**Numerical Simulation of the water-exit process of the missile based on Moving Particle Semi-Implicit method**

Qiaorui Wu\(^1,2,*,\)  Lei Wang\(^1,\) Yonghe Xie\(^1,2,\)  Zhenhuang Du,  Huimin Yu

\(^1\)School of Naval Architecture and Mechanical-electrical Engineering, University of Zhejiang Ocean, Zhoushan 316022, China  
\(^2\)Key Laboratory of Offshore Engineering Technology of Zhejiang Province, Zhoushan 316022, China

*Corresponding author e-mail: qiaoruidhd@163.com

Abstract. The phenomenon of a water mound can show up in the process of water-exit of the missile [1], and the a water mound will break and splash in a very short time, which will exert an extra force on the missile and therefore will affects the movement of the missile as well as the gesture when the missile comes out the water. Therefore, investigation into the water-exit process of the missile is of great significance to the launch of the underwater missiles. The traditional grid methods always fail to simulate the whole the water-exit process because of the large deformation of the mesh. The paper adopts a meshless method, called Moving Particle Semi-Implicit method (MPS), to study the water-exit process of the missile. Based on verifying it by the dam break simulation, MPS is applied to simulate the water mound during the water-exit process, which is aiming at providing some reference for the relevant theoretical research.

Keywords: Moving particle semi-implicit method; water-exit; dam break

1. Introduction

Moving particle semi-implicit method (MPS) is relatively new meshless method based on Lagrange particles [2], which replaces the mesh in the traditional numerical methods, applies some numerical operators according to the relationship of the neighboring particles to take the place of traditional difference terms. As such, the numerical diffusion resulted from solving the convection term can be avoided [3]. Meanwhile, the particle moves freely and separately under the external force and internal force from its neighboring particles, so MPS has absolute advantages in solving problems with large deformation, even with the splashing of free water surface.

During the water-exit process of the underwater launching missile, when the missile approaches the water free surface, some water will be carried and raised because of the high speed of the missile, which is called the phenomenon of a water mound. The water mound will grow larger and higher with the movement of the missile, and then because of the gravity force itself the bump will break away from the surface of the missile and splash. For such problems, the traditional mesh methods always lose efficacy because of the large deformation of the mesh.

Subsequently, the paper focuses on the application, and develops the relative numerical model of the MPS. The classical dam break model [4-5] is adopted to verify the program. On this base, the water-exit
The process of the missile is simulated numerically, and the mechanism and variation law of the water mound during this process is analyzed and summarized, which is aiming at providing some useful reference for the relative engineering practice.

2. Basis of MPS

For impressible viscous flows, the continuity equation (Law of mass conservation) and Navier-Stokes equation (Law of momentum conservation) are given by:

\[
\begin{align*}
\frac{d\rho}{dt} &= \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \\
\frac{d\mathbf{v}}{dt} &= f - \frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{v}
\end{align*}
\]

(1)

Where \( \rho \) is fluid density, \( \mathbf{v} \) is fluid velocity, \( f \) is the external force, \( p \) is pressure and \( \nu \) is the kinematic viscosity coefficient.

3. Numerical Model of Original MPS

The kernel function proposed by Koshizuka and Oka in 1996 is employed, which is used most widely in MPS [6-7]:

\[
w(r) = \begin{cases} 
\frac{r_e - 1}{r} & (r < r_e) \\
0 & (r > r_e)
\end{cases}
\]

(2)

Where \( r \) is the distance between particle, \( r_e \) is the effective radius of particle, the value of which directly affects the number of neighboring particles.

The number density of particle can be obtained by:

\[
\langle n \rangle_i = \sum_j w(|r_i - r_j|)
\]

(3)

The Gradient model is described by:

\[
\langle \nabla \phi \rangle_i = \frac{d}{n} \sum_{j \neq i} \left( \phi_j - \phi_i \right) \frac{(r_j - r_i)}{|r_j - r_i|^2} W(|r_j - r_i|)
\]

(4)

Where \( d \) is the dimension, \( \phi_i \) is the minimum value of the neighboring particles of particle \( i \), \( \phi_i = \min(\phi_j) \), which is effective to avoid instability resulting from minus pressure gradient.

