# Numerical investigation on the water entry impact characteristics of autonomous underwater vehicles

Ahmad Zamir Chaudhry, Yao Shi, Guang Pan
School of Marine Science and Technology,
Northwestern Polytechnical University, Xi'an, China
Email: ahmadzamirch@mail.nwpu.edu.cn

Abdul Shakoor, Syed Shah Khalid
CESAT, Islamabad, Pakistan

***Abstract*—Water entry of autonomous underwater vehicles (AUVs) is an unsteady and complex process accompanied by a huge hydrodynamic impact force which consequently affects the structure globally and locally. Therefore, precise modeling of this phenomenon is indispensable for the structure design of the vehicle. In this article, numerical model employing an Arbitrary-Lagrangian Eulerian (ALE) formulation is used to study the water entry impact of AUV. A penalty coupling algorithm will be employed which allows the interaction between the solid and the fluids. The feasibility and precision of the numerical technique is validated by the experimental data of the water entry of a decelerating object. After validation, the proposed numerical method is employed to examine the hydrodynamic behavior of AUV water entry under various launch parameters at the initial stage of impact. Numerical results from ALE method are also compared with smooth particle hydrodynamics (SPH) method. This reveals that ALE method can accurately simulate large deformation problems with less computational cost. The analysis results indicate that the time period at which the impact acceleration reaches its maximum value decreases as the launch velocity of the AUV increases. Axial and radial impact loads are calculated at various launch angles for fixed impact velocity of the vehicle. It is shown that oblique water entry of AUV is more sensitive to the radial impact load. It is concluded that water entry angle and launch velocity are the crucial parameters greatly influencing the impact characteristics of the AUV. Quantitative comparison between numerical and experimental data proves that the proposed numerical algorithm can reliably be used for water entry impact problems at high velocities.**

***Keywords—Water entry; autonomous underwater vehicles; hydrodynamic impact force; Arbitrary-Lagrangian Eulerian (ALE)***

## I. INTRODUCTION

Water entry is highly nonlinear and transient phenomenon that encompasses the interaction of solid object with air and water. The hydrodynamic impact between AUV and water may induce impulsive impact loads of large magnitude which can severely damage the structure. Therefore, the precise calculations of impact load characteristics of the AUV during the early stage of water entry are of paramount significance for the AUV operative design.

Significance and severity of water entry problem has motivated numerous researchers during the past century. Von Karman [1] carried out pioneering analytical studies on the hydrodynamics of a landing seaplane floats by considering simplified two dimensional and rigid model. Wagner [2] generalized the work of Von Karman and developed the theory of water entry by considering the impact of two dimensional object into the water including the effect of water splash-up and proposed an asymptotic solution. Wagner work is still considered as a benchmark for analytical studies. Cole et al. [3] carried out experimental studies on the water entry of pointed nose objects. He used full-sized models and compared the results with test data of small scaled models equipped with data acquisition system onboard.

In recent years, research on the water entry of projectiles has been significantly increased. It has extensive range of applications in marine geoscience, civil and commercial sector but the most imperative applications in this field are in the military use [4]. The study in this field with precision and accuracy is of key importance. Therefore, advances in analytical, experimental and numerical solutions including the development of finite element methods and new modeling techniques for analysis of dynamic characteristics upon fluid structure interaction are rapidly increased. Numerical techniques are developed to investigate the hydrodynamic characteristics of two-dimensional and axisymmetric arbitrary-shaped bodies upon water entry with the assumption of incompressible and inviscid fluid with gravity and surface tension effects were neglected [5], [6]. Theoretical and experimental investigations were performed on the cavity dynamics of spheres and cylinders while entering into water [7]–[9]. It was concluded that the cavities created by impacting cylinders were almost the same size as the cylinder whereas the cavities formed by impacting spheres were larger than the sphere. They also concluded that the cavity pinch-off time is proportional to the diameter of the impacting body, but was independent of the impact velocity. It was also observed that the performance of the projectiles was greatly enhanced by decreasing the tip size. In addition, length-to-diameter ratio and tip shape also improves the performance of the projectile during water entry.

