Proceedings of the Thirty-fourth (2024) International Ocean and Polar Engineering Conference **www.isope.org**
Rhodes, Greece, June 16–21, 2024

ISBN 978-1-880653-78-4; ISSN 1098-6189

Two-Way Coupled CFD-FEA Method for Dam-Break Flows Impacting on the Elastic Beam

Jiawei Xiao[1], Xiao Wen[2], Jianhua Wang[1], Decheng Wan[1]*

[1] Computational Marine Hydrodynamics Lab (CMHL), School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
[2] Marine Design and Research Institute of China, Shanghai, China

*Corresponding Author

ABSTRACT

Ships sailing in rough seas will encounter phenomena such as green water which could be simplified as dam break problem. Such phenomena, usually accompanied by tremendous hydrodynamic loads acting on the structure, may cause considerable structural deformation and even have severe challenges to the structural safety. This paper proposed a two-way coupled CFD-FEA method to study dam-break flow impacting on a stiffed elastic beam. The method is constructed based on preCICE, which is an open-source coupling library for partitioned multi-physics simulations. The flow field is solved by RANS method with OpenFOAM and the structural part is solved by FEA with Calculix. An investigation on mesh independence and the comparison of results with other methods have been conducted to validate the simulation results. The effect of the longitudinal bone reinforcement is studied in the paper. The results indicate that the developed method can effectively simulate the influence of longitudinal bone. Longitudinal bones can reduce the deformation of beams, which is consistent with the design of wave deflectors.

KEY WORDS: Two-way coupled; CFD-FEA method; dam break; elastic beam.

INTRODUCTION

In rough sea conditions, green water phenomenon could cause large slamming loads over short duration. This phenomenon can often be numerically studied through simplified problems that share similar fundamental flow characteristics, such as the dam-break problem. Greco (2007) concluded that the occurrence of dam-break type (type DB) water-on-deck scenarios in ocean engineering is more prevalent than that of plunging wave type (type PW). The slamming loads in dam-break problem will cause sever structural damage. Hence, investigating slamming loads and hydroelastic response is very important for practical engineering designs.

Many experiments and numerical simulations have investigated the interactions between dam-break flow and an obstacle, especially in cases where the structures exhibit elasticity. Liao (2015) investigate the phenomenon of free surface flow impacting on elastic structures. A series of quasi two-dimensional experiments on dam-break with an elastic beam are conducted. The main features of free surface flow impacting on elastic structures including large impacting force, structural vibration, violent free surface flow, are investigated. Yilmaz (2021) presents a newly-designed experimental setup based on the interaction of a dam-break flow with an elastic sluice gate in the context of the hydroelasticity problems. The digital video images were conducted to calculated the displacements of the elastic sluice gate and water level evolutions at various points. Hu (2023) proposed a fully-coupled computational fluid dynamics (CFD) and computational solid mechanics (CSM) model to study the hydroelastic behavior of a vertical wall in periodic waves. It is concluded that the flexible wall's wave reflection, run-up, and loading is less than the rigid one's. The decreases also occur when the waves become shorter.

The impact of a dam-break flow on an elastic beam is a typical case for studying the interaction between nonlinear fluid dynamics and structures. With the development of numerical method. Many scholars compared their numerical results with benchmark experiment measurements to validate their solvers and algorithms.

Walhorn (2005) proposes a numerical method for the analysis of fluid–structure interaction problems with free surface flows using mesh method. A consistent space–time finite element discretization for both continua and a strong coupling algorithm is applied to the highly non-linear problem for achieving high convergence of the coupled solution. The level set method is applied for capturing complex free surfaces. To take discontinuities into account modified ansatz functions are used for finite elements cut by the interface. The two-dimensional dam break example is conducted to confirm the validation of the method.

The large mesh deformation under elasticity may cause traditional mesh methods divergence. Currently, most numerical simulations focus on using meshless particle methods for FSI calculations. Meshless methods can be broadly divided into SPH and MPS methods. Ryzhakov (2010) presents a numerical method based upon the utilization of a Lagrangian description for both the fluid and the structure. The fluid part uses a linear displacement–pressure interpolation pair while the structure utilizes a standard displacement-based formulation. A series of numerical examples were applied to validate the proposed method, including dam failure examples. The results indicate that the method could deal with FSI problems involving arbitration variables in the shape of the fluid domain. Khayyer (2018) developed an enhanced fully-Lagrangian meshfree computational method for simulating incompressible fluid–elastic structure interactions. The method corresponds to a SPH (Smoothed Particle Hydrodynamics)-based coupled FSI (Fluid–Structure Interaction) solver. The dam break with an elastic gate is conducted to verify the applicability of the method.

