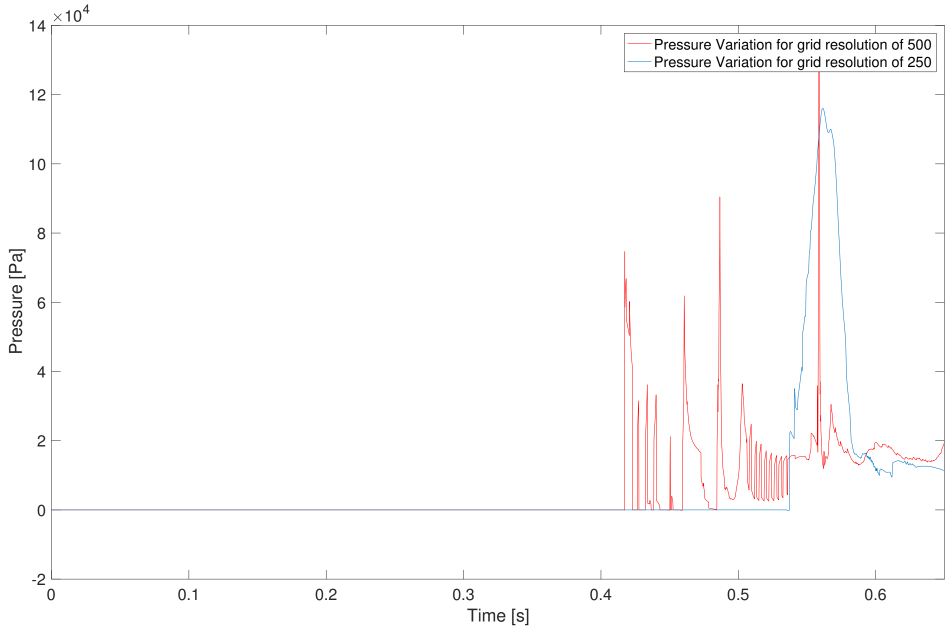


Figure B.13: Pressure behavior over two different grid sizes on structure kept at 3.0m

The course of two pressure for different grid size is totally divergent. The initial impact of water on the structure is different, unusual high peaks are observed for grid resolution of 500, initial impact peak for this cannot be assumed since there are very high-pressure peaks all over. However, the pressure variation for grid 250 is also irregular, the peak rises after initial impact but declines rapidly and continues to stabilize after 0.6s. This observation in both the results is untrustworthy. The red line is highly unstable compared to blue line because of formation of water droplets observed in cell labels.

B.3. SYSTEMATIC VARIATION FOR LOWER GRID RESOLUTION ON IDEALIZED DECK

The grid resolution of $250 \cdot 1 \cdot 100$ is seen to be more stable and accurate compared to its counterpart. But it cannot be trusted yet, So systematic variation for this grid resolution will be performed to verify the accuracy and will be used to find the relation between pressure and velocity of the fluid. The distance of the structure on deck is varied from 1.5m to 3.0m in the x-direction for a ComFLOW domain while wave height of 1m is used. The pressure and velocity

variation over distance are monitored. In figure B.14, the primary y-axis represents pressure variation over distance ranged from 1.5m to 3.0m while secondary y-axis indicates velocity variation for the same distance. It is observed that the velocity continues to rise as the distance is increased however it is not proportional to rise in pressure which is seen to fluctuate in and around 30000Pa. There are incline and decline in pressure with a variation of distance, however, the velocity keeps rising with an increase in distance.
There were no irregularities in water behavior to be seen as the distance from the wave increased which was a serious issue for grid size $500 \cdot 1 \cdot 200$ which resulted in irregular peaks and water droplet formation. However, no relation can yet be obtained from these pressure and velocity calculations.

