



Numerical assessment of symmetric wedge water-entry impact using OpenFOAM

Basar Ibrahim^{1,*}, Hussein Zekri^{2,3}

¹ *Department of Mathematics, Faculty of Science, University of Zakho, Zakho, Kurdistan Region-Iraq*

² *College of Engineering, The American University of Kurdistan, Duhok, Kurdistan Region-Iraq*

³ *Department of Mechanical Engineering, College of Engineering, University of Zakho, Zakho, Kurdistan Region-Iraq*

Abstract. The hydrodynamic problem of a two-dimensional symmetric wedge vertically entering the water, initially flat, with a deadrise angle of 30° is considered at a constant speed of $3m/s$. The solver overInterDyMFoam in OpenFOAM has been used for the simulation. The effects of turbulent and laminar flow assumptions and for the laminar flow, the effect of viscosity, gravity, and surface tension on pressure distributions along the wedge's walls have been investigated during the early stage of penetration. The $k - \varepsilon$ method has also been used to model the turbulence effect. It is illustrated that turbulence, viscosity, and surface tension have negligible effects on the pressure distribution during the early stage of wedge-water entry. However, the pressure distribution is found to be significantly influenced by gravity.

2020 Mathematics Subject Classifications: 76B10, 76B15, 76-10

Key Words and Phrases: Wedge water-entry, pressure distribution, OpenFOAM

1. Introduction

Slamming loads when a body penetrates water is of great importance in designing the structure of the body. To refer to some previous studies in this field one should start with [25] and [26] who conducted ground-breaking research on estimating slamming loads on the alighting of seaplanes. Since their pioneering work, many researchers have contributed to further investigate this phenomenon. An excellent piece of art of theoretical study in water-entry problems can be seen in [13] and [8]. For more Wagner-based theory see [32],[14],[17], and [15].

In modeling water-entry problems with high speed some parameters, for example, viscosity, surface tension, and gravity are safe to be disregarded. See [2] and [4]. However,

*Corresponding author.

DOI: <https://doi.org/10.29020/nybg.ejpam.v15i4.4615>

Email addresses: basar.ibrahim@stud.uoz.edu.krd (B. Ibrahim),
hussein.zekri@auk.edu.krd (H. Zekri)

the inconsideration of some of these parameters for cases with low and moderate speed may result in inaccurate measurements. See [22], [31], [9], [3], and [24]. Gravity is one of the parameters that plays an important role in water-entry problems with low and moderate entry velocity. This is observed in modeling the emergency landing of aircrafts on water as it is mentioned in [9], also see [1] and [12].

Numerous numerical techniques have been used to model water-entry problems. [33] used Boundary Element Method (BEM) to study the water entry of two-dimensional bodies with arbitrary cross-sections and contrasted results with similarity solutions. They further developed their approach in [32]. With the same numerical approach, BEM, the hydrodynamic of a freely-falling two-dimensional wedge water-entry investigated by [28]. With a laminar assumption, [6] solved this issue using the finite element method (FEM). The Finite Difference Method (FDM) was employed by [29] to address the water-entry problem while simultaneously assuming a laminar flow. In two studies by [5] and [21], they addressed the water-entry problem using the BEM, while the viscosity and turbulence effects were not counted. In the following, we refer to some works that have covered the effect of gravity, surface tension, viscosity, and turbulence in water-entry problems.

In [23] the effect of gravity is considered while investigating wedge water-entry waves. However, for the potential flow theory, it is challenging to adequately represent the nonlinear free surface during a breaking wave, assuming that the flow is irrotational and inviscid. [30], they looked at how gravity affects the 2D flow that results from a symmetrical blunt body moving vertically and slowly into initially still water. They showed that the gravitational correction is of order $O(\delta t^{3/2})$, where t is the time and the parameter δ is small, on each of the size of the wetted region (excluding jets), the free surface elevation, the body's hydrodynamic force, and pressure distribution along the wetted region. They found that gravity changes the surface elevation; however, the gravitational adjustment is minor. Also, they found that gravity reduces the pressure near the contact point and increases it over the middle of the wetted region of the blunt body.

An analytical study presented by [9] to model the water entry of two-dimensional and axisymmetric bodies that accounts for gravity. They also used the fully nonlinear potential flow solver and carried out the simulation. They showed that only after the inclusion of gravity an accurate pressure distribution and the force acting on the body can be achieved. [24] investigated the slamming force decomposition concept for arbitrary 2D shapes with the effect of gravity included. For relatively small values of the entry speed, they found a significant contribution of gravity on the slamming.