The Laplacian model of \( \phi \) at point \( i \) is expressed as

\[
\langle \nabla^2 \phi \rangle_i = \frac{2d}{n^2} \lambda \sum_{j \neq i} \left( \phi_j - \phi_i \right) W(|r_j - r_i|)
\]

(5)

Where \( \lambda \) can be calculated by the following equation [8]:

\[
\lambda = \frac{\int w(r)r^2 dV}{\int w(r)dV} \approx \frac{\sum_{j \neq i} |W(|r_j - r_i|)| |r_j - r_i|^2}{\sum_{j \neq i} W(|r_j - r_i|)}
\]

(6)
The effective radius \( r_e = 2.1L_0 \) (\( L_0 \) is the initial distance of particles) is applied for particle number density, Gradient model and Laplacian model.

For the free surface boundary condition, the free surface particles can be judged by [9-10]:

\[
\langle n_i \rangle^* < b n^0
\]

where \( \beta \) is a parameter below 1.0 and \( \beta = 0.97 \) is selected. The dynamic free surface condition is satisfied by taking the atmospheric pressure \( p = p_{atm} = 0 \) for the free surface particles.

In order to avoid misjudging the fluid particles near the boundary as free surface particles, the solid boundary is discretized into three layers of particles. Moreover, a repulsive force [11] is employed to prevent fluid particles from penetrating the solid boundary.

4. Verification of the numerical model

Dam break [12] is a classic example to simulate the fluid flow with free surface. Therefore, the semi-implicit method for moving particles is verified by the dam break simulation. The model adopted is shown in Fig.1, and it is discretized into 1122 particles, of which 648 are fluid particles, and 474 are solid wall particles and fluid boundary particles, as shown in Fig.2.

**Figure 1.** Physical figure of dam break model.  
**Figure 2.** Numerical model of dam break.

Fig.3 shows the collapse shapes of water column at 0.1s, 0.15s, 0.20s, 0.25s, 0.30 and 0.35s respectively. As can be seen from Fig. 3, the liquid column flows to the right end at the bottom of the container after collapse.
4

(d) $t=0.20\,\text{s}$  
(e) $t=0.30\,\text{s}$  
(f) $t=0.35\,\text{s}$

Figure 3. Phenomenon of dam break simulation.

Fig. 4 shows the time-varying curve of the distance between the leading edge of the liquid column and the solid wall. The MPS results are compared with experimental results and data from other numerical methods [13-14]. It can be seen from Fig. 4 that the results agree well.

Fig. 4. Comparison of dam break results.

5. simulation of water-exit of the missile

In this paper, the model of water-exit adopted is shown in Fig.5, the corresponding numerical model is shown in Fig.6, which contains 3712 particles, including 2860 fluid particles, 286 fluid boundary particles and 566 solid wall particles. The missile moves upward at a constant velocity of 1m/s from 2.2L below the free surface. Other calculation parameters are consistent with the dam break model.

Fig. 5. Physical model of water-exit.  
Fig. 6. Numerical model of water-exit of the missile.

As can be seen from Fig.7, due to the shape of the missile and the viscosity of water, when it starts to move, bubble areas are gradually formed near the shoulder and tail of the missile [15-16]. Meanwhile, a water mound appears at the free surface. Then, with the movement of the missile, the water mound becomes more and more obvious.
Figure 7. Phenomenon of water-exit of the missile.
The bubble area near the tail grows first and then begins to decrease. At \( t=0.072s \), water impacts the tail of the missile, where jet impact occurs. At the same time, the bubble area around the shoulder becomes larger and moves backward, and it begins to break around \( t=0.078s \). In addition, due to the reflection of the boundary of the solid wall, one layer of floating particles appears on the free surface within \( t=0.060s \) to \( t=0.120s \). Finally, the missile gets out of the water with severe water splash. In this process, the bubble area near the shoulder of the missile gradually disappears. At the same time, the water mound will fall and break in a very short time and the liquid surface shows very large deformations.