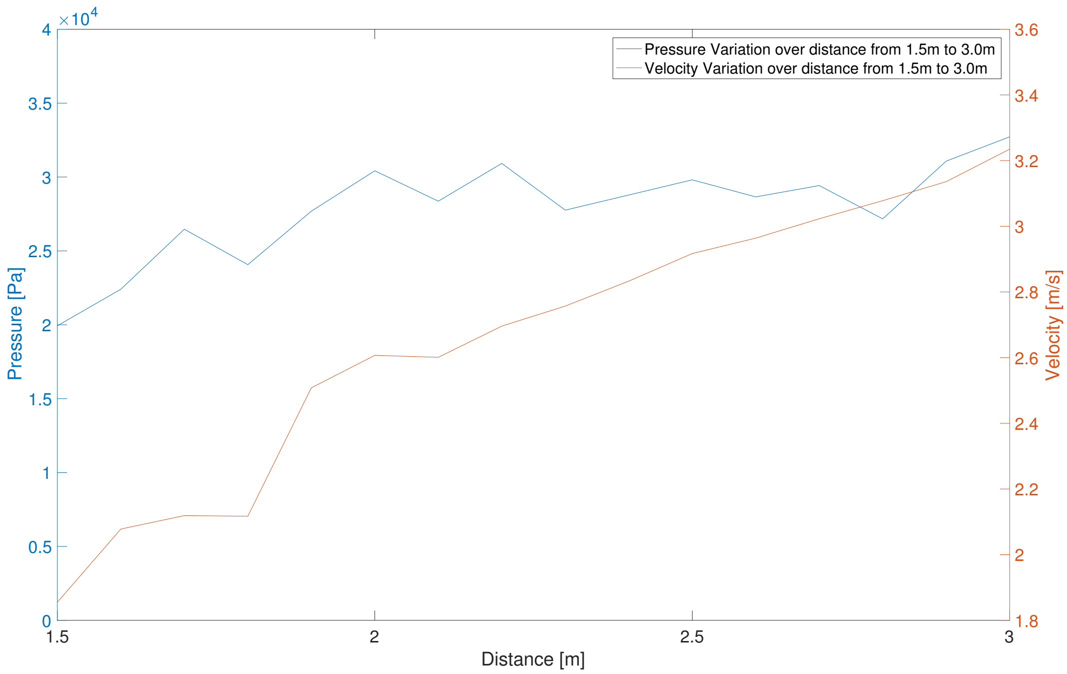


Figure B.14: Pressure and velocity variation over a distance from 1.5m to 3.0m

B.4. VARIATION OF DISTANCE ON IDEALISED DECK FOR $CFL_{max} = 0.8$

The distance of structure on deck is varied similarly to cases with dam break simulations on boundary conditions. Here the pressure graphs are presented for each case from 1.5m to 3.0m with $250 \cdot 1 \cdot 100$ grid resolution. This table B.1 indicates the impact pressure peak obtained from every simulation performed in ComFLOW. The pressure values are obtained from the monitor point graph in ComFLOW. It is refined in time with comparison to snapshots to obtain three values which are averaged to obtain the final pressure value.

Systematic Variation for Idealized dam break wave					
1m wave height					
Distance	Pressure 1	Pressure 2	Pressure 3	Average	Velocity
1,5	24940	19130	15730	19933,3333	1,855
1,6	27270	22140	17810	22406,6667	2,078
1,7	32060	25030	22320	26470	2,119
1,8	24710	19640	27860	24070	2,117
1,9	34340	25540	23180	27686,6667	2,508
2	35720	27490	28060	30423,3333	2,607
2,1	32550	24150	28400	28366,6667	2,601
2,2	37290	26600	28880	30923,3333	2,696
2,3	34630	25290	23360	27760	2,757
2,4	28930	24320	33100	28783,3333	2,833
2,5	31450	27360	30620	29810	2,917
2,6	33090	28660	24220	28656,6667	2,964
2,7	34020	29390	24860	29423,3333	3,023
2,8	31230	25590	24690	27170	3,078
2,9	36560	29120	27550	31076,6667	3,136
3	35030	33290	29850	32723,3333	3,235

Table B.1: Pressure calculations for dam break waves of 1m height

The graphs below represent the pressures obtained for the simulation for 1m input wave at all the locations while performing the systematic study, the comFLOW shows reasonable pressure behaviour for these cases.