Gravity's influence studied by [19] when a Computational Fluid Dynamics (CFD) program was used to numerically solve the Reynolds-averaged Navier-Stokes (RANS) equations. One of their computation examples is a semicircular cylinder with a radius of $5.5m$ entering the water at a constant speed of $10m/s$. Calculations were made to determine the pressure peaks that happen as the cylinder submerges. In their calculations, gravity, surface tension, and turbulence were all taken into account. Additionally, they ran calculations with each of these three effects turned off separately. They demonstrated that only gravity was found to have a substantial impact on their problem, with data showing that the absence of gravity reduced the slamming force coefficient by about 23.2%. In

this study, we shall verify the findings of [19] by using OpenFOAM, including viscosity, turbulence, gravity, and surface tension, when each of these four features was turned off separately. The water-entry problem studied by [20] using the finite element-based finite volume method (FEM-FVM). They analyzed how different deadrise angles and velocities of laminar and turbulent flow influence the water-entry problem. They implemented the $k - \varepsilon$ method to model the turbulence effects. They came to the conclusion that at the peak and hollow locations, there are no appreciable discrepancies in the free surface and force estimates when considering the impact of turbulence on water-entry free surfaces and forces.

In this study the incompressible flow CFD solver in OpenFOAM [10], [11], and [27] is utilized to study the following. The difference between the laminar and turbulent flow and for the case of laminar flow we compare the individual influence of each of viscosity, gravity, and surface tension on the pressure distribution of the symmetric wedge water-entry problem with a deadrise angle of 30° and a constant velocity of $3m/s$.

This paper is structured as follows. The numerical model and its implementation in OpenFOAM are described in Section 2. Numerical validation and results are shown in Section 3 and in Section 4 discussions are manifested. In Section 5, we have drawn the conclusions.

2. Numerical Model

With the continuous increase in computing speed, CFD has gained popularity in commercial use and academic research. As a powerful assistant tool besides experiments and analytical studies, CFD can obtain the parameter fields that are difficult or impossible to be measured and analyzed. But the verification and validation of the simulation results are still necessary to ensure that the results are reliable. The Finite Volume Method (FVM) is the numerical method used in OpenFOAM for fluid flow simulation.

2.1. Governing equations

The continuity equation and associated momentum equations are, respectively:

$$\nabla \cdot \mathbf{U} = 0, \quad (1)$$

$$\frac{\partial}{\partial t} \rho \mathbf{U} = -\nabla \cdot (\rho \mathbf{U} \mathbf{U}) + \nabla \cdot \boldsymbol{\tau} - \nabla p + \rho \mathbf{g}, \quad (2)$$

where " ∇ " is the gradient operator vector, " $\boldsymbol{\tau}$ " is the shear-rate tensor, " \mathbf{U} " is the velocity vector of the fluid, " ∇p " is the pressure gradient, " $\mathbf{g} = (0, g, 0)$ " is the gravitational acceleration vector, and " ρ " is the density.

The VOF (Volume of Fluid) [7] method is built in overInterDyMFoam solver for two isothermal, incompressible fluids with mesh motion, mesh deformation, and adaptive remeshing capabilities. The continuity equation for water (liquid) is as follows:

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\alpha_l \mathbf{U}) + \nabla \cdot [\mathbf{U}_c \alpha_l (1 - \alpha_l)] = 0, \quad (3)$$

where \mathbf{U}_c represents the relative velocity vector, which can be used to create a precise free surface contour. α_l is the volume fraction of the water, and the corresponding volume fraction of air is:

$$\alpha_{air} = 1 - \alpha_l, \quad (4)$$

The interface is localized at the cells where $0 < \alpha_l < 1$. In the VOF method, the transport properties of each fluid are obtained by operating a volume average:

$$\rho = \alpha_l \rho_l + (1 - \alpha_l) \rho_{air}, \quad (5)$$

$$\mu = \alpha_l \mu_l + (1 - \alpha_l) \mu_{air}, \quad (6)$$

where ρ is the mass density of the water and air mixture and μ is its dynamic viscosity. The mass densities of air and water are ρ_l and ρ_{air} , respectively. The dynamic viscosities of air and water are μ_l and μ_{air} , respectively.

2.2. Turbulence model

The two-equation models include the $k - \varepsilon$ (k-epsilon) turbulence model, where the turbulent kinetic energy k and the rate at which that energy dissipates ε solve model transport equations for two turbulence quantities, [16] and [18]. The most common CFD reference, the standard $k - \varepsilon$ turbulence model, was used. For k and ε the transport equations are as follows:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + P_{kb} \quad (7)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \varepsilon) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} (C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon + C_{\varepsilon 1} P_{\varepsilon b}). \quad (8)$$

where μ_t is the turbulent viscosity; P_k is the turbulent kinetic energy production rate; P_{kb} and $P_{\varepsilon b}$ are the buoyancy effects; and $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and σ_ε are constants, model coefficients.