Zheng (2020) developed an integral version of the MPS model to deal with the interactions between fluids and structures. The finite elements for structural discretization serve as ghost cells for interaction force calculation using the newly extended GCB model. The effectiveness and accuracy of the model are validated via two numerical examples, i.e., the dam break test and the water flow in a rotating gear test. Zhang (2021) proposed a partitioned MPS-FEM method which is employed for two-dimensional and three-dimensional free surface flow interacting with deformable structures. Two types of interpolation techniques, including the Shape Function Based Interpolation Technique (SFBI) and Kernel Function Based Interpolation Technique (KFBI) are proposed on the fluid-structure isomers interface. The performance of MPS-FEM coupled method is validated by a series of two-dimensional Fluid-Structure Interaction including the dam break experiment. Wu (2022) using the mixed-mode function-modified moving particle semi-implicit (MPS) method. And the numerical model based on the MPS method is set up to predict the dam-break wave impact load on an elastic beam and compared with the experimental measurements to verify the feasibility of the method.

McLoone (2022) developed a novel fluid–structure interaction (FSI) scheme, with the meshless finite volume particle method (FVPM) for fluid dynamics and the finite element (FE) solver FEBio for solid mechanics. The method proposed here ensures that particles interact with the correct neighbor particles and boundaries, regardless of the particle overlap with the thin structure, allowing for particle size that can be chosen independently of structure thickness. The new method is validated for a hydrostatic water column on an elastic beam, a dam-break with elastic gate, a dam-break with a downstream elastic wall, and a 2-D model of a heart valve leaflet.

This paper proposed a two-way coupled CFD-FEA numerical simulation technique based on grid method. The introduction of finite element calculation methods makes it possible to calculate elastic beams with attachments. The numerical results would be compared with the experiment results to verify its application. The effects of the bones is discussed.

NUMERICAL METHOD

In this section, a two-way coupled CFD-FEA method is proposed to investigate the dam break problem. The main framework of the coupling strategy will be discussed in this section. The fluid domain is solved by OpenFOAM with RANS model and VOF method. Calculix, an open-source FEA software, is employed to solve the structure part. The coupling library for partitioned multi-physics simulations, known as preCICE, is utilized to couple the fluid part and the structure part using a strong implicit way.

Fluid set up

The interFoam solver in OpenFOAM ESI v2306 is employed in the fluid part. The dynamic mesh method is adopted in the solver. OpenFOAM uses the PIMPLE algorithm - a combination of Pressure-Implicit with Splitting of Operators (PISO) algorithm and Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm - to decouple velocity and pressure.

The use of incompressible models to study the dam break problem is common as well as stable approach in the current research field. The equations of the RANS method are shown as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[v\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

When it comes to the free surface capture, the VOF method with artificial compression is used to solve the problem. The transport equation of the phase fraction is

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot \left[\left(U - U_g \right) \alpha \right] + \nabla \cdot \left[U_r \left(1 - \alpha \right) \alpha \right] = 0 \tag{3}$$

where α is the phase fraction between 0 and 1. Different values of α represents the following meanings:

$$\begin{cases} \alpha = 0 & \text{air} \\ \alpha = 1 & \text{water} \\ 0 < \alpha < 1 & \text{interface} \end{cases} \tag{4}$$

Structure set up

The structural responses of the beam are calculated using Calculix, a free software three-dimensional structural finite element program which makes use of the Abaqus input format. It is possible to use commercial pre-processors. The four-node continuum shell element (S4R) and eight-node brick element with reduced integration solid element (C3D8R) is used to discretize the wedge structure. One end of the beam is fixed to form a cantilever beam structure. The dynamic movement of the wedge is described via displacement field u and the equation is shown below:

$$M\ddot{u} + K\dot{u} + Cu = q \tag{5}$$

where M, K, C are the structure mass, stiffness and damping $n \times n$ matrix. The damping matrix used Rayleigh damping which is a linear combination of mass and stiffness of the structure. The structural problem is solved by using finite element method.