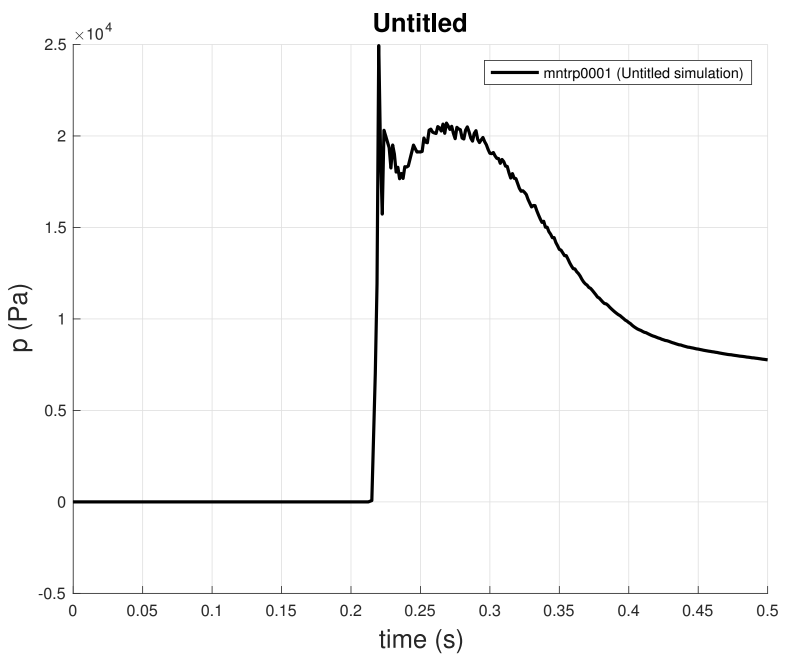


Figure B.15: Pressure at 1.5m distance for 1m wave in time

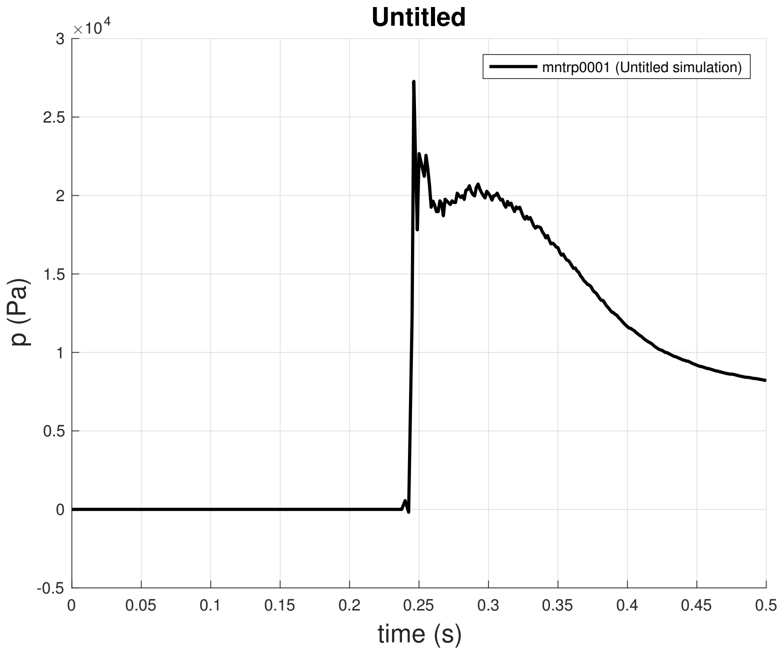


Figure B.16: Pressure at 1.6m distance for 1m wave in time

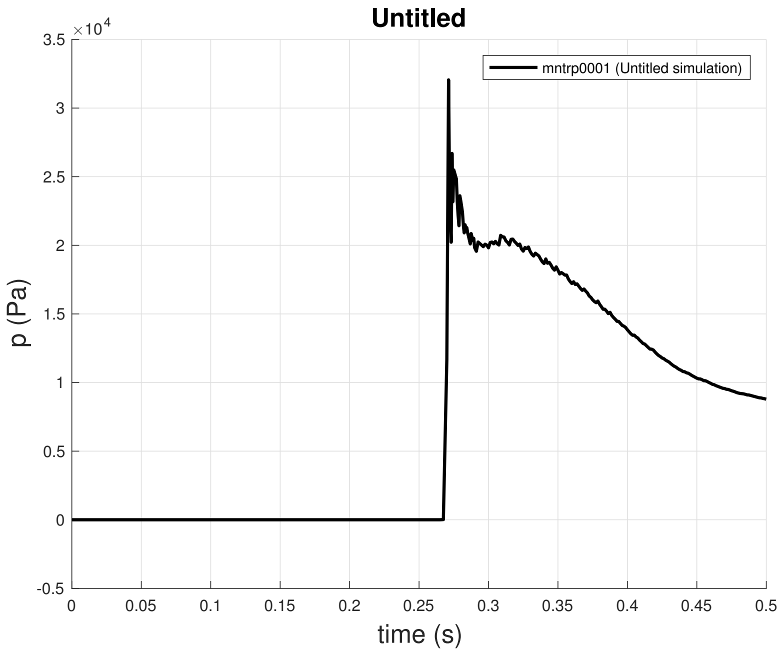


Figure B.17: Pressure at 1.7m distance for 1m wave in time

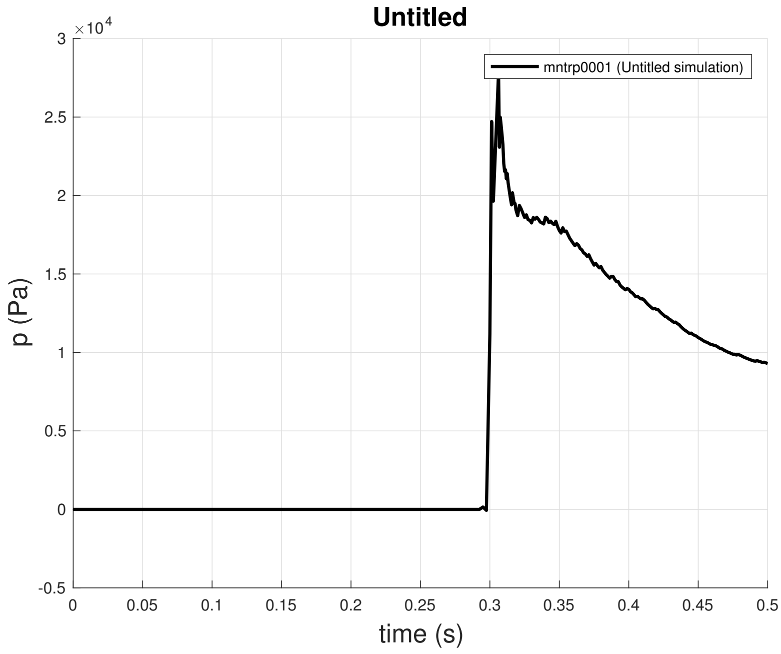


Figure B.18: Pressure at 1.8m distance for 1m wave in time

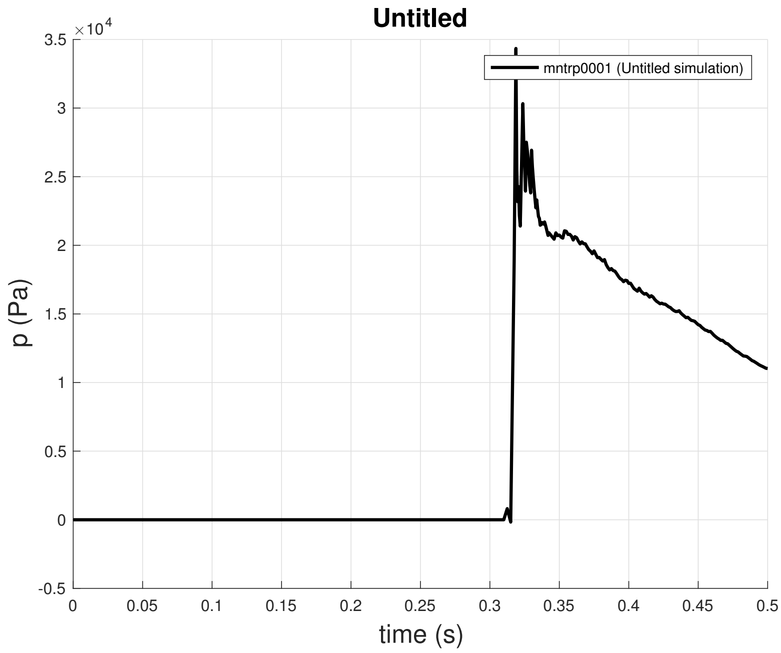