Alizadeh et al. [10] experimentally and numerically investigated the water entry impact of a blunt body focusing on the jet formation. Simulation was performed using a commercial CFD package (FLOW-3D). By comparing the results of numerical simulation with experiments, it was concluded that high grid resolution especially in the vicinity of the body is required in order to capture jet formation and splashes around the body correctly. Challa et al. [11] investigated water entry impact characteristics of water landing object and used finite-element ALE formulation and smoothed particle hydrodynamics (SPH) methods. They concluded that

978-1-7281-4676-8/20/$31.00©2020 IEEE

SPH algorithm is less intricate in simulating the water entry process because it is a mesh-free numerical technique and it can easily model the impact problems with large deformation. However, the SPH method is computationally less efficient than ALE method. Erfanian et al. [12] also numerically studied the hydrodynamic behavior of low speed hemispherical head projectile upon water entry. Hassoon et al. [13] performed numerical simulation of the water entry of two dimensional rigid structures using ALE formulation built-in explicit finite element in ABAQUS software. The prediction of maximum pressure and hydrodynamic forces were compared with the analytical formulations of the rigid body. Yan et al. [14] investigated water entry of an air-launched AUV with various launch velocities and angles using experimental tests as well as numerical simulations. For numerical modeling of the water entry process of full-size AUVs, they employed coupled FEM and SPH method using LS-DYNA software. The air cavity shape, trajectory and the impact accelerations of AUV obtained from numerical results were compared with that of experiments. It was concluded from the good comparative results that the proposed numerical method is suitable for water entry problems with large deformation. Chen Chen et al. [15] numerically investigated the compressibility effects and hydrodynamic characteristics of multiphase cavitating flow during the water-entry process for a projectile at transonic speed. It was observed that drag coefficient, cavity shape and cavitation in the cavity are mainly affected by the compressibility of fluids. Derakhshanian et al. [16] compared the performance of different numerical algorithms for the simulation of oblique water entry of objects at low velocities. Through experimental validation, it was confirmed that ABAQUS software can reliably be used for oblique water entry problems. Shi et al. [17] employed explicit finite element method to design the mitigator of AUV and its buffering performance analysis. In another study, Shi et al. [18] investigated the cavity dynamics and hydrodynamic impact characteristics of high speed projectiles using numerical simulations. Gao et al. [19] numerically studied the oblique water entry of projectiles at different launch conditions. It was observed that projectile shows ricochet behavior and less smooth cavity wall at small water entry angle.

In this study, water entry impact characteristics of autonomous underwater vehicle are investigated using explicit finite element method by employing ALE formulation in LS-DYNA software. Numerical algorithm is validated by the experimental work of Aristoff et al. [9]. Hydrodynamic behavior of AUV is studied at various launch velocities and water entry angles. Results obtained from ALE method are compared with SPH results. Good quantitative comparison between numerical and experimental results establishes that the employed numerical technique is capable of simulating the high speed water entry problems to an acceptable accuracy.

## II. Governing Equations and Numerical Model

### *A. Governing Equations*

For large deformation fluid-structure interaction (FSI) problems, neither the Lagrangian nor the Eulerian formulations alone can accurately predict the complete simulation. Therefore, ALE method is best suited for these problems, as it contains both Lagrangian and Eulerian formulations. It uses Lagrangian formulation for the structural elements and Eulerian formulation for the fluid domain, and a coupling method is introduced for fluid and structure interaction. In this numerical study, the penalty coupling algorithm is employed with explicit FEM method using Navier-Stokes solver. The fluid motion equations and conservation equations are presented here [20]. Navier-Stokes equations with ALE description can be described as follow:

$$\frac{df(\vec{X},t)}{dt} = \frac{df(\vec{x},t)}{dt} + \left(\vec{v}-\vec{w}\right).\vec{grad}\, f\left(\vec{x},t\right) \tag{1}$$

where X is the Lagrangian coordinate, x the ALE coordinate and (v – w) is the relative velocity between the material and the mesh.