2.3. Model and Mesh Setup

Figure 1 shows the initial two-dimensional geometry of the symmetric wedge water-entry problem where x and y are the horizontal and vertical axes, respectively. The wedge (has a deadrise angle of 30° and a width of $1.2m$), with a constant velocity of $3m/s$ moves vertically into the water that is initially at rest. The geometry of the domain is a $10m \times 4m \times 0.01m$ cuboid, where the third dimension is neglected. The number of cells for the background (water and air) is 180000 and for the wedge is 75600. To keep the courant number at 0.2 we performed the simulation with time step 5×10^{-5} . With these number of cells and time step, each simulation is performed within 10 hours. To satisfy the accuracy and the far-field conditions of the water entry problem, the domain is considered to be large enough that the wedge's motion has zero effect on the velocity and pressure field on the boundaries.

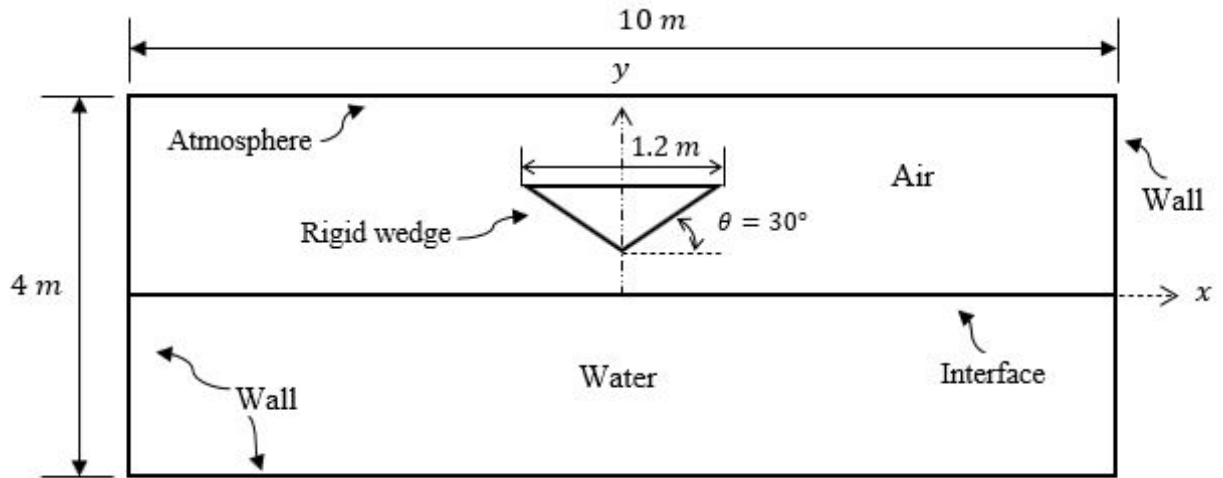


Figure 1: The initial two-dimensional geometry of the wedge water-entry problem.

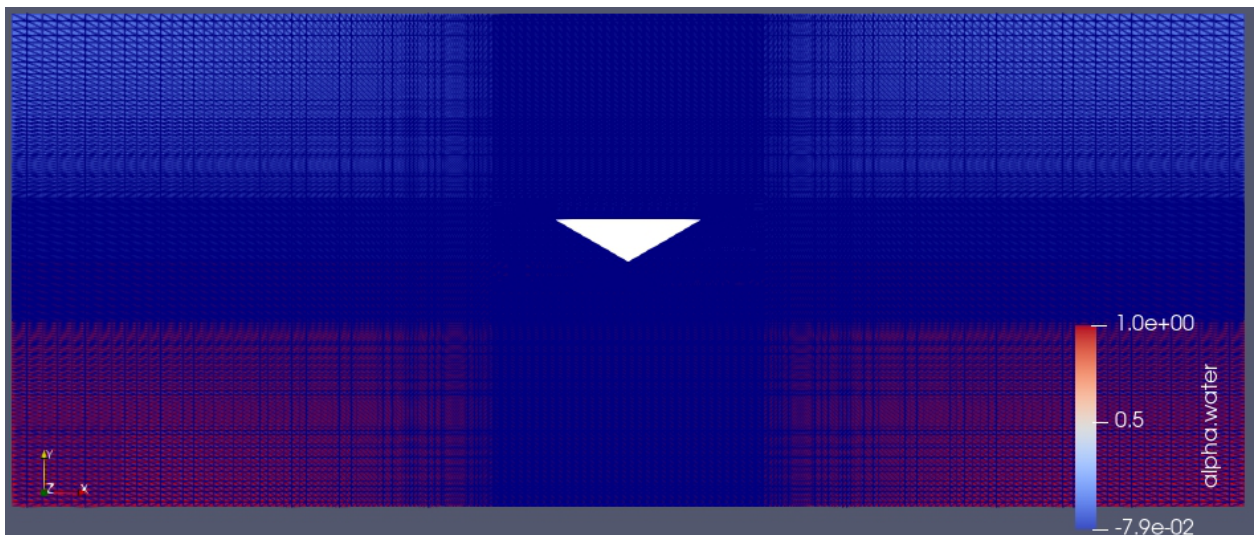


Figure 2: Numerical modeling setup and computational mesh.

