Pertanika J. Sci. & Technol. 24 (2): 465 - 473 (2016)

SCIENCE & TECHNOLOGY

Journal homepage: http://www.pertanika.upm.edu.my/

On the Extension of Moving Particle Method for Flow Computation in Irregular Flow Domain

Ng, K. C.[1]*, Sheu, T. W. H.[2] and Hwang, Y. H.[3]

[1]*Center of Fluid Dynamics (CFD), Department of Mechanical Engineering, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia*

[2]*Center for Advanced Studies in Theoretical Sciences (CASTS), National Taiwan University, Taipei, Taiwan*

[3]*Department of Marine Engineering, National Kaohsiung Marine University, Kaohsiung 805, Taiwan*

ABSTRACT

A new numerical scheme based on the particle method, namely the Moving Particle Pressure Mesh (MPPM) method, has been previously developed by the authors to address the limitation of the conventional Moving Particle Semi-implicit (MPS) method in simulating incompressible flow. In this paper, we shall investigate on a more practical way to extend our MPPM method to handle complex geometry, i.e. by employing an embedded unstructured mesh system to cope with an arbitrarily-complex flow domain. No-slip boundary condition is modelled via placing a series of fixed particles at the wall boundaries, negating the use of ghost particles which are difficult to generate. In order to verify our numerical procedure, the vortex-shedding process behind a cylinder is computed and it is found the numerical result is agreeable with the reference solution.

Keywords: Moving Particle Semi-implicit (MPS), Moving Particle Pressure Mesh (MPPM), CFD, incompressible flow, unstructured mesh, semi-Lagrangian

INTRODUCTION

The original Moving Particle Semi-implicit (MPS) method was developed by Koshizuka and Oka (1996) to simulate incompressible flow. One of the main advantages of MPS is that the convective term (i.e. the non-linear term in Navier-Stokes equation) is discretised in the Lagrangian manner which can circumvent the issue of convective instability as commonly found in the Eulerian scheme (e.g. finite-volume method, see Ng et al., 2006a, 2006b, 2007, 2008; Ng 2009). To date, the MPS method has been extensively used in various engineering applications, such as breaking

Article history:
Received: 17 February 2016
Accepted: 22 April 2016

E-mail addresses:
nkching@uniten.edu.my (Ng, K. C.),
twhsheu@ntu.edu.tw (Sheu, T. W. H.),
yhhwang@mail.nkmu.edu.tw (Hwang, Y. H.)
*Corresponding Author

ISSN: 0128-7680

wave (Koshizuka et al. 1998; Lee et al. 2011), two-phase flow (Chen et al. 2010), mixing problem (Ng et al. 2013; Ng & Ng 2013) and many others.

While realising the fact that the computational domain for practical fluid flow problem is geometrically complex, we have notified that the original MPS scheme poses a few problems while treating complex flow boundary. First, the information related to proper reinforcement of a uniform initial particle number density (commonly denoted as n_0) throughout the flow field involving an arbitrarily complex flow boundary, is rather limited. Second, the current treatment of no-slip wall boundary condition (e.g. mirror/ghost particles) on complex flow boundary is cumbersome (Akomoto, 2013).

We have previously worked on a possible way to partially address the problems mentioned above. In our previous work, namely the Moving Particle Pressure Mesh (MPPM) method (Hwang 2011; Ng et al. 2014), we have made use of an embedded pressure mesh to resolve the continuity equation, thereby negating the use of particle number density. However, owing to the Cartesian nature of the embedded pressure mesh in MPPM, we are able to consider only the simple flow problem (i.e. rectangular flow domain). In spite of this, we realise that this problem can be resolved by employing a more robust pressure mesh system (e.g. unstructured mesh) to cope with the complex domain.

In the current work, we shall report on our recent numerical results obtained based on the unstructured pressure mesh. A typical flow past a cylindrical bluff body will be considered and the numerical results will be validated with the well-established numerical results published in the renowned *Journal of Fluid Mechanics* (Zovatto & Pedrizzetti, 2001).