Figure B.19: Pressure at 1.9m distance for 1m wave in time

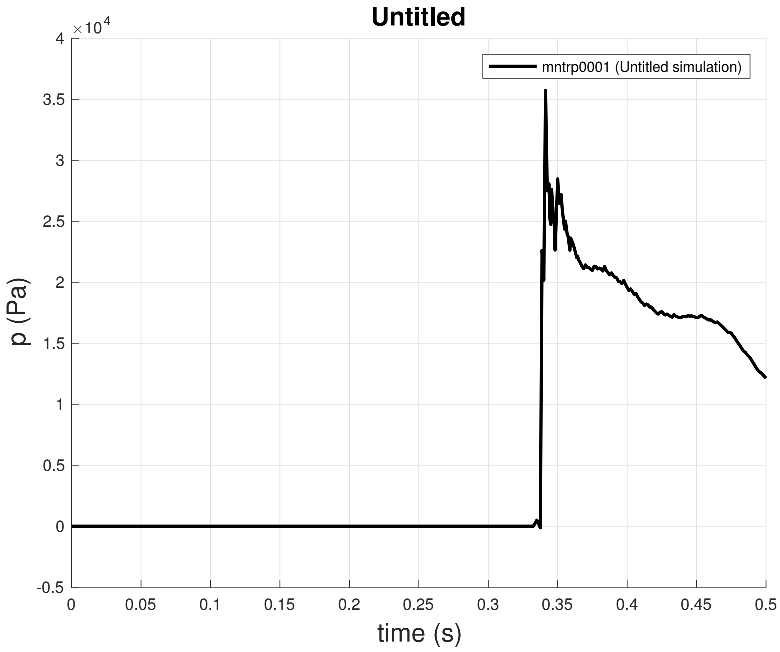


Figure B.20: Pressure at 2m distance for 1m wave in time

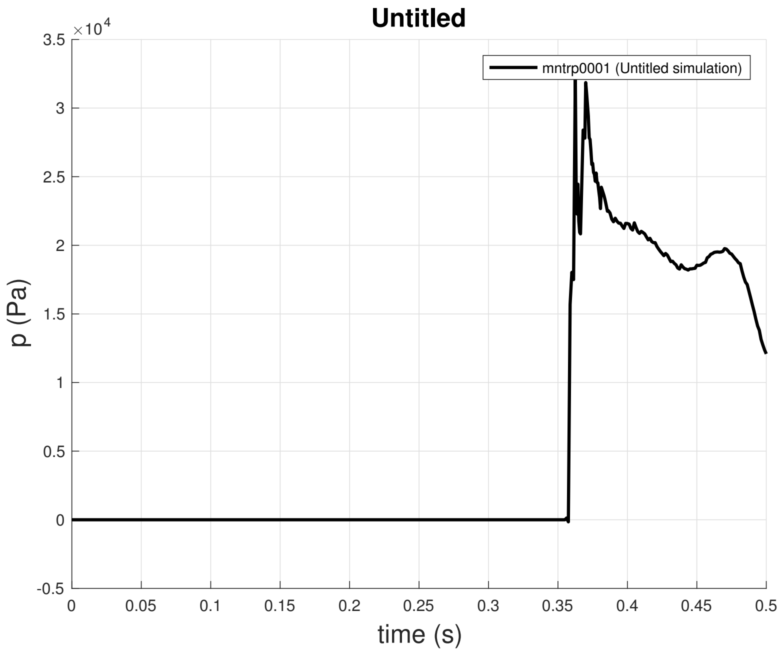


Figure B.21: Pressure at 2.1m distance for 1m wave in time

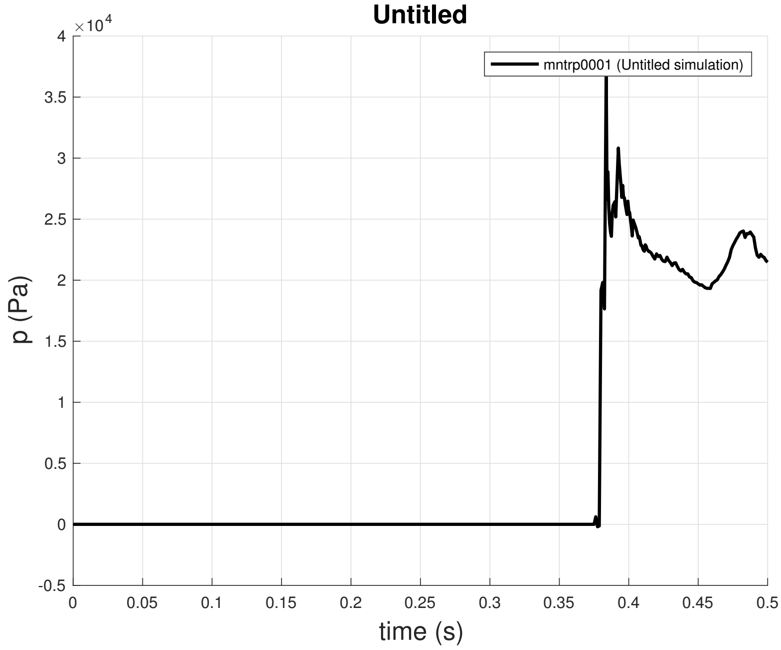


Figure B.22: Pressure at 2.2m distance for 1m wave in time

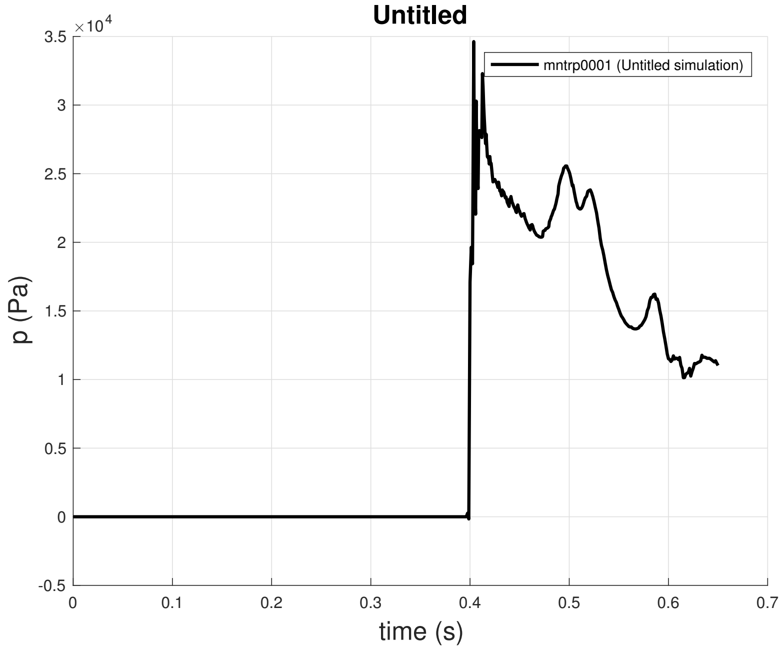


Figure B.23: Pressure at 2.3m distance for 1m wave in time