Two-way Coupled method

In this paper, the multi-physics field coupling library preCICE is used to couple the above fluid solver with the structure solver to achieve a two-way coupled solver. preCICE is an open-source massively parallel system-based coupling library for partitioned multi-physics field simulations jointly developed by the Technical University of Munich and the University of Stuttgart in Germany using C++. It is powerful enough to be used as a third-party coupling tool to couple OpenFOAM flow field calculations with other open-source FEM solvers such as Calculix. For coupling solutions, preCICE uses adapter as an interface to interpolate and exchange data directly without modifying the underlying code, just by calling the libpreCICE library in each open-source program. The coupled diagram based on preCICE is shown in Fig. 1.

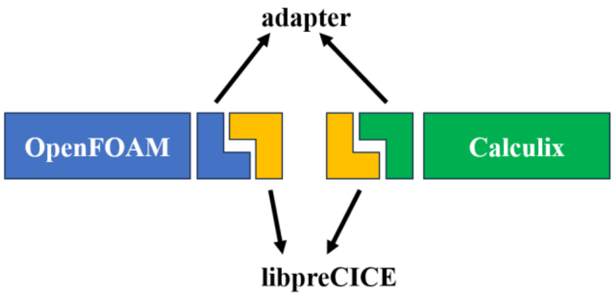

Fig. 1 Schematic diagram of preCICE data exchange.

As shown in Fig 2, the FSI solver calculates the forces on the surface of the structure in the fluid solver section and passes them to the structure solver module via preCICE. Calculix calculates the structural deformation caused by the structural forces, and then passes it back to the fluid solver module to solve the Laplace deformation equation to update the shape of the body mesh, realizing a two-way coupling. preCICE uses RBF method to map the force and displacement between OpenFOAM and Calculix. RBF short for Radial basis function mapping computes a global interpolant on one mesh, which is then evaluated at the other mesh. The global interpolant is formed by a linear combination of radially-symmetric basis functions centered on each vertex, enriched by one global linear polynomial. There are several models to choose for RBF method for the mapping problem. This paper uses rbf-compact-tps-c2 to do the mapping work. Mathematically, implicit coupling schemes lead to fixed-point equations at the coupling interface. A pure implicit coupling without acceleration corresponds to a simple fixed-point iteration, which still has the same stability issues as an explicit coupling. We need acceleration techniques to stabilize and accelerate the fixed-point iteration. PreCICE offers three different acceleration method. We choose IQN-ILS (aka. Anderson acceleration) method to accelerate the work. And the maximum iterations are 100 within one timestep.

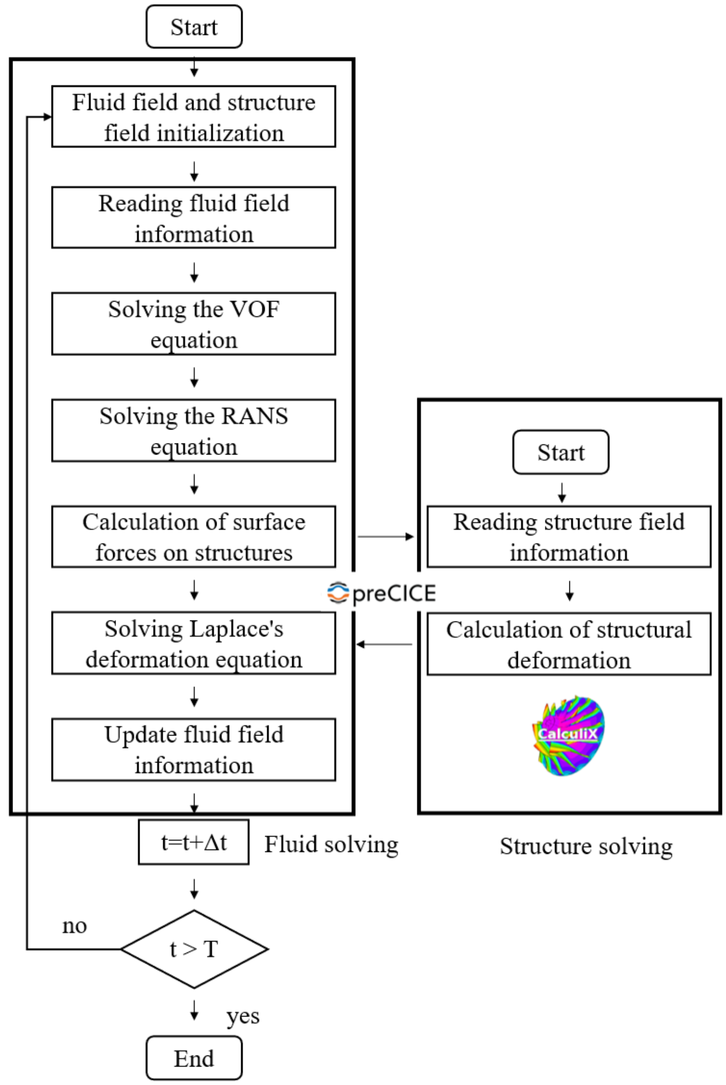

Fig. 2 FSI Solver Flowchart.