In ALE formulation, conservation equations of mass, momentum, and energy for a Newtonian fluid in the reference domain are described by:

$$\frac{\partial \rho}{\partial t} + \rho\, div\left(\vec{v}\right) + \left(\vec{v}-\vec{w}\right) grad\left(\rho\right) = 0 \tag{2}$$

$$\rho \frac{\partial \vec{v}}{\partial t} + \rho\left(\vec{v}-\vec{w}\right).\overline{\overline{grad}}\left(\vec{v}\right) = \vec{div}\left(\overline{\overline{\sigma}}\right) + \vec{f} \tag{3}$$

$$\rho \frac{\partial e}{\partial t} + \rho\left(\vec{v}-\vec{w}\right).\overline{grad}\left(e\right) = \overline{\overline{\sigma}} : \overline{\overline{grad}}\left(\vec{v}\right) + \vec{f}.\vec{v} \tag{4}$$

where ρ is the density and σ is the total Cauchy stress which is given by

$$\overline{\overline{\sigma}} = \mu\left(\overline{\overline{grad}}\left(\vec{v}\right) + \overline{\overline{grad}}\left(\vec{v}\right)^{T}\right) - p.\overline{\overline{Id}} \tag{5}$$

where p is the pressure and μ is the dynamic viscosity

### *B. Numerical Model and itsValidation*

In present study, monolithic scheme will be used for ALE formulation in which the governing equation for the structure and fluid domain will be solved simultaneously for all the unknown variables.

The interaction between solid and fluids is computed based on the penalty coupling algorithm using LS-DYNA. In fully coupled FEM-ALE method, coupling method calculates the forces at the interface because of impact of the structure (slave) on the fluid (master), these forces are applied to the master and slave nodes and prevent the nodes from penetrating through interface surface and behaves like the spring system as shown in Fig. 1. To satisfy the equilibrium condition, the penalty force is applied to the master and slave nodes in opposite directions. Total energy remains conserved with this coupling method. Penalty forces are directly proportional to the penetration depth of nodes from contact surface and are defined by

$$F = -k.d \tag{6}$$

Where k is the stiffness of the spring and d is the penetration depth of the nodes.

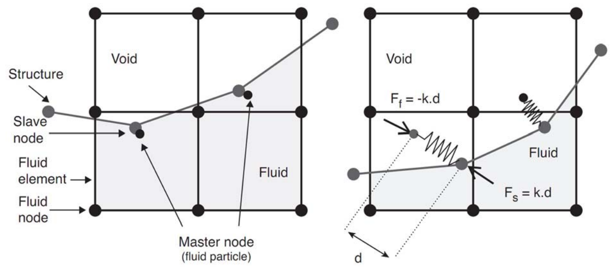


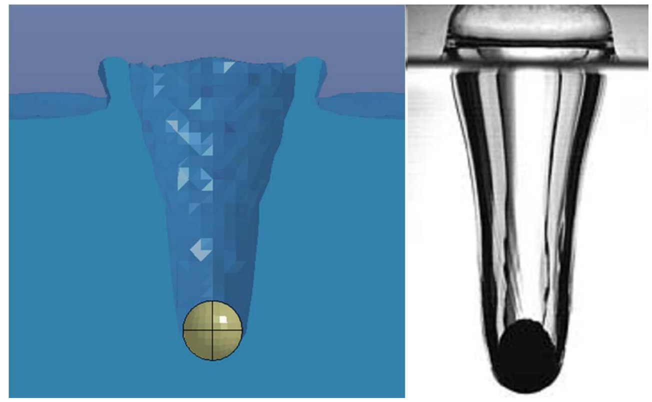

Fig. 1: Schematic of penalty method before and after coupling

The reliability and accuracy of the numerical model are validated by the experimental results of free fall decelerating sphere [9]. Experimental data of steel sphere with density of 7860 kg/m$^3$ is chosen for verification. Diameter of the sphere is 0.0254m and the water entry velocity is 2.17m/s. For numerical simulation, sphere is considered as rigid material whereas air and water are assumed as incompressible fluids. Both solid and fluids are modeled with Solid 164 3D elements which is an 8-nodes brick element. The water and air are considered as a Null material which is a constitutive material model and used with an equation of state (EOS) to give total stress of the material. The Linear Polynomial EOS is specified to the air domain while the Gruneisen EOS is used for the water field. For water and air, cut-off pressure and dynamic viscosity are defined for numerical cavitation and numerical stability respectively. Characteristics parameters of air and water are presented in Table 1. Size of the computational domain and mesh size are adjusted during numerical investigation so that further change in size has no considerable effect on the numerical results.