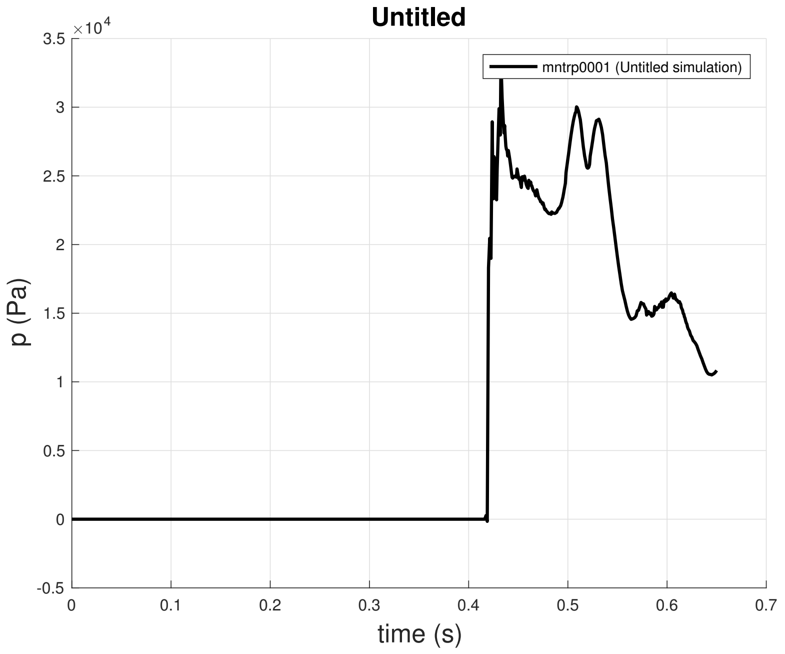


Figure B.24: Pressure at 2.4m distance for 1m wave in time

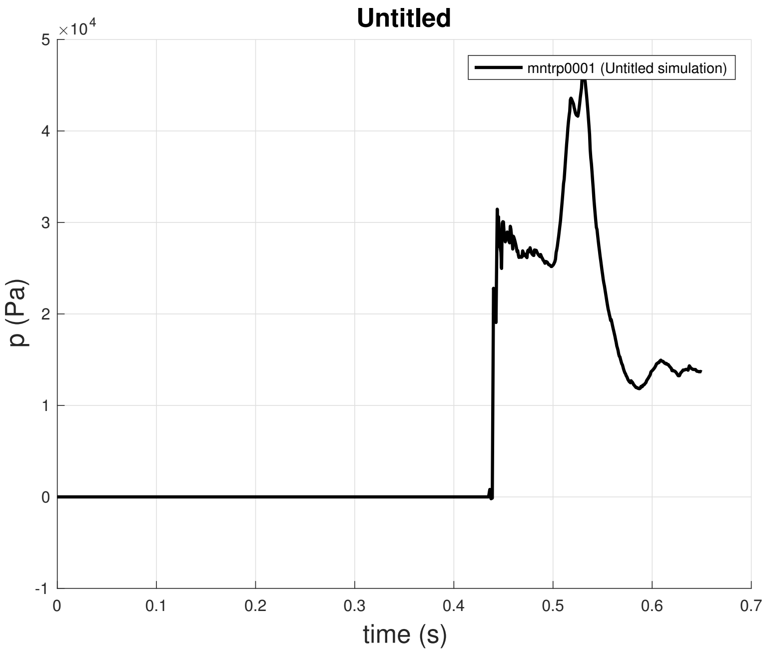


Figure B.25: Pressure at 2.5m distance for 1m wave in time

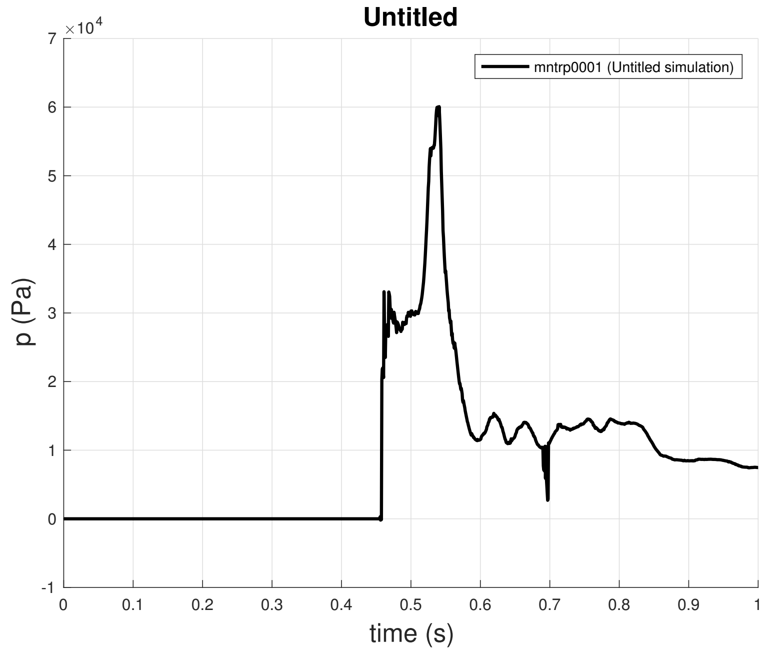


Figure B.26: Pressure at 2.6m distance for 1m wave in time

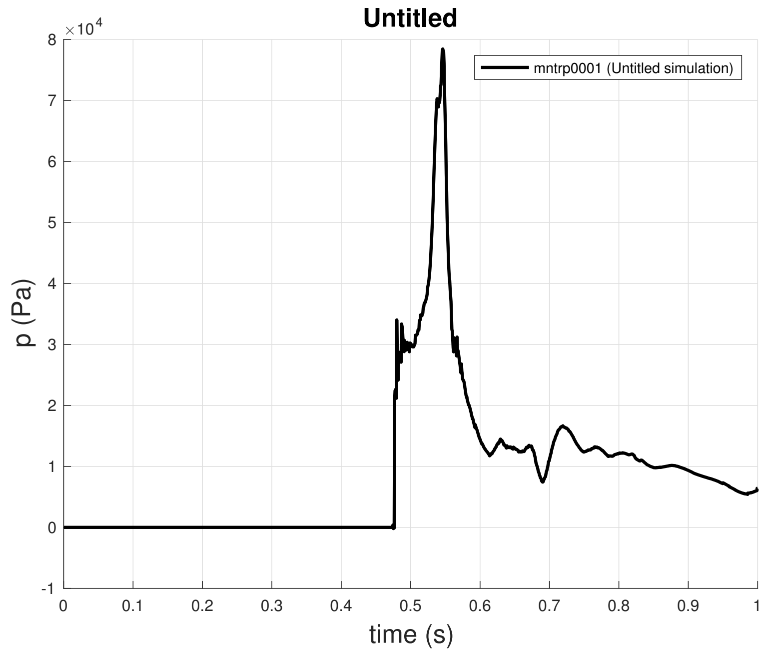


Figure B.27: Pressure at 2.7m distance for 1m wave in time

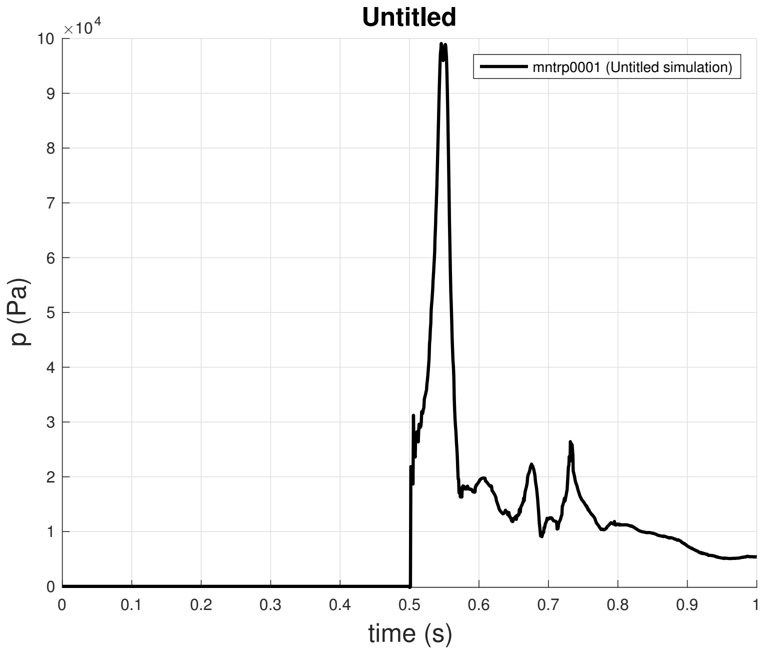