NUMERICAL SIMULATIONS

Validation of CFD-FEA Method

A benchmark simulation example conducted by Idelsohn (2008) and Yang (2016) is introduced in this section to investigate the interaction between a dam break and an elastic structure. The two-dimensional fluid domain is 0.584 m long and 0.4 m high. The volume of water's length and width is 0.292 m and 0.146 m. The beam's structure size is 0.0012m × 0.08m. It is located 0.146 m downstream from the end of the water volume. As shown in Fig. 3, x and y representing the horizontal and vertical components is defined in the coordinate system. The origin is located at the bottom left-hand corner of the domain. All boundaries are treated as walls with no-slip conditions, except for the top boundary which is considered as an atmosphere boundary with free inlet and outlet flow.

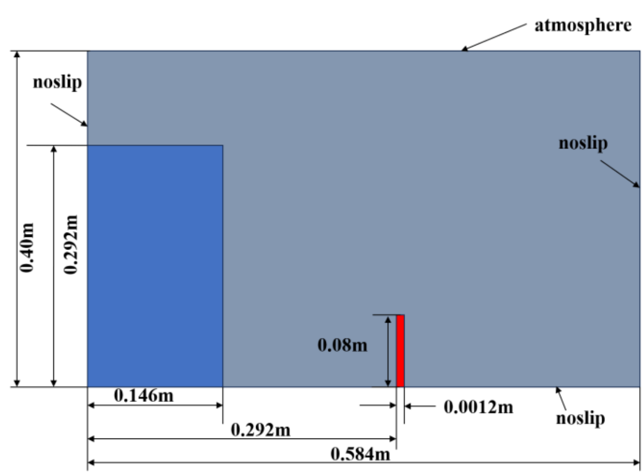

Fig. 3 Diagram of the numerical setup for the dam break test case.

The structural parameters are shown in Table 1. The Young's modulus E of structure is 1 MPa, density ρ=2.5×10^3 kg/m^2. The passion ratio is 0. A relatively small Young's modulus of the structure means that the structural beam is relatively soft, and therefore undergoes significant deformation during the impact of dam break flow. Due to the use of dynamic mesh updating in this method, the large deformation at the top of the secondary beam may lead to a decrease in mesh quality, which in turn may cause computational divergence. To ensure the convergence of calculations and smooth coupling, the calculation time step is set to 0.0002s.

Table 1. Parameter settings of fluid and structure simulations.

Parameter	Value
E	1 MPa
ρ	2.5×10^3 kg/m^2
Poisson's ratio	0
Δt	0.0002 s

To investigate the sensitivity of the fluid mesh three cell size ranging from 2mm 4mm to 6mm is adopted in this simulation. The numerical time history results of horizontal top displacements in the structure are further compared with three meshes, as shown in Fig 4. It could be concluded all three meshes maintain a favourable time history shape of the beam top displacement, and exhibit relatively small computational errors in peak value. Table 2 presents the specific numerical values of each mesh and the error relative to the fine mesh. The coarse mesh presents the maximum displacement at the top, as the beam becomes softer. The above phenomenon may be caused by a larger interpolation error between the coarse mesh force and displacement. For the sake of computational efficiency and efficiency, the medium size grid with 11160 cells was adopted. The

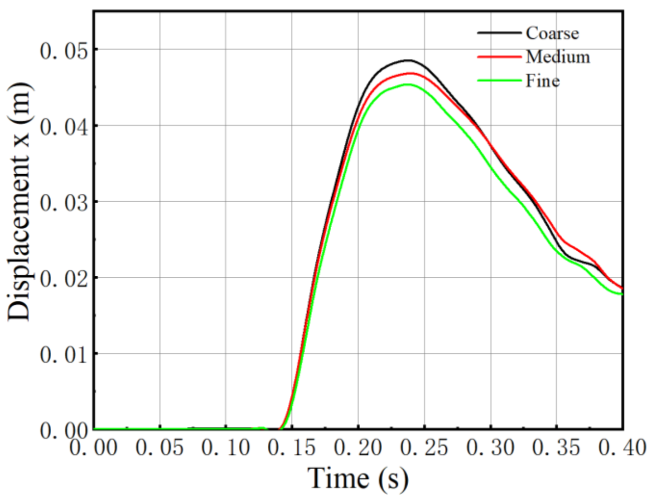

Fig. 4 Comparison of the horizontal top displacement of beam with three different meshes.

