Development of a Lagrangian Meshless Flow Solver based on the Moving Particle Semi-implicit (MPS) Method

K C Ng\(^1\), Y L Ng\(^1\) and M Z Yusoff\(^1\)

\(^1\)Centre for Advance Computational Engineering (CACE), Universiti Tenaga Nasional, Malaysia.  
\(^2\)Email: nkching@uniten.edu.my

Abstract. This paper discusses on the development of a meshless flow solver based on the Lagrangian particle method. The differential operators are discretised by using the particle interaction models proposed in the numerical framework of the Moving Particle Semi-implicit (MPS) technique. The MPS method is attractive from the viewpoint of no mesh is required and fluid is purely represented by points (the virtue of meshless algorithm). Some flow applications attempted by the current meshless solver will be shown and results are compared with the published experimental data.

1. Introduction

There are various numerical techniques available nowadays to simulate fluid flow. The mesh-based technique, particularly the Finite Volume Method (FVM), has gained immense popularity within the CFD community since the past few decades. In fact, most of the CFD commercial software has adopted FVM as the core numerical method in developing the fluid flow and heat transfer algorithms.

Although FVM has proven its versatility in many engineering problems, one should take note on the limitations of most of the mesh-based methods, for instances: inaccuracy due to inferior mesh quality, numerical diffusion due to convection discretization, smearing of highly rigorous free surface, interpolation error due to local remeshing when moving-boundary problem is encountered, and many others. The development of the Lagrangian particle method aims to circumvent the above issues. Fluids are treated as particles that are freely to move in space (Lagrangian); therefore, the diffusion due to convection discretization can be reasonably eliminated. Being meshless in nature, the information of mesh structure (such as normal area vector, volume of polygons, etc.) is not needed and the free surface can be traced easily and naturally; enabling one to capture the evolution (both space and time) of free surface effectively. MPS method has been used mainly to capture the rapidly changing flow interfaces such as those encountered in the dam-break problem [1], flow boiling [2] general two-phase flow [3] and even heat transfer problem such as Rayleigh-Benard convection [4].

The current paper intends to provide a brief introduction on the MPS method. In the following sections, the numerical methods of MPS will be highlighted, followed by its applications that have been attempted by the authors recently.
2. Numerical Method
The full incompressible Navier-Stokes equation is considered in the current work:

\[
\frac{1}{\rho} \frac{D\rho}{Dt} + \nabla \cdot \mathbf{u} = 0, \tag{1}
\]

\[
\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho} \nabla P + \nu \nabla^2 \mathbf{u} + g. \tag{2}
\]

Here, \( \nu \) is the kinematic viscosity, \( \rho \) is the fluid density and \( g \) is the gravitational acceleration. The above equations are solved in the Lagrangian manner. The differential operators (gradient and Laplacian operators) appeared in the governing differential equations are treated in the following manner:

\[
\nabla \phi_i = \frac{d}{n^0} \sum_{j \neq i} \frac{\phi_j - \phi_{\text{min}}}{|\mathbf{r}_j - \mathbf{r}_i|^2} \mathbf{r}_j \cdot \mathbf{w}(|\mathbf{r}_j - \mathbf{r}_i|) \tag{3}
\]

\[
\nabla^2 \phi_i = \frac{2d}{\lambda_i n^0} \sum_{j \neq i} (\phi_j - \phi_i) \mathbf{w}(|\mathbf{r}_j - \mathbf{r}_i|) \tag{4}
\]

where

\[
\lambda_i = \frac{\sum_{j \neq i} |\mathbf{r}_j - \mathbf{r}_i|^2 \mathbf{w}(|\mathbf{r}_j - \mathbf{r}_i|)}{\sum_{j \neq i} \mathbf{w}(|\mathbf{r}_j - \mathbf{r}_i|)}. \tag{5}
\]

\( \mathbf{r}_i \) is the position vector of particle \( i \). \( \phi_{\text{min}} \) is the minimum \( \phi \) value of neighbouring particles within the radius of influence \( r_e \). \( d \) is the number of dimensions (\( d=2 \) for 2D and \( d=3 \) for 3D). \( n^0 \) and \( \mathbf{w}(|\mathbf{r}_j - \mathbf{r}_i|) \) are the initial particle number density and kernel function respectively. The initial particle number of a particle \( i \) can be computed as the summation of kernel function \( \mathbf{w}(|\mathbf{r}_j - \mathbf{r}_i|) \) of its neighboring particles within its radius of influence.