Figure B.28: Pressure at 2.8m distance for 1m wave in time

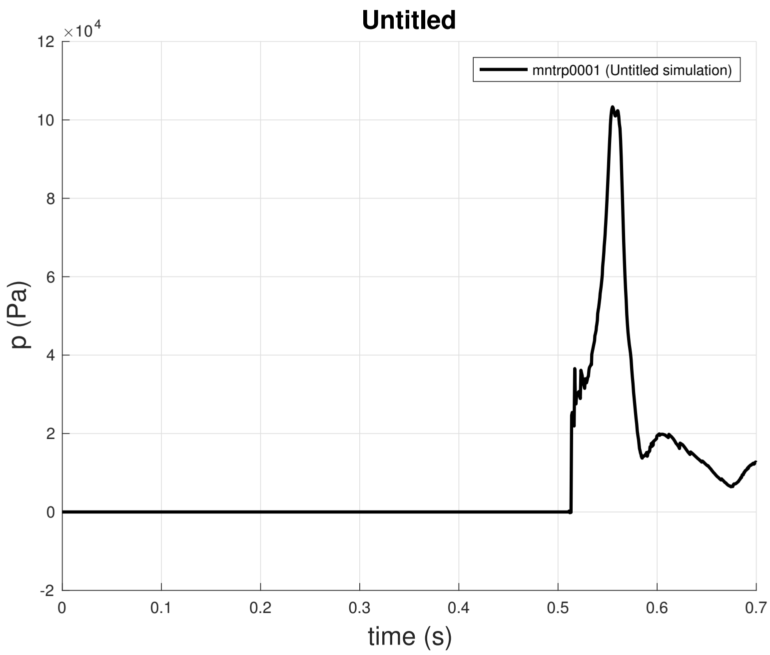


Figure B.29: Pressure at 2.9m distance for 1m wave in time

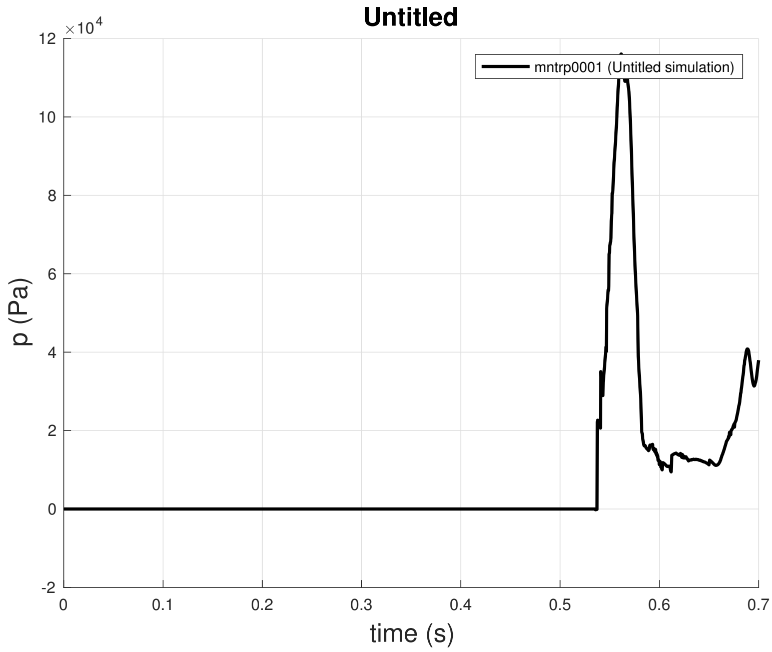


Figure B.30: Pressure at 3m distance for 1m wave in time

C

STRUCTURE IN IRREGULAR WAVES

C.1. FORCE ON STRUCTURE

Figure C.1 represents the force exerted on the structure on deck for three different wave heights for same wave period. No force is observed till 3s since there is no water on the deck, after 3s initial wave hits the deck thus the force exerted is observed to be small, however with the increase in time the wave height increases and thus higher force is exerted on the structure for all three waves. Force on structure depends on the amplitude of irregular waves generated in ComFLOW. Higher the amplitude, higher is the wave height and thus higher the water on a deck which results in a higher force on the structure with increase in time.