The horizontal displacements of the free end of the flexible beam are shown in Fig. 5, compared with the results obtained by particle finite element method (PFEM) (Idelsohn (2008)) and SPH-EGB method (Yang (2016)). The time history of the horizontal displacements of the CFD-FEA method is in good agreement of the literatures' results. The max top displacement of the CFD-FEA result is 0.04681 m. Compared with Idelsohn and Yang's results the relative error is lower than 1.35%.

Table 2. Structural deformation results of different cell type.

Cell type	Max top displacement	Relative error
Coarse	0.04851 m	6.9%
Medium	0.04681 m	2.3%
Fine	0.04536 m	/

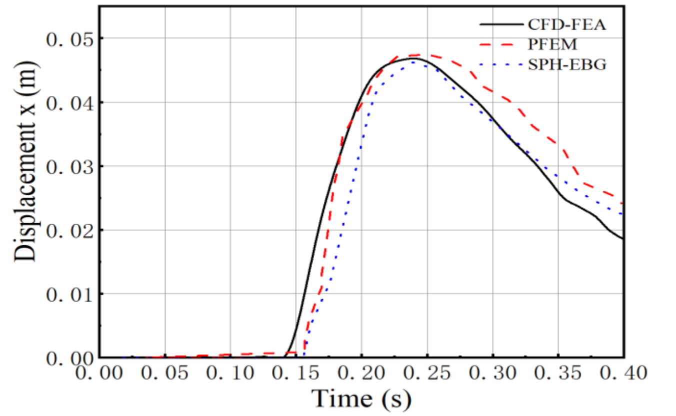

Fig. 5 Comparison of the top displacement of beam with different methods.

Table 3. Structural deformation results of different methods.

Simulation method	Max top displacement	Relative error
CFD-FEA	0.04681 m	/
PFEM	0.04745 m	1.35%
SPH-EBG	0.04622 m	1.28%

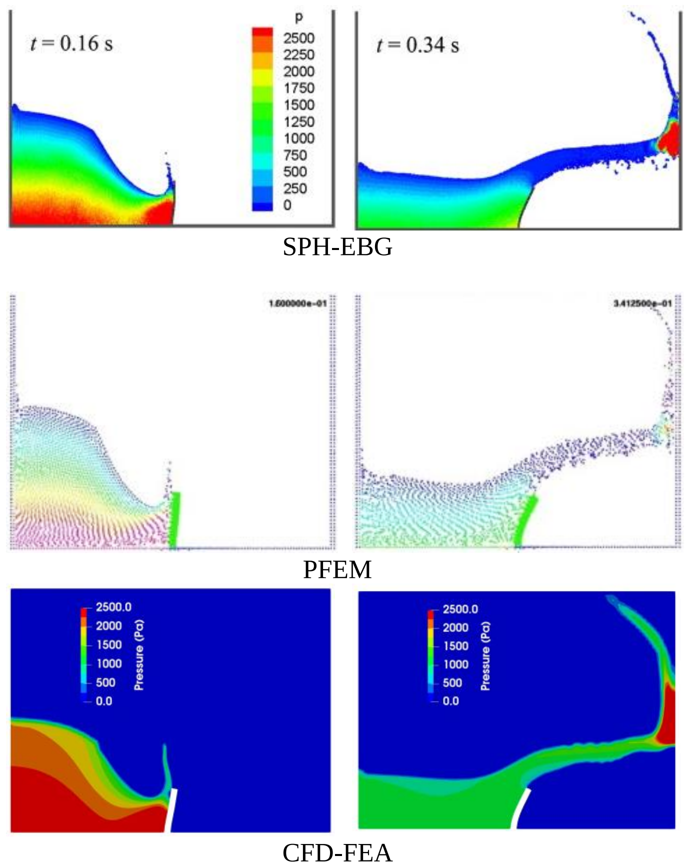

Fig. 6 Comparison of CFD-FEA simulation with PFEM and SPH–EBG results at time 0.16 s, 0.34 s. The color shows the pressure of the fluid.