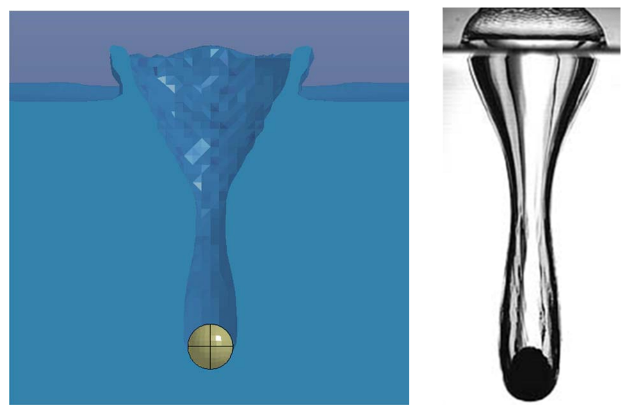
TABLE 1: CHARACTERISTICS PARAMETERS OF FLUID DOMAIN

| Fluid domain | Density, ρ (kg/m$^3$) | Cut-off pressure (Pa) | Dynamic viscosity coefficient |
|---|---|---|---|
| Water | 998 | -10.0 | 8.684e-4 |
| Air | 1.25 | -1.0 | 1.7465e-5 |

Fig. 2 compares the numerical results of present study and experiments of Aristoff et al. [9] for cavity shapes and positions at two time instances (40.9 ms and 61.9 ms) after the sphere enters into the water. The moment when the sphere touches the water surface is taken as initial time (t = 0). Results indicate that the numerical method can appropriately simulates the phenomenon of cavity evolution which includes cavity formation, cavity growth, splash and necking. It is seen that sphere transfer its kinetic energy to the fluid during the impact which in turns splashes outwards. Fig. 3 plots the time histories of vertical penetration depth of free falling sphere for the present numerical study and experimental data. Penetration depth in the numerical simulation is slightly more than experimental value in the later stage of the simulation but this deviation is very small and can be neglected. Good agreement is seen between the numerical and experimental results which indicate that the established numerical model can reliably be used for water entry impact problems.



(a)



(b)

Fig. 2: Cavity shape and position of water entering sphere a) Numerical and experimental results at 40.9ms b) Numerical and experimental results at 61.9ms

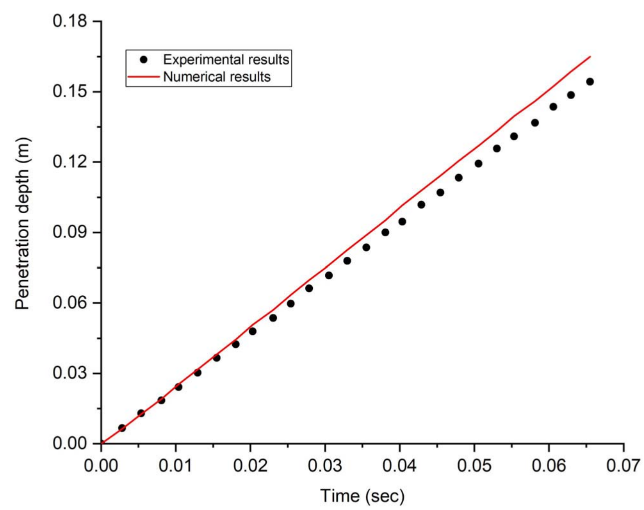



Fig. 3: Comparison of numerical results and experimental data (Aristoff et al.) for penetration depth of sphere

## III. Results and Discussion

After validation of numerical model, impact characteristics of full size AUV are investigated during water entry process using ALE formulation in LS-DYNA software. The AUV is modeled with Solid 164 3D elements and is assumed as a rigid body and the movement equation are employed to control its motion based on the rigid body kinematics. The material of the shell of vehicle is aluminum alloy grade 6061-T6. Physical properties of the vehicle are presented in Table 2. Structured finite element meshed model of the AUV is shown in the Fig. 4. In order to reduce the computational time and due to the symmetry of the system, half symmetric model is used for the simulations. Fluid boundaries are considered as non-reflecting (NR) to counter the wave reflection effects. AUV along with computational domain and boundary conditions are shown in Fig. 5. Mesh size plays an important role in the ALE simulation results for large deformation problems. Convergence study is performed to obtain the appropriate mesh size of Eulerian elements and Lagrangian elements so that simulation results are independent of mesh density. After many iterations, the total numbers of structure elements are 1600 and fluid elements are 2625000. This convergence study is not presented here.