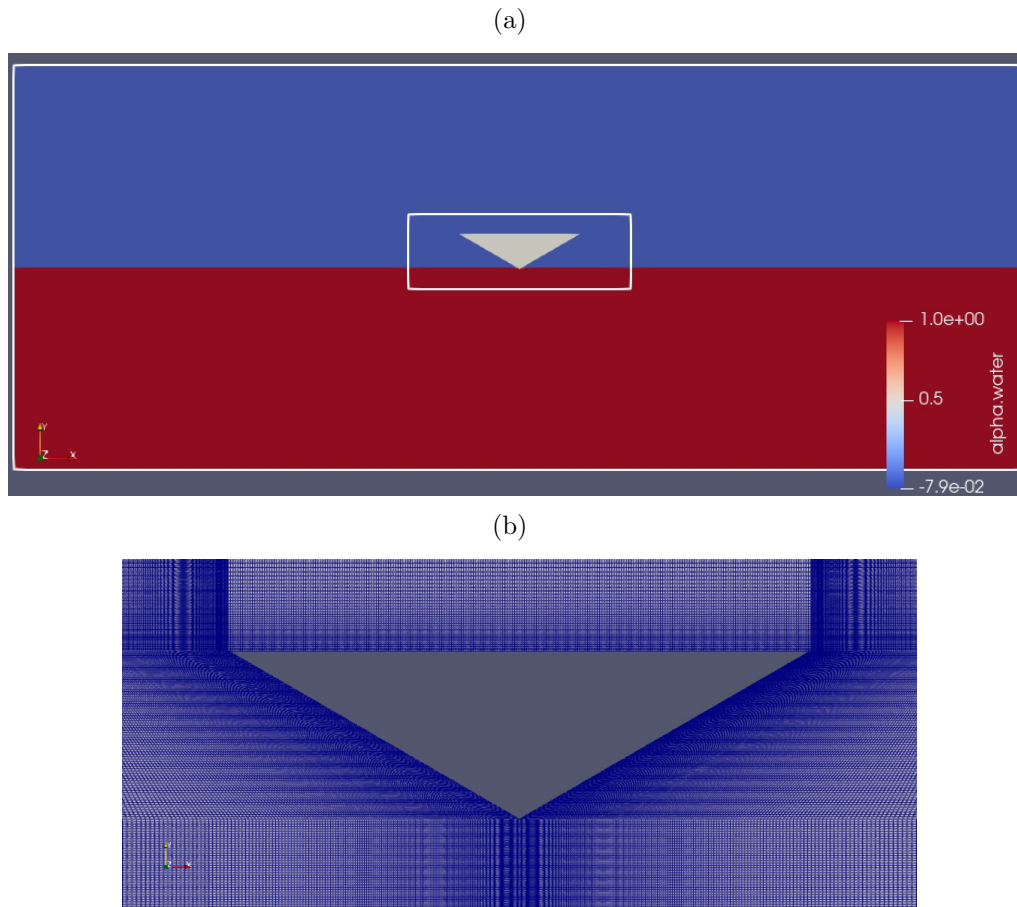


Figure 3: Numerical modeling setup and computational mesh

As shown in Figure 1, the interface is initially set at $y = 0$, which consists of 3 cells vertically. The top boundary is named atmosphere, and the left, right, and bottom boundaries are named walls.

For the geometry, the blockMesh utility is used to establish the computational mesh and setFields to generate fields (see Figure 2). The cell size gradually increases toward the border walls and the wedge is simulated in the center of the sub-mesh (see Figure 3a), the mesh around the wedge is depicted in Figure 3b in the block's center, where the size of the cells is uniform.

3. Numerical results

We initially conduct a validation to confirm our approach and numerical process. We consider the vertical water entry into the calm water of a symmetrical wedge with deadrise angles of 45° at a constant speed of $4m/s$, and the numerical outcomes are evaluated against the data given in [33].

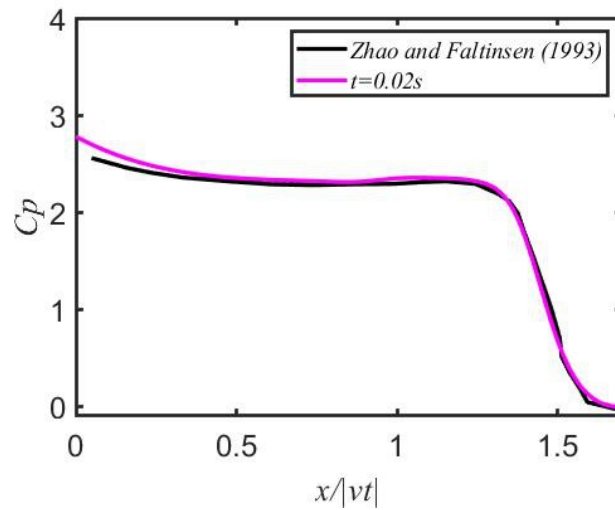


Figure 4: Pressure distribution on the wedge surface.