MATHEMATICAL MODEL AND NUMERICAL METHODS

Governing Equation

The 2D incompressible flow equations, which consist of the continuity and momentum equations, are solved in the current work:

$$\nabla \bullet \vec{u} = 0 \qquad [1]$$

$$\rho \frac{D\vec{u}}{Dt} = -\nabla P + \mu \nabla^2 (\vec{u}) \, . \qquad [2]$$

Numerical Methods

Eqns. (1&2) are solved by using the fractional-step method. The velocity of a fluid particle i at time level n+1 can be computed via:

$$\vec{u}_i^{n+1} = \vec{u}_i^n + \frac{\Delta t}{\rho_i}\left(\mu_i \nabla^2 (\vec{u})_i^n - \nabla P_i^{n+1}\right), \qquad [3]$$

and the new position of the particle i can be updated as:

$$\vec{r}_i^{n+1} = \vec{r}_i^n + \Delta t \vec{u}_i^{n+1} \, . \qquad [4]$$

Eqn. [4] is mainly adopted to address the convective effect of fluid flow in particle-based method.

As seen from Eqn. [3], the viscous term and pressure gradient term are treated in an explicit and implicit manner respectively. In order to ensure the divergence-free velocity condition as enforced in Equation [1], the pressure term must be corrected accordingly via the Pressure Poisson Equation (PPE):

$$\frac{\Delta t}{\rho_M}\nabla^2 P_M^{n+1} = \nabla \bullet \vec{u}_M^{\,*} . \qquad [5]$$

Here, the superscript * denotes the intermediate time level, a state where the fluid flow velocity is obtained by considering only the viscous term (i.e. neglecting the effect of pressure gradient term in Eqn. [3]). The subscript M indicates mesh level, indicating that Eqn. [5] is solved on the mesh level (i.e. pressure mesh) instead, which is in contrast with Eqn. [3]. In the current work, Eqn. [5] is discretised on an unstructured mesh system. Once the correct pressure field is obtained at the mesh level, the particle's velocity is corrected corresponding to the pressure gradient. Meanwhile, the face velocities at the mesh level can be correspondingly corrected to ensure divergence-free condition. It is worth to mention here that in MPPM, the pressure and velocity are stored in a staggered mesh manner, following the philosophy of the classical Marker and Cell (MAC) method. The numerical details of MPPM method can be found in our previous work (Hwang, 2011).

RESULT

The method has been applied to solve the incompressible flow pass through a non-rectangular body. A cylindrical bluff body of diameter D = 0.2m is considered in this case, whereby it is contained in a flow channel of length L = 10.5m. The distance apart between the top and bottom walls (i.e. width of the channel W) is 1.0m. The inflow x-velocity profile is assumed to be parabolic (which yields an averaged inflow velocity U = 1.0m/s). For this particular flow case where the origin is placed at the lower left corner of the flow domain (refer to Figure 1), the inflow x-velocity profile is:

$$u(y) = 6(y - y^2), \qquad [6]$$

while the inflow y-velocity is zero. Mass balance is ensured via specifying the same velocity profile at the outlet.

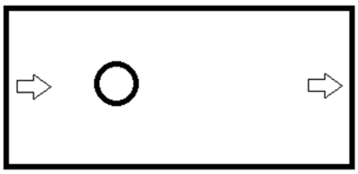


Figure 1. Schematic diagram of the problem involving flow passing through a cylinder (of diameter D = 0.2m) placed in the middle of the flow channel of length L = 10.5m and W = 1.0m. The distance between the inlet and the cylinder centre is 3.0m

The flow Reynolds number is defined as $Re = U.W/\upsilon$, where υ is the kinematic viscosity of the fluid. An unstructured quadrilateral mesh consisting of 16704 elements is used to discretise the flow domain, while the flow particles (necessary to solve the momentum equation, see Eqn. (3)) are placed at the centroid of the quad-mesh initially. In order to model the no-slip boundary condition at the wall, fluid particles with fixed velocity (i.e. u=v=0.0) are placed along the wall segment, without the necessity to employ the ghost/mirror particles. The implementation of boundary condition without introducing ghost particles have been discussed in Ng et al. (2015). Figure 2 shows the initial layout for the unstructured pressure mesh and fluid particles of this flow problem.