Figure 6 compares the CFD-FEA method with the flow field evolution in the above literatures. At 0.16s, the water climbed along the beam. The middle of the beam first experiences a large horizontal displacement, while the free end of the beam maintains its original position due to inertia, resulting in a bow shape. At the same time, the jet splashed out along the beam, causing fragmentation. At 0.34 seconds, the jet splashed onto the right wall, generating a peak slamming pressure. At this point, all the water is loaded on the left side of the beam, causing the beam to bend. However, due to the maximum impact pressure already leaving the surface of the beam, the deformation of the beam is slightly reduced. Overall, the structural response and flow field results obtained from CFD-FEA numerical simulation are highly consistent with the literature content. The above results verify that the proposed CFD-FEA method is applicable to the study of elastic beam dam failure cases.

Evolution of the fluid domain

This part focuses on the relationship between the time history of displacement at the top of an elastic beam during dam break flow impact and the spatiotemporal distribution characteristics of the impact loads. Some pressure slices at specific displacement moments will be compared and analyzed. Fig. 7 shows the time history of horizontal displacement. Four moments is selected from the fig. At 0.134 s, the beam bent 0.00043 m in the negative x-axis direction before starting to deform. At 0.237 s, the beam reached its maximum deformation value of 0.04681 m. At 0.548 s, after deformation and recovery, the beam shifted 0.00612 m towards the negative x-axis direction. At 0.657 s, the beam was offset in the positive direction of the x-axis by 0.00475 m.

Fig. 8 shows the pressure distribution in the flow field at different times. At 0.134 s, the dam break flow had not yet climbed to half of the elastic beam, and the deformation of the elastic beam was caused by the large impact pressure. The deformation developed from the fixed end to the free end, and the position of the maximum deformation coincided with the development of the root of spray. At 0.237 s, the horizontal displacement at the free end reaches its maximum value. At this point, the jet has crossed the elastic beam and developed towards the right wall. The extreme pressure is concentrated at the root, and the dominant factor promoting horizontal deformation becomes the gravity of water loaded on the surface of the elastic beam. At 0.548 s, the water on the elastic beam decreases, and at the same time, the water on the right wall is slammed and flows back, causing the beam to rebound on the right side. At 0.659 seconds, after the backflow of water slammed, a large amount of residual water load on the left side regained control, causing the elastic beam to oscillate with large amplitude back and forth until the water finally stabilized.

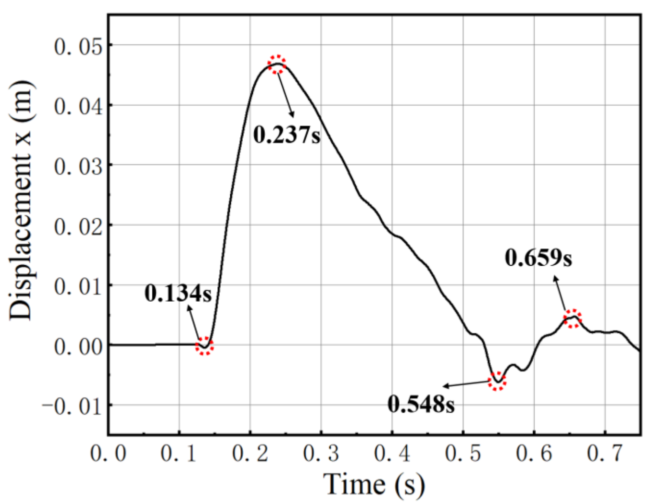

Fig. 7 Time history of horizontal displacement.

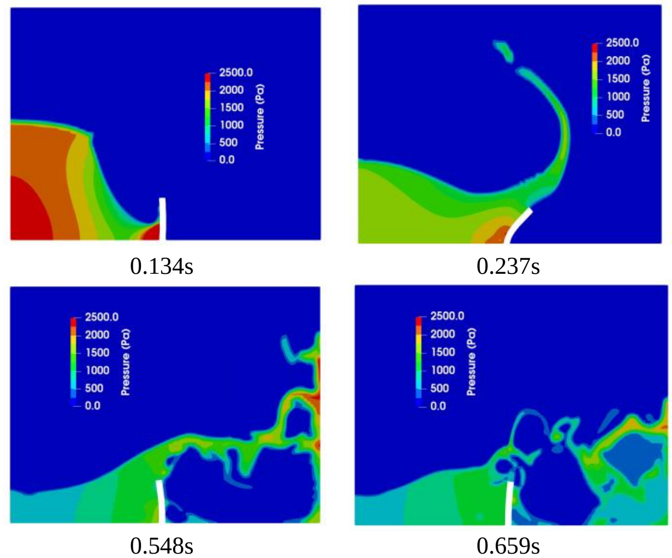

Fig. 8 Pressure snapshot of different moments.