Figure 4 displays the pressure coefficient, C_p , which is in good agreement with the work in [33]. Here C_p is defined as $C_p = (p - p_0) / (0.5\rho v^2)$, where p is the pressure on the surface of the wedge, v is the vertical velocity, and $p_0 = 1$ bar.

As it was already said, the current study is concentrated on how turbulent assumptions and laminar assumptions affect the pressure distribution on the body surface of the wedge water-entry problem, as well as surface tension, viscosity, and gravity for the laminar case. The problem is solved for a deadrise angle of 30° with laminar and turbulent assumptions. After that, the simulation for the laminar case computation is carried out with and without each of these features (gravity, viscosity, and surface tension) switched off individually. The duration of the simulation is short. The penetration depth is comparable with the height of the water pile-up. In this work, only the results on the right-half of the wedge are shown because of the symmetry of this problem.

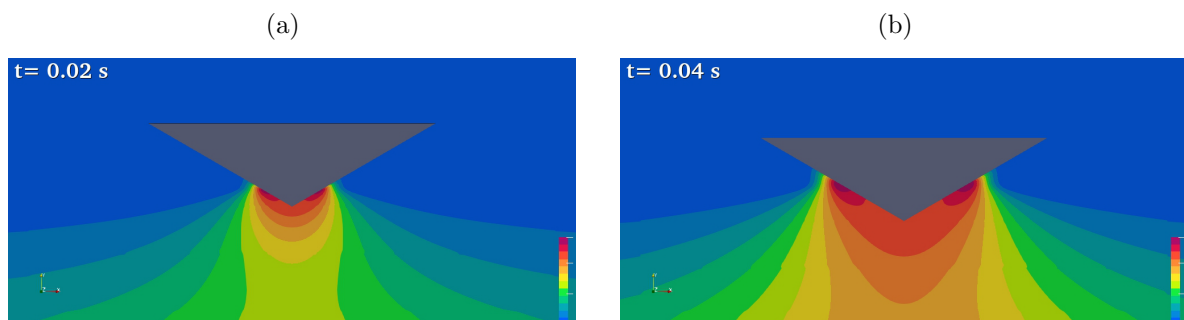


Figure 5: Predicted pressure contours during wedge water entry for the laminar case with all features included. At $t = 0.02s$ (left) and $t = 0.04s$ (right).

Figure 5 shows predicted pressure contours of the wedge with deadrise angles of 30° with a constant vertically downward velocity of $3m/s$ at two different time steps. The spray root is where there is the highest pressure. According to [32], the wedge's keel is where there is the highest pressure when the deadrise angle is more than 40° . They showed that when the deadrise angle is less than 40° before separation, the pressure is at its highest around the spray root of the jet. After the flow divides, the highest pressure rapidly decreases near the separation point and shifts to the keel of the wedge.

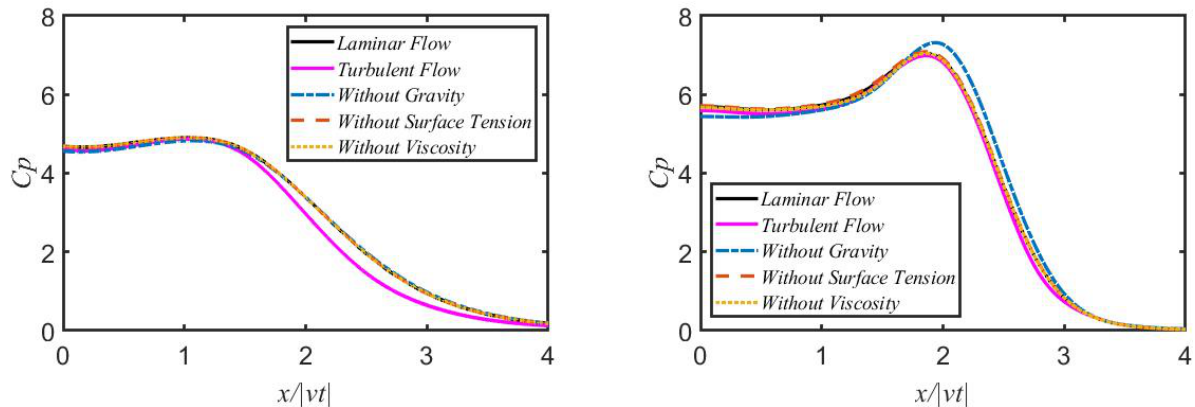


Figure 6: Non-dimensional pressure distribution comparison of the water entry of the wedge. At $t = 0.02s$ (left) and $t = 0.04s$ (right).