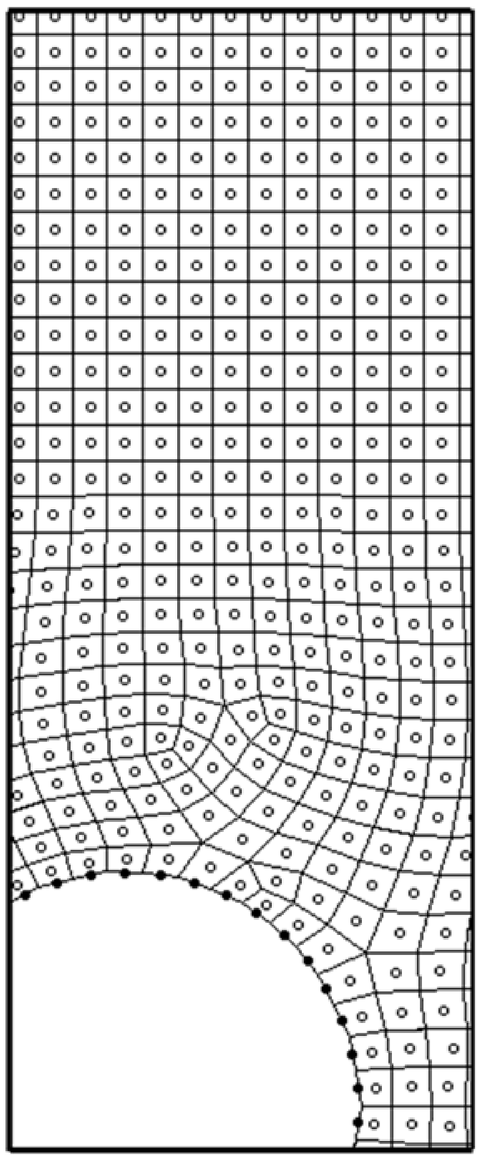


Figure 2. The pressure mesh in the vicinity of the cylinder body. Fixed wall particles (solid black circles) are placed along the cylinder body without the necessity of employing ghost/mirror particles

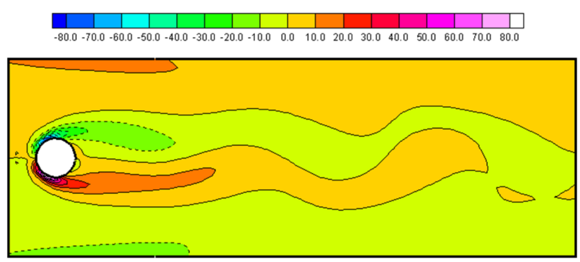
In order to initiate the flow instability, a skewed inflow x-velocity profile is introduced within a short time frame $0 < t < 0.1$ s, defined as:

$$u(y) = \begin{matrix} 1.5 & y < 0.5 \\ 0.5 & y \geq 0.5 \end{matrix}. \quad [7]$$

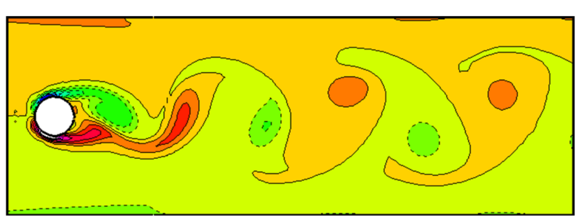
This skewed inflow velocity profile is crucial to hasten the occurrence of vortex shedding behind the cylinder, if any. For this unsteady simulation, we have adopted a Courant number of 0.5.