Effects of Longitudinal bone reinforcement

In order to reduce the impact of wave slamming, ships will install wave deflectors on their decks. Longitudinal bones are usually added to strengthen the wave board. This section will investigate the effect of longitudinal bones reinforcement on the deformation of elastic beams in dam break problem. The structural modeling is shown in Fig. 9. The main unit of the beam is modeled using solid elements of C3D8R, while the longitudinal beam is modeled using shell elements of S4R. The main unit of the beam is modeled using solid elements of C3D8R, while the longitudinal beam is modeled using shell elements of S4R. The material setting of the longitudinal bone is the same as the main body of the beam, with a thickness of 0.0005 m. Calculix will elevate shell elements to solid elements based on thickness settings during the calculation process, as shown in Fig. 10. The time step is still 0.0002 s.

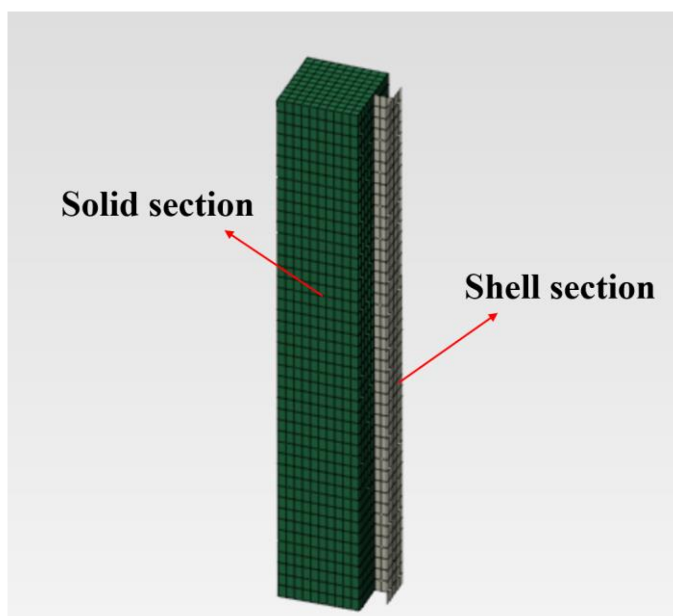

Fig. 9 Structural modeling of the stiffened beam .

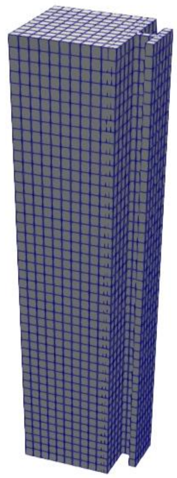

Fig. 10 Structural cells involved in calculations.

Figure 11 shows the comparison of the maximum top horizontal displacement between stiffened and unstiffened beam. The presence of longitudinal bones reduces the horizontal displacement at the top. The relative error is approximately 13.7%. From the flow field diagram in Fig. 12, the peak pressure is still concentrated at the root when it reaches its peak. Currently, the dominant factor causing the deformation of the beam is still the gravitational load of water, which is consistent with the situation without the addition of a beam.

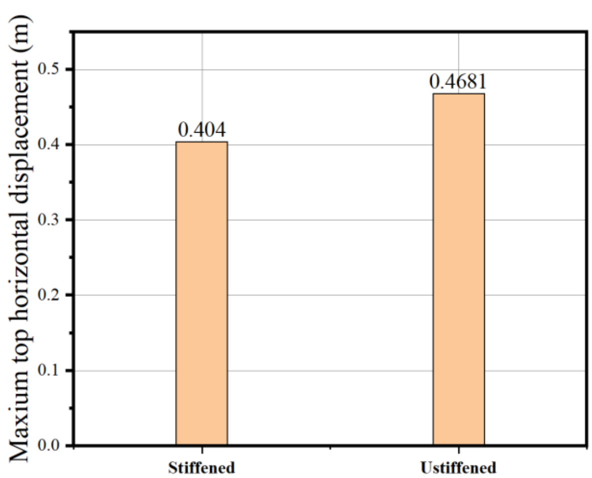

Fig. 11 Comparison of the maximum top horizontal displacement between stiffened and unstiffened beam

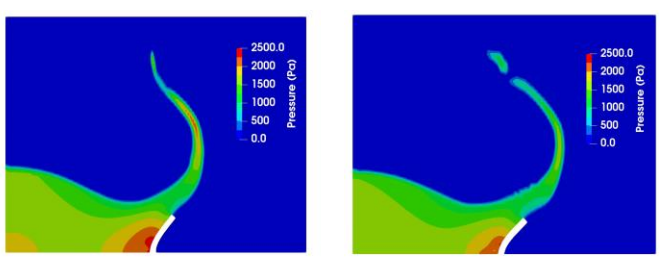

Fig. 12 Comparison of stiffened and unstiffened beam's top horizontal displacement.