TABLE 2: PHYSICAL PROPERTIES OF THE AUV

| Mass (Kg) | Shell Thickness (mm) | Length (mm) | Diameter (mm) |
|---|---|---|---|
| 30 | 10 | 1421 | 200 |

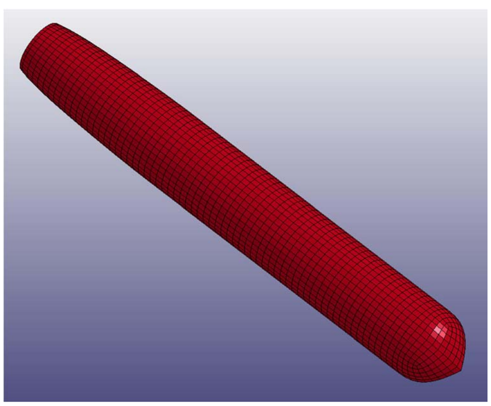


Fig. 4: Meshed model of the AUV

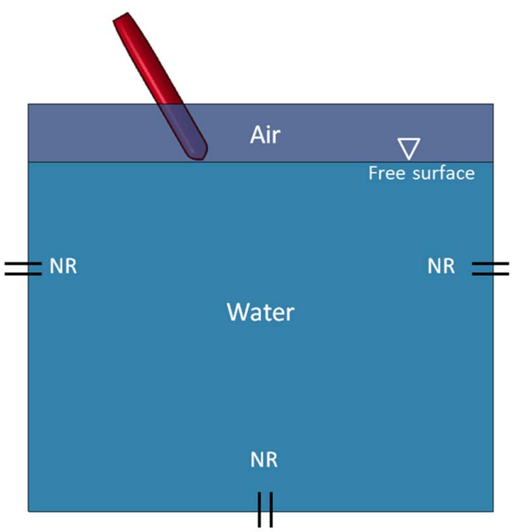
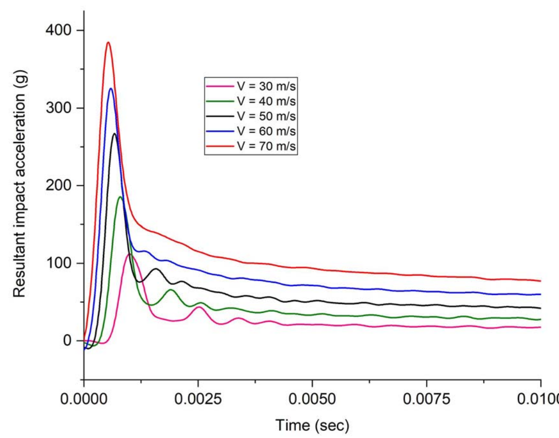