4. Discussions

During the early stage of impact and at two different time steps the results of non-dimensional pressure distribution along the wetted surface of the wedge are drawn. Figure 6 displays the pressure coefficient, C_p . The effects of turbulent and laminar flow assumptions and the influence of viscosity, gravity, and surface tension on the pressure distribution along the surface of the wedge of this wedge water-entry problem with a deadrise angle of 30° and a constant velocity vertically downward of $3m/s$ are plotted at two different but early time steps in Figure 6. It shows that there is no significant difference between laminar and turbulent simulations. Also, viscosity and surface tension effects are found to be insignificant. This confirms the results by [19] and [20]. The only factor that is found to be significantly influential is gravity. The effect of gravity is more visible on pressure distribution compared with the other features, even during the early stage of impact. The non-dimensional pressure distribution is noticeably affected by gravity. This can be observed in Figure 7d. From the right figure, at $t = 0.04s$, it is noticeable that gravity also affects the size of the wetted region.

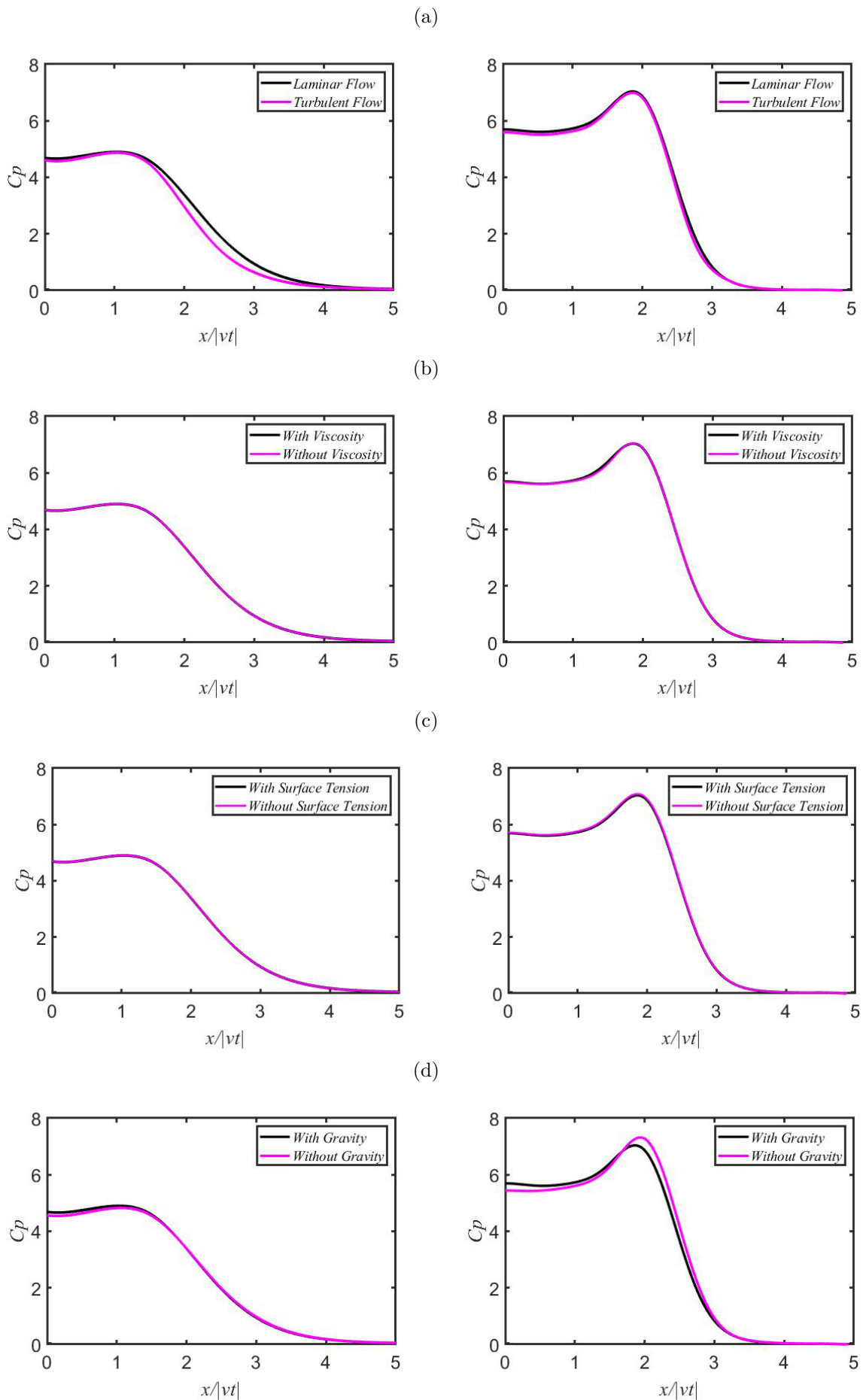


Figure 7: Non-dimensional pressure distribution comparison of the water entry of the wedge. At $t = 0.02s$ (left) and $t = 0.04s$ (right).