Two flow cases (Re = 240 and Re = 1000) have been considered in the current work. For the case of Re = 240, a relatively mild vortex shedding process behind the cylinder can be observed from the vorticity plot reported in Figure 3(a). The y-velocity plot is shown in Figure 4, indicating that the shedding is intensified while the Reynolds number is increased to 1000. Figure 5 reports the spatial distribution of static pressure at t=80s. It is interesting to note the smoothness of the static pressure field computed from our particle method, which is hardly attainable by employing the conventional MPS methods.

In order to validate the shedding frequencies of the flow cases, the y-velocity values at point located at 1.0m downstream from the cylinder centroid are numerically measured. Results are shown in Figure 6. It is found that the period (T) is 0.844s and 0.679s for Re = 240 and Re = 1000, respectively. Our numerical results are very close to that reported in Zovatto and Pedrizzetti (2001), i.e. 0.85s (for Re = 240) and 0.67s (for Re = 1000). As observed from Figure 6, as Re increases, the shedding frequency increases (decrease of T).

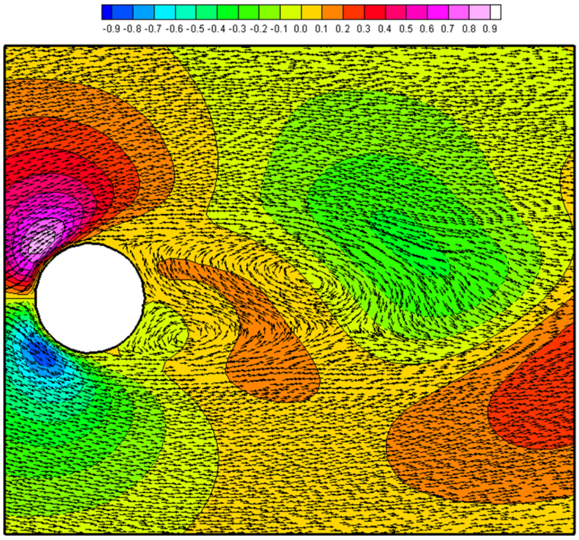


(a)

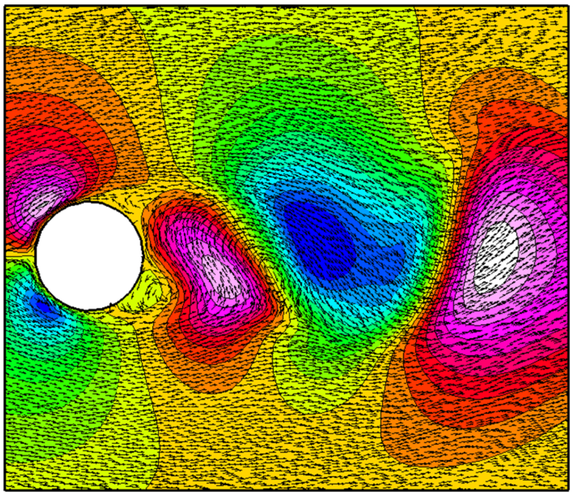


(b)

Figure 3. Vorticity [s^{-1}] plot for (a) Re = 240 and (b) Re = 1000. Negative vorticity values are marked with dashed lines. t = 80s

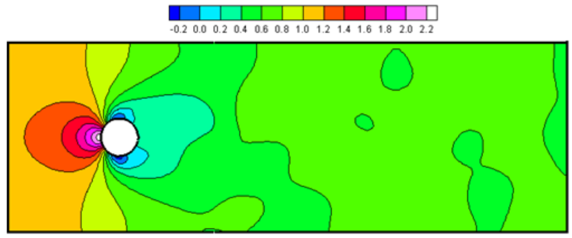


(a)