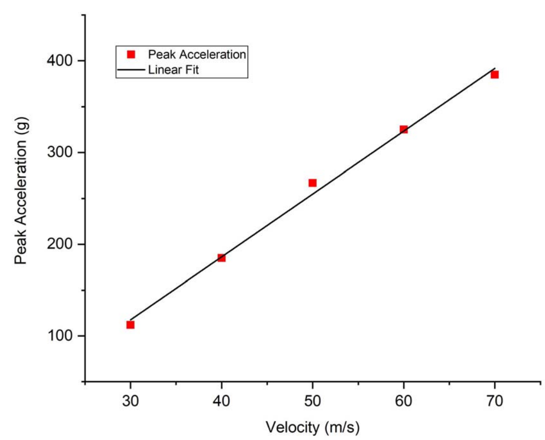

Fig. 5: Computational domain and boundary conditions

*A. Effect of water entry velocity on resultant impact acceleration*

Water entry of AUV at different impact velocities are simulated for fixed entry angle of $60^0$. The plot of resultant accelerations versus time for different entry velocities is depicted in Fig. 6a. It is seen that by increasing the water entry velocity, peak resultant acceleration increases and time to reach the peak acceleration is reduced with the increase in the impact velocity. It is also observed that acceleration curve stables more quickly for higher velocities. Fig. 6b shows a linear relationship between peak impact acceleration and entry velocity. Fig. 7 shows the comparison between present ALE method and SPH results of earlier study [14] for the peak resultant accelerations at different velocities. It is noticed that ALE algorithm gives slightly higher peak values of accelerations than SPH method. It can be due to the grid distortion at high velocities as ALE method is highly dependent on mesh size of the impact domain whereas SPH is a mesh free technique but main drawback of the SPH method over ALE is high computational cost [11]. It is evaluated from Fig. 7 that the maximum relative deviation from SPH results is 15.6% (for entry velocity of 40m/s). This shows that overall numerical results using ALE method are quantitatively well compared with those of SPH method.




(a)




(b)

Fig. 6: a) Time histories of resultant acceleration at different entry velocities with constant entry angle of $60^0$ b) Peak impact accelerations at different velocities

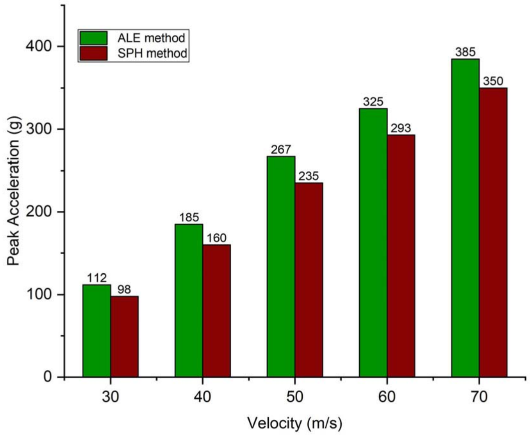
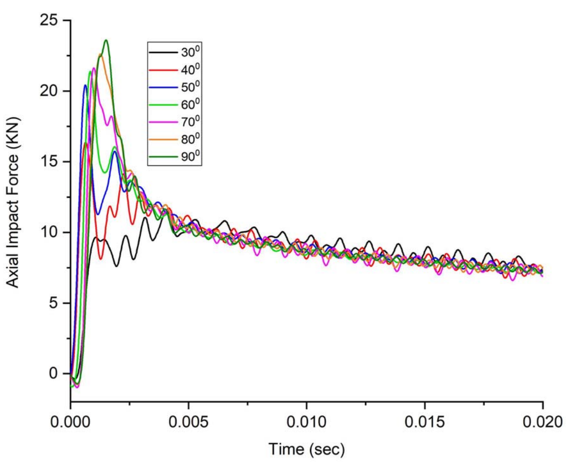


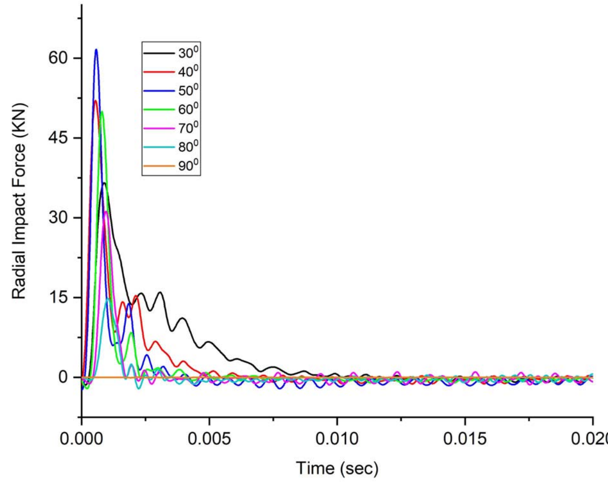

Fig. 7: Comparison between present study and SPH results for peak accelerations at different entry velocity