Figure 7a compares the numerical results of the non-dimensional pressure distribution in laminar flow and turbulent flow simulation at the early stage of impact. There is no significant difference between the peak pressure, which may typically be disregarded in this field of problems. There is some significant difference at time step $t = 0.02s$. However, this does not belong to the wetted surface on the wedge (excluding the jets). In Figure 7b and Figure 7c, the pressure distribution with and without viscosity and surface tension effects are shown and are graphically identical. The effects of viscosity and surface tension are extremely small and negligible. Gravity significantly contributed to the pressure at the second time step, as illustrated in Figure 7d. By including gravity, a slight peak pressure decrease appeared. At the instant $t = 0.02s$, gravity increases the pressure along the wetted region, while at $t = 0.04s$, this contribution is different. The correction due to gravity is such a way that pressure increases in the interval around the center of impact and decreases in the inner region where the maximum pressure occurs. Similar results analytically (using the perturbation methods) are shown in [30] for a blunt body with a small deadrise angle.

5. Conclusions

The present study uses overInterDyMFoam solver in OpenFOAM to investigate the effects of turbulent and laminar flow assumptions and the influence of viscosity, gravity, and surface tension on pressure distributions on the symmetric wedge with a deadrise angle of 30° vertically entering the calm water at the constant speed of $3m/s$. The $k - \varepsilon$ technique has been used to model the turbulence effect. The numerical scheme which is used for simulation is validated with the non-dimensional pressure distribution data published in [33]. Figure 4 shows that a good agreement is achieved.

Simulations of wedge-entry into water at an early stage of impact have been carried out. The first simulation performed explored the difference between laminar and turbulent flow shown in Figures 7a. Both of the figures Figure 7a show that the non-dimensional pressure distribution on the side of the wedge experiences an insignificant difference when turbulence is accounted for. (This is to be expected owing to the short time scale of this stage of the impact, during which fluid inertia dominates the pressure response). Then we performed a simulation with laminar flow to investigate the influences of viscosity, surface tension, and gravity on the wedge impact. Each of these three physical aspects of the flow is switched off individually, and the results are displayed in Figures 7b,7c,7d. The four plots in Figure 7b,7c show that viscosity and surface tension make a negligible contribution to the non-dimensional pressure distribution along the surface of the wedge. However, in contrast to the turbulent, viscous, and surface-tension effects, the influence of gravity is more significant on the non-dimensional pressure distribution. Gravity is even noticeable during this early stage of impact. Both of the plots (left and right) in Figure7d show that gravity changes the pressure distribution. In Figure 7d (at time $t = 0.04s$), it is clear that gravity reduces the value of the maximum in pressure and moves it closer to the centreline ($x = 0$). These influences grow over time. The pressure distribution is increasingly affected by gravity as time goes by. These numerical results show the same

influences due to gravity as analytical results for a blunt body with a small deadrise angle, obtained using perturbation methods [30]. For a blunt body, the theory shows a reduction in the maximum value of the pressure and a shift in its position, closer to the centreline ($x = 0$).

References

- [1] L Bensch, V Shigunov, G Beuck, and H Söding. Planned ditching simulation of a transport airplane. In *KRASH users seminar*, pages 411–439, 2001.
- [2] Mark J Cooker and DH Peregrine. Wave impact pressure and its effect upon bodies lying on the sea bed. *Coastal Engineering*, 18(3-4):205–229, 1992.
- [3] AC Fairlie-Clarke and T Tveitnes. Momentum and gravity effects during the constant velocity water entry of wedge-shaped sections. *Ocean Engineering*, 35(7):706–716, 2008.
- [4] Odd M Faltinsen. *Hydrodynamics of high-speed marine vehicles*. Cambridge university press, 2005.
- [5] Jie Gao, Yong Hu Wang, and Ke An Chen. Numerical simulation of the water entry of a wedge based on the complex variable boundary element method. In *Applied Mechanics and Materials*, volume 90, pages 2507–2510. Trans Tech Publ, 2011.
- [6] Luo Hanbing, Shan Wang, and C. Guedes Soares. Numerical prediction of slamming loads on a rigid wedge subjected to water entry using an explicit finite element method. *Advances in marine structures*, pages 41–47, 2011.
- [7] Cyril W Hirt and Billy D Nichols. Volume of fluid (vof) method for the dynamics of free boundaries. *Journal of computational physics*, 39(1):201–225, 1981.
- [8] S. D. Howison, J. R. Ockendon, and S. K. Wilson. Incompressible water-entry problems at small deadrise angles. *Journal of Fluid Mechanics*, 222:215–230, 1991.
- [9] F Hulin, A Del Buono, Alan Tassin, G Bernardini, and A Iafrati. Gravity effects in two-dimensional and axisymmetric water impact models. *Journal of Fluid Mechanics*, 944, 2022.
- [10] Hrvoje Jasak. Error analysis and estimation for the finite volume method with applications to fluid flows. 1996.
- [11] Hrvoje Jasak. Openfoam: open source cfd in research and industry. *International Journal of Naval Architecture and Ocean Engineering*, 1(2):89–94, 2009.
- [12] TI Khabakhpasheva, AA Korobkin, Kevin J Maki, and Sopheak Seng. Water entry and exit with large displacements by simplified models. In *Proceedings of the 31 St International Workshop on Water Waves and Floating Bodies, Michigan, USA*, 2016.