6. Conclusion
In this paper, MPS is verified by investigating the dam break model that it is capable of dealing with fluid flows with large defamations. Based on these, MPS is applied to simulate the whole water-exit process of the missile, including the initial movement, steady movement in the water as well as piercing the water surface. This has great significance to investigate the forming, growing and breaking mechanism of the bubble areas near the shoulder and the tail of the missile. Moreover, the results in the paper can provide some reference to the forming and breaking mechanism of the water bump during the water-exit process of the missile.

Due to the limitation of time and equipment, the number of particles used in the simulation is relatively small, which leads to the unsatisfactory effect of the simulation of the broken water mound. Therefore, the next research objective is to optimize the algorithm, expand the two-dimensional to three-dimensional, to better simulate the water-exit process and then reveal the mechanism of the water mound phenomenon.

Acknowledgements
This work is supported by National Natural Science Foundation of China (51679217), Zhejiang provincial Natural Science Foundation of China (LQ17E090003), the Doctoral Scientific Research Foundation of Zhejiang Ocean University (21045012715), Zhejiang Province Key R&D Projects in 2018 (2018C03043), and Zhoushan Science and Technology Project (2016C41008).

References
[1] Sun S L. High frequency fluid-solid coupling problem with free surface effect[M]. Harbin Engineering University: PhD thesis. 2010: 2-3.
[2] Gong S J, Pan X J, Yao Z Q. Research progress and application of semi-implicit method for moving particles[J]. Journal of Jiangsu University of Science and Technology, 2009, 23(6):484-489.
[3] Liu G. R, Liu M. B, Xu Han Xu, et al (Translation). Hunan: Hunan University Press, 2005:1-26.
[4] Xu G, Duan W Y. Simulation of MPS algorithm for free surface flow simulation[J]. Journal of Harbin Engineering University, 2008, 29(6):539-542.
[5] Rong S S, Chen B. A Semi-implicit Simulation Method for Moving Particles Simulating Free Surface Problems[J]. Journal of Xi'an Jiaotong University, 2010, 44(11): 109-112.
[6] Pan X J, Zhang H X, Sun X Y, Study on the applicability of kernel function in mobile ion semi-implicit method[J]. Marine Engineering,2010,28(4):70-76.
[7] Pan X J, Zhang H X. Study on Pressure Oscillation in Slosh Simulation of Moving Ion Semi-Implicit Method[J]. Hydrodynamics Research and Progress(A),2008,23(4):453-463.
[8] Masayuki T, Takayuki M. Stabilization and smoothing of pressure in MPS method by Quasi-Compressibility[J]. Journal of Computational Physics, 2010, 229:4279-4290.
[9] Lee C K, Noguchi H, Koshizuka S. Fluid-shell structure interaction analysis by coupled particle and finite element method[J]. Computers and Structures.2007, 85: 688-697.
[10] Pan X J, Zhang H X, Sun X Y. A Free Surface Recognition Method Adapted to Mobile Ion Semi-Implicit Method[J]. Journal of Shanghai Jiaotong University, 2010, 44(1): 134-138.
[11] MONAGHAN J J. Simulation free surface flow with SPH[J]. Journal of Computational Physics, 1994, 110: 399-406.
[12] Ashtiani B A, Farhadi L. A Stable moving-particle semi-implicit method for free surface flows[J]. Fluid Dynamics Research, 2006, 38: 241-256.
[13] Lee B H, Park J C, Kim M H, et al. Numerical simulation of impact loads using a particle method[J]. Ocean Engineering, 2010, 37: 164-173.
[14] Zhang Y X, Wan D C. Application of MPS Method in 3D Dam Breaking Problem[J]. Marine Engineering Hydrodynamics, 2011, 41(2): 140-154.
[15] Zhai J X, Wei Y J, Wang C, et al. Numerical Simulation of Hydrodynamic Characteristics of Submarine Launched Missile in Vertical Launching Process[J]. Acta Armamentari SINICA, 2010, 31(10):1303-1309.
[16] Wei H P, Guo F M, Quan X B. Study on surface cavitation characteristics of submarine-launched missiles[J]. Journal of Astrona.