The above equations are solved for each time step, following the fractional-step method for coupling between the continuity and momentum equations.

3. Result and Discussion

3.1. Sloshing in a Container
The MPS code is used to simulate the sloshing flow inside an oscillating tank. The boundary condition as well as the experimental data reporting on the time variation of the moving free-surface is available from [5].

During the simulation, the time step size and initial particle spacing is prescribed as 0.0005 sec. and 3.33mm, respectively. Figure 1 shows the comparison of instantaneous water level at \( t=1.652s \) between those tested and simulated by using the current method. As seen, the predicted data agree
considerably well with those measured, showing that the current numerical procedure is suitable to predict liquid sloshing.

![Figure 1. Comparison of free surfaces at t=1.652 sec.](image1)

3.2. Oscillating Impeller in a Cylindrical Tank

The test case considered here is the process of stirring of fluid in a cylindrical vessel agitated by a plate impeller with an infinitely long height. It has been anticipated by [6] that the reciprocating motion of the impeller may enhance the mixing performance. The sinusoidal motion of the impeller can be obtained from [6].

Figure 2 shows the instantaneous streakline at t~10s obtained from MPS and experimental observation [6]. The agreement is promising.

![Figure 2. Streaklines obtained by MPS (left) and experimental method (right) at t~10s. Experimental data is obtained from [6.](image2)](image2)
3.3. Dam Break

Dam-break flow occurs when a column of water is subjected to collapse due to gravity after the sudden removal of a barrier. Figure 3 shows the predicted locations of water particles at $t=1.0\text{s}$. Indeed, by assuming laminar flow in the current work, rigorous movement of flow particles can be observed, and the proper description of eddy viscosity is effective, at least to certain extent, in suppressing these unphysical scatterings.

![Figure 3. Location of water particles at $t=1.0\text{s}$. The experimental data is obtained from [7].](image)

4. Conclusions

The basic algorithm of the meshless code based on Moving Particle Semi-implicit (MPS) method has been developed and verified against the experimental data. Future work would be the incorporation of more complex flow models in order to study other interesting flow physics such as granular flow, multiphase flow (solid-liquid, gas-liquid), etc. Meanwhile, we will be looking into some numerical aspects of MPS in order to enhance its numerical stability.

Acknowledgement

The first author is highly indebted to Professor Komoda and Professor Koshizuka for giving their written permissions in using the experimental data appearing in Figure 2 and Figure 3, respectively, in the current paper.

References

[1] Kondo M and Koshizuka S 2011 Improvement of Stability in Moving Particle Semi-Implicit Method International Journal for Numerical Methods in Fluids 65 638-654.
[2] Koshizuka S, Ikeda H and Oka Y 1999 Numerical Analysis of Fragmentation Mechanisms in Vapor Explosions Nuclear Engineering and Design 189 423-433.
[3] Shirakawa N, Yamamoto Y, Horie H and Tsunoyama S 2003 Analysis of Subcooled Boiling with the Two-Fluid Particle Interaction Method Journal of Nuclear Science and Technology 40 (3) 125-135.
[4] Zhang S, Morita K, Fukuda K and Shirakawa N 2006 An Improved MPS Method for Numerical Simulations of Convective Heat Transfer Problems International Journal for Numerical Methods in Fluids 51 31-47.
[5] Andrillon Y and Alessandrini B 2004 A 2D+T VOF Fully Coupled Formulation for the Calculation of Breaking Free-Surface Flow Journal of Marine Science and Technology 8 159-168.
[6] Komoda Y, Senda S, Takeda H, Hirata Y and Suzuki H 2012 2D Fluid Deformation Induced by a Rotational Reciprocating Plate Impeller in a Cylindrical Vessel 14th European Conference on Mixing, Warszawa 10-13 September 2012 211-216.
[7] Koshizuka S, Tamako H and Oka Y 1995 Particle Method for Incompressible Viscous Flow with Fluid Fragmentation Computational Fluid Dynamics Journal 4 29-46.