### B. *Effect of water entry angle on axial and radial impact force*

Fig. 8 shows the time histories of impact force curve in axial and radial direction for different water entry angles at fixed velocity of 40m/s. It is seen that impact force in axial direction increases with the increase in the entry angle and is maximum when AUV enters the water vertically. However radial impact force is maximum at water entry angle of $50^0$. To clearly understand the impact behavior of AUV at different angle, the peak axial and radial impact forces are plotted against various entry angles for fixed launch velocity (40m/s) as shown in Fig. 9. The variation of the radial load against water entry angle is different as compared to axial load. It can be observed that the radial impact force is zero or very small for vertical water entry and radial impact force increases with the decrease in water entry angle and reaches its maximum value at the entry angle of $50^0$. Afterwards, as the angle decreases, radial force also decreases. At small water entry angles $40^0$ or less, AUV exhibits ricochet behavior [19], [21] and wetting surface of AUV head decreases at initial impact. Therefore, radial impact force also decreases at small water entry angle. It is clearly seen that impact load in radial direction is greater than the axial impact load for oblique water impact. It demonstrates that the oblique water entry of AUV is more sensitive to the radial impact load.




(a)




(b)

Fig. 8: Time histories of impact force at various entry angles for a fixed launch velocity of 40m/s (a) Axial impact force (b) Radial impact force

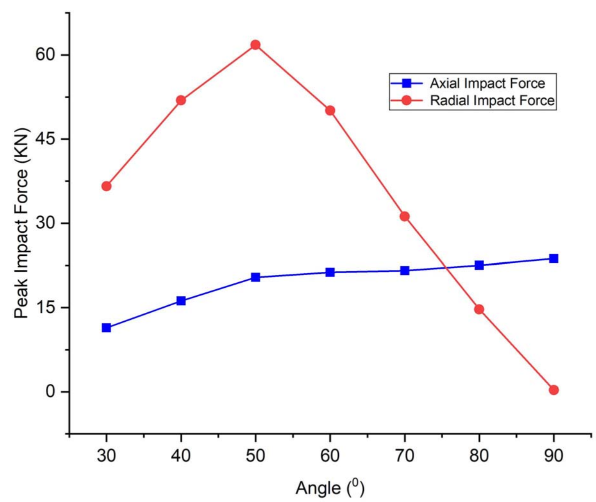



Fig. 9: Axial and radial peak impact force at various entry angles (fixed launch velocity of 40m/s**)**

## IV. CONCLUSION

In this paper, fully coupled FEM-ALE algorithm is employed to study the hydrodynamic behavior of full size AUV. Numerical method is validated through the available experimental data of decelerating sphere. Cavity evolution and displacement of sphere obtained from numerical results are well compared with experiment. Good agreement reveals the reliability and accuracy of the proposed numerical method for simulating the water entry impact problems.

Resultant impact accelerations of AUV at different velocities are calculated using ALE formulations and are compared with SPH results of previous study. Maximum relative deviation from SPH results is 15.6% but overall results are quantitatively well compared. As ALE method is computationally more efficient than SPH method, therefore numerical results further validates the effectiveness of ALE method for water entry problems at high velocities.

Numerical simulations indicate that the initial water entry velocity and angle has a great influence on the impact characteristics of the AUV. For oblique water entry of AUV, impact load in radial direction plays a significant role. This study provides reference data for the structure design of AUV. In future studies, effect of different lengths and head shapes of AUV on impact load characteristics and cavity dynamics will be explored.

## ACKNOWLEDGMENT

This work was funded by the National Natural Science Foundation of China (Grant No. 51479170, 51709229 and 11502210).

## REFERENCES


[1] T. Von Karman, "The impact on seaplane floats during landing, Technical Report 321, NACA," 1929.

[2] H. Wagner, "Phenomena associated with impacts and sliding on liquid surfaces," *J. Appl. Math. Mech.*, vol. 12, pp. 193–215, 1932.

[3] J. K. Cole, C. E. Hailey, W. T. Gutierrez, and M. T. Ferrario, "An experimental investigation of high-speed water-entry for full-size and scale-model pointed nose vehicles," *NASA STI/Recon Tech. Rep. N*, vol. p. 94, 1993.

[4] R. B. Wynn *et al.*, "Autonomous Underwater Vehicles (AUVs): Their past, present and future contributions to the advancement of marine geoscience," *Mar. Geol.*, vol. 352, pp. 451–468, 2014.