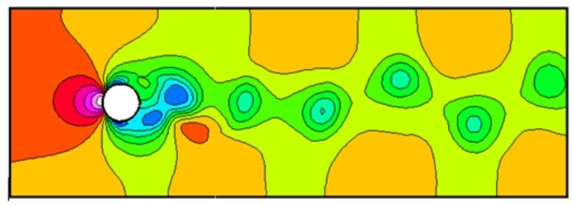


(b)

Figure 4. y-velocity [m.s^{-1}] plot for (a) *Re* = 240 and (b) *Re* = 1000. *t* = 80s. The velocity vectors are positioned at the centroid of the flow particles



(a)



(b)

Figure 5. Static pressure [Pa] plot for (a) *Re* = 240 and (b) *Re* = 1000. *t* = 80s

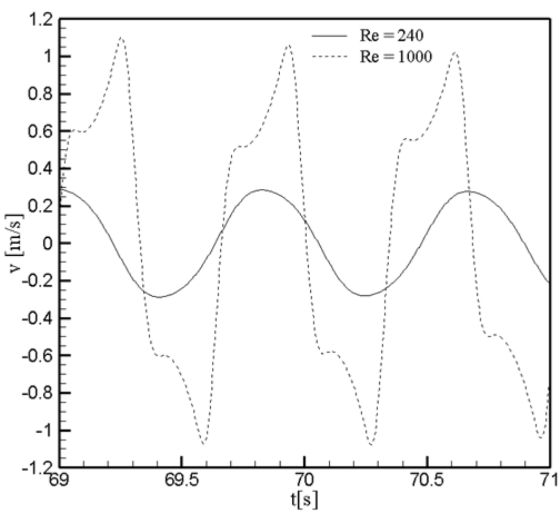


Figure 6. Time evolution of *y*-velocity at point located 1.0 m downstream from the cylinder centroid at different Reynolds number

CONCLUSION

In the current work, we have implemented an unstructured pressure mesh system as a solution to extend our previous MPPM particle method to solve an incompressible fluid flow problem in a non-rectangular flow domain. Although data structure is relatively complex compared with that of our original MPPM solver which employed a Cartesian mesh to resolve the continuity equation, we find that our current method is robust and it has a great potential to resolve industrial flow problems involving very complex flow domains. Combined with the inherited benefits of moving particles used to resolve the convective term, we foresee that this method is accurate in computing flow at high *Re* (convection-dominated flow).

A few interesting problems have been observed during the course of our study. Generally, the total execution time of most of the particle methods (including our current one) is relatively longer compared with the conventional Eulerian scheme, mainly due to the CPU time spent to process the scattered moving particles. Also, the order of accuracy of the discretisation method of the diffusion (or viscous) term is highly dependent on the instantaneous locations of the fluid particles, which may diminish the overall accuracy of the flow solver. As such, we are currently considering ways to shorten the overall execution time of the flow solver by parallelisation (via OPENMP and GPU). A more proper way to discretise the viscous term must be sought as well.

ACKNOWLEDGEMENTS

We acknowledge with gratitude the financial support provided by the Ministry of Education, Malaysia (Fundamental Research Grant Scheme Ref. No: FRGS/2/2013/TK01/UNITEN/02/1) and the Ministry of Science, Technology and Innovation (MOSTI), Malaysia (Project No: 06-02-03-SF0258) .

REFERENCES

Akimoto, H. (2013). Numerical simulation of the flow around a planning body by MPS method. *Ocean Engineering*, *64*, 72-79.

Chen, R., Tian, W., Su, G. H., Qiu, S., Ishiwatari, Y., & Oka, Y. (2010). Numerical investigation on bubble dynamics during flow boiling using moving particle semi-implicit method. *Nuclear Engineering and Design*, *240*(11), 3830-3840.

Hwang, Y. H. (2011). A moving particle method with embedded pressure mesh (MPPM) for incompressible flow calculations. *Numerical Heat Transfer, Part B*, *60*(5), 370-398.

Koshizuka, S., & Oka, Y. (