CONCLUSIONS

This paper proposed a two-way coupled CFD-FEA method which combines the interFoam solver in OpenFoam v2306 with the finite element solver Calculix. The open-source multi-physics coupling library preCICE is introduced in this paper to do the communication and coupled work. The validation is carried out for a benchmark example of an elastic beam in dam break problem. The error between the top displacement obtained from numerical simulation and the literature (Idelsohn (2008) and Yang (2016)) is less than 1.35%. The flow field evolution is also in good agreement with the results in the literature. The results show the applicability of the solver for simulating structural responses of the elastic beam in dam break problem.

The evolution of dam break flow field considering elastic beams is also investigated. Flow field analysis shows that as the dam break flow develops, the elastic beam will oscillate under the impact of water flow on both sides. The developed solver to investigate the effect of longitudinal bones reinforcement on the deformation of elastic beams in dam break problem. The results show that the longitudinal reinforcement method can effectively reduce the top displacement of elastic beams under the impact of dam break flow. The analysis results further validate the current design concept of ship wave blocking beams.

Future work will focus on the material of beams and the influence of longitudinal or transverse bone arrangement methods.

ACKNOWLEDGEMENTS

This work is supported by the National Natural Science Foundation of China (52131102), to which the authors are most grateful.

REFERENCES

Greco M, Colicchio G and Faltinsen O M, 2007. Shipping of water on a two-dimensional structure Part 2 J. Fluid Mech. 581 71-99

Hu Z, Huang L and Li Y, 2023. Fully-coupled hydroelastic modeling of a deformable wall in waves Coastal Engineering 179

Idelsohn S R., Marti J., Limache A., et al., 2008. Unified lagrangian formulation for elastic solids and incompressible fluids: application to fluid–structure interaction problems via the pfem. Comput Methods Appl. Mech. Eng., 197 (19): 1762-1776.

Khayyer A, Gotoh H, Falahaty H and Shimizu Y, 2018. An enhanced ISPH–SPH coupled method for simulation of incompressible fluid–elastic structure interactions Comput. Phys. Commun. 232: 139-64.

Liao K, Hu C and Sueyoshi M, 2015. Free surface flow impacting on an elastic structure: Experiment versus numerical simulation Appl. Ocean Res. 50: 192-208.

Mcloone M and Quinlan N J, 2022. Coupling of the meshless finite volume particle method and the finite element method for fluid–structure interaction with thin elastic structures Eur. J. Mech. B-Fluid. 92: 17-31.

Ryzhakov P B, Rossi R, Idelsohn S R and Onate E, 2010. A monolithic Lagrangian approach for fluid–structure interaction problems Comput. Mech. 46: 883-99.

Walhorn E, Kölke A, Hübner B and Dinkler D, 2005. Fluid–structure coupling within a monolithic model involving free surface flows Comput. Struct. 83: 2100-11.

Wu J, Zhang G, Jiang Y and Yang X, 2022. Numerical Simulations on the Flooding into a Damaged Cabin with a Flexible Bulkhead Based on the Mixed-Mode Function-Modified MPS Method J. Mar. Sci. Eng. 10.

Yang X, Liu M, Peng S, et al, 2016. Numerical modeling of dam-break flow impacting on flexible structures using an improved SPH–EBG method, Coastal Engineering, 108: 56-64.

Yilmaz A, Kocaman S and Demirci M, 2021. Numerical modeling of the dam-break wave impact on elastic sluice gate: A new benchmark case for hydroelasticity problems Ocean Eng. 231.
Zhang G, Zha R and Wan, D C, 2022. MPS–FEM coupled method for 3D dam-break flows with elastic gate structures, European Journal of Mechanics / B Fluids, 94: 171-189.
Zheng Z, Duan G, Mitsume N, Chen S and Yoshimura S, 2020. An explicit MPS/FEM coupling algorithm for three-dimensional fluid-structure interaction analysis Eng. Anal. Bound. Elem. 121 192-206.