[5] D. Battistin and A. Iafrati, "Hydrodynamic loads during water energy of two-dimensional and axisymmetric bodies," *J. Fluids Struct.*, vol. 17, no. 5, pp. 643–664, 2003.

[6] M. S. Park, Y. R. Jung, and W. G. Park, "Numerical study of impact force and ricochet behavior of high speed water-entry bodies," *Comput. Fluids*, vol. 32, no. 7, pp. 939–951, 2003.

[7] V. Duclaux, F. Caillé, C. Duez, C. Ybert, L. Bocquet, and C. Clanet, "Dynamics of transient cavities," *J. Fluid Mech.*, vol. 591, pp. 1–19, 2007.

[8] T. T. Truscott, D. N. Beal, and A. H. Techet, "Shallow angle water entry of ballistic projectiles," *7th Int. Symp. Cavitation*, 2009.

[9] J. M. Aristoff, T. T. Truscott, A. H. Techet, and J. W. M. Bush, "The water entry of decelerating spheres," *Phys. Fluids*, vol. 22, no. 3, pp. 1–8, 2010.

[10] M. R. Alizadeh, A. Rabiee, and M. M. Alishahi, "Numerical and Experimental Investigation of a water entry problem with impact effects," in *13th Annual & 2nd International Fluid Dynamics Conference*, 2010, pp. 1–11.

[11] R. Challa, S. C. Yim, V. G. Idichandy, and C. P. Vendhan, "Rigid-Object Water-Entry Impact Dynamics: Finite-Element/Smoothed Particle Hydrodynamics Modeling and Experimental Validation," *J. Offshore Mech. Arct. Eng.*, vol. 136, no. 3, 2014.

[12] M. R. Erfanian, M. Anbarsooz, N. Rahimi, M. Zare, and M. Moghiman, "Numerical and experimental investigation of a three dimensional spherical-nose projectile water entry problem," *Ocean Eng.*, vol. 104, pp. 397–404, 2015.

[13] O. H. Hassoon, M. Tarfaoui, and A. El Malki, "Slamming Impact Simulation of 2D Water Entry for Rigid Structures," *VII Eur. Congr. Comput. Methods Appl. Sci. Eng.*, no. June, pp. 1582–1593, 2016.

[14] G. X. Yan, G. Pan, Y. Shi, L. M. Chao, and D. Zhang, "Experimental and numerical investigation of water impact on air-launched AUVs," *Ocean Eng.*, vol. 167, no. February, pp. 156–168, 2018.

[15] C. Chen, T. Sun, Y. Wei, and C. Wang, "Computational analysis of compressibility effects on cavity dynamics in high-speed water-entry," *Int. J. Nav. Archit. Ocean Eng.*, vol. 11, no. 1, pp. 495–509, 2019.

[16] M. S. Derakhshanian, M. Haghdel, M. M. Alishahi, and A. Haghdel, "Experimental and numerical investigation for a reliable simulation tool for oblique water entry problems," *Ocean Eng.*, vol. 160, pp. 231–243, 2018.

[17] Y. Shi, X. fu Gao, and G. Pan, "Design and load reduction performance analysis of mitigator of AUV during high speed water entry," *Ocean Eng.*, vol. 181, pp. 314–329, 2019.

[18] Y. Shi, G. Pan, G. X. Yan, S. C. Yim, and J. Jiang, "Numerical study on the cavity characteristics and impact loads of AUV water entry," *Appl. Ocean Res.*, vol. 89, no. May, pp. 44–58, 2019.

[19] J. Gao, Z. Chen, W. T. Wu, and X. Li, "Numerical investigations on the water entry of cylindrical projectiles with different initial conditions," *Appl. Sci.*, vol. 9, no. 9, 2019.

[20] M. Souli and D. J. Benson, *Arbitrary Lagrangian- Eulerian and Fluid – Structure Interaction - Numerical Simulation*. Wiley Publications, New York, 2010.

[21] G. X. Yan, G. Pan, Y. Shi, and J. T. Ma, "Numerical investigation of the skip characteristics of a disk based on the coupled FEM-SPH method," *Mod. Phys. Lett. B*, vol. 33, no. 20, 2019.