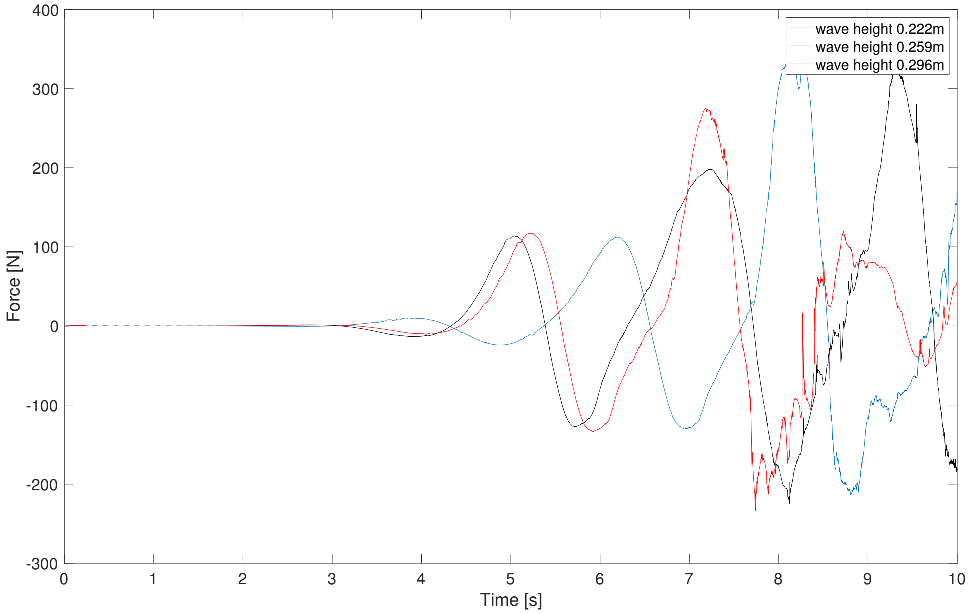


Figure C.1: Force exerted by water on deck on the structure for three waves

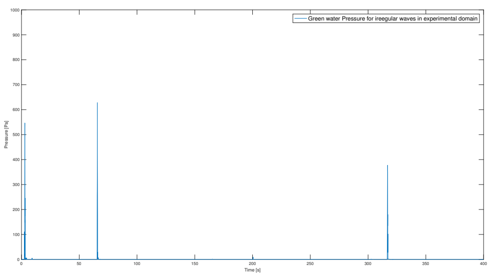
D

Simulations and Proposed experimental setup on Structure in Irregular waves

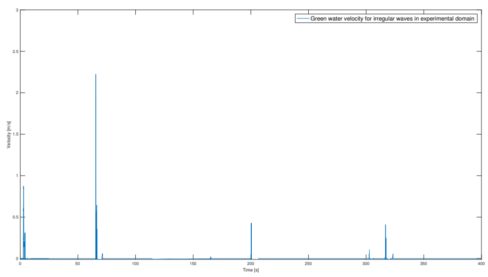
Extreme waves must be considered in the design of offshore vessels, which results in flowing of sea water onto the deck area. This phenomenon is what we call green water because of the lower freeboard of the vessels. Evaluation of this phenomenon is already done with numerical simulations in the previous chapters, this chapter focuses on model tests experiment. It is based on a selection of critical wave conditions, since it is evident that the most severe green water events are not caused by the waves with highest amplitudes. Detailed analysis is beyond just a linear potential theory because wave breaking and free surface fragmentation have to be taken into consideration [27].

D.1. Simulations in ComFLOW with Experimental Domain

Before proceeding with the experimental procedures, simulations are performed keeping same dimensions used in the experiment. The simulation domain is 45m long, 4.22m wide and water depth of 0.6m is kept to observe green water on deck. The pontoon is placed at a distance of 25m from the wave maker. Wave gauges are placed in front and rear of the pontoon to observe mean water level throughout the simulation. Absorbing boundary conditions are used to minimize the numerical reflections in the domain. Monitor points and Force box is also designed around the substructure to calculate the pressure and force respectively, exerted by green water.



(a) Pressure vs Time



(b) Velocity vs Time

Figure D.1: Pressure and velocity in time for Comflow simulation with an experimental domain

Figure D.1 indicates the pressure and its respective velocity exerted by green water events for a duration of 400s. The high velocity causes the simultaneous high pressure peak in this simulation.

D.2. EXPERIMENTAL SETUP

Experiments are conducted at the main Towing tank in a laboratory of Mechanical, Maritime and Materials Engineering(3ME) at TU Delft. This towing tank facility is 142.00m long, 4.22m wide while 2.50m depth can be reached for any experiment purpose. The wavelength of 0.30 – 6.0m regular and irregular can be generated. It has the capability of Flap and piston type wave maker. The water depth for this experiment is kept 0.6m. The wooden pontoon is placed at 25m from the wave maker in the x-direction of the tank representing vessel in an offshore location. The pontoon is 1m long, 4.22m wide and bow height is 0.2m and draft is 0.3m. The structure is placed on this pontoon which is 0.5m from the deck and is 0.2m high. In this way, the resulting pontoon is at the same time very rigid and sufficiently lightweight to reduce inertia. The schematic top view of the experiment is represented in figure D.2. In order to measure the net hydrodynamic force and pressure acting on the structure on deck, the sub-structure is equipped with a load cell inside in which the impact force is entirely transmitted to the load cell.

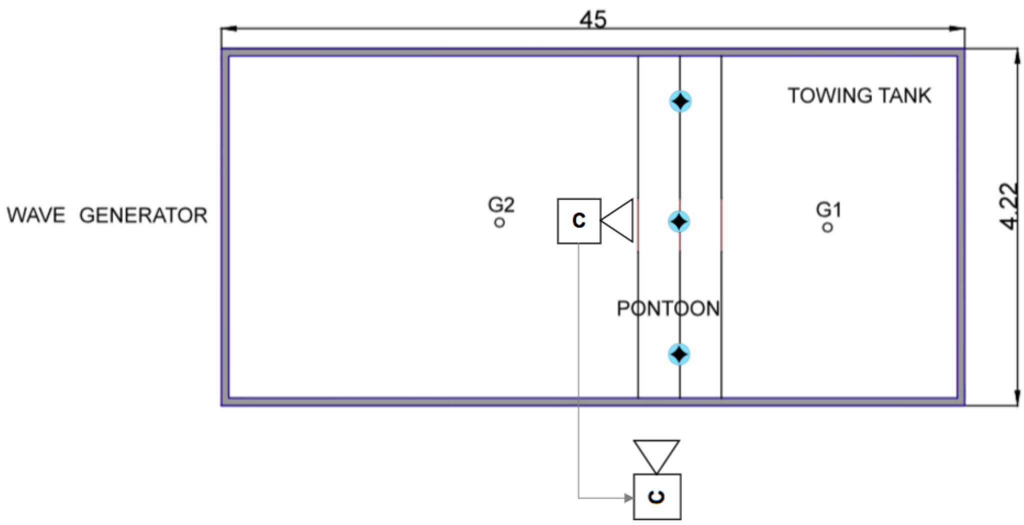


Figure D.2: Plan view sketch of the experimental set-up (measures not scaled, for representation purpose only) G1 and G2 indicates water surface gauges.

This sub-structure is specifically designed to fit all the load cells inside for monitoring purpose. Observations are monitored with two standard camera represented by "C" in figure D.2, one directed to the towing tank through a window where pontoon is placed while another camera is pointed at the pontoon from top to observe the wave shape. For, wave profiles, the camera is located above the maximum surface elevation i.e. 0.6m for the observation of the green water event and the type of wave which causes it. 50 frames can be taken per second, which is sufficient to observe green water phenomenon. Water height, pressures, and forces are calculated during the experiment. Wave gauges are placed in the experimental domain, One probe is placed in front and one probe is placed at the back of the pontoon represented by G2 and G1 respectively in figure D.2 to check the mean water level throughout the experiment. The pontoon is kept with same width as the experimental basin to reduce the complexity of fluid-structure interaction and keeping it as a two dimensional experiment similar to numerical simulations. This substructure will be used to identify the green water impact pressure and forces.