- [13] A A Korobkin and V V Pukhnachov. Initial stage of water impact. *Annual Review of Fluid Mechanics*, 20(1):159–185, 1988.
- [14] AA Korobkin. Asymptotic theory of liquid–solid impact. *Philosophical Transactions of the Royal Society of London. Series A: Mathematical, Physical and Engineering Sciences*, 355(1724):507–522, 1997.
- [15] AA Korobkin. Second-order wagner theory of wave impact. *Journal of Engineering Mathematics*, 58(1):121–139, 2007.
- [16] Brian Edward Launder and Dudley Brian Spalding. The numerical computation of turbulent flows. In *Numerical prediction of flow, heat transfer, turbulence and combustion*, pages 96–116. Elsevier, 1983.
- [17] James Mark Oliver. *Water entry and related problems*. PhD thesis, 2002.
- [18] Wolfgang Rodi. *Turbulence models and their application in hydraulics: A state-of-the-art review*. Routledge, 2017.
- [19] MOHAMMED S Seif, S Mousaviraad, S Saddathosseini, and Volker Bertram. Numerical modeling of 2-d water impact in one degree of freedom. *Síntesis Tecnológica*, 2(2):79–83, 2005.
- [20] Roya Shademani and Parviz Ghadimi. Numerical assessment of turbulence effect on forces, spray parameters, and secondary impact in wedge water entry problem using k- ϵ method. *Scientia Iranica*, 24(1):223–236, 2017.
- [21] Hanbing Sun, Jin Zou, Jiayuan Zhuang, and Qingxu Wang. The computation of water entry problem of prismatic planing vessels. In *2011 3rd International Workshop on Intelligent Systems and Applications*, pages 1–4. IEEE, 2011.
- [22] Hui Sun and Odd M Faltinsen. The influence of gravity on the performance of planing vessels in calm water. *Journal of Engineering Mathematics*, 58(1):91–107, 2007.
- [23] SY Sun, SL Sun, and GX Wu. Oblique water entry of a wedge into waves with gravity effect. *Journal of Fluids and Structures*, 52:49–64, 2015.
- [24] Zhe Sun, Xu-peng Sui, A Korobkin, Li Zou, and Zhi Zong. Slamming force decomposition with gravity effect. *Journal of Fluids and Structures*, 114:103694, 2022.
- [25] T Von Karman and FL Wattendorf. The impact on seaplane floats during impact. *NACA TN*, 321, 1929.
- [26] Herbert Wagner. The phenomena of impact and planing on water. *National Advisory Committee for Aeronautics Translation 1366, ZAMM*, 12:193–215, 1932.
- [27] Henry G Weller, Gavin Tabor, Hrvoje Jasak, and Christer Fureby. A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in physics*, 12(6):620–631, 1998.

- [28] GX Wu, H Sun, and YS He. Numerical simulation and experimental study of water entry of a wedge in free fall motion. *Journal of Fluids and Structures*, 19(3):277–289, 2004.
- [29] Qingyong Yang and Wei Qiu. Numerical solution of 3-d water entry problems with a constrained interpolation profile method. *Journal of offshore mechanics and Arctic engineering*, 134(4), 2012.
- [30] Hussein J Zekri, Alexander A Korobkin, and Mark J Cooker. Gravity effect on water entry during an early stage. *Journal of Fluid Mechanics*, 916, 2021.
- [31] Hussein Jebrail Zekri. *The influence of gravity on fluid-structure impact*. PhD thesis, University of East Anglia, 2016.
- [32] Ran Zhao, O Faltinsen, and J Aarsnes. Water entry of arbitrary two-dimensional sections with and without flow separation. In *Proceedings of the 21st symposium on naval hydrodynamics*, pages 408–423, 1996.
- [33] Rong Zhao and Odd Faltinsen. Water entry of two-dimensional bodies. *Journal of fluid mechanics*, 246:593–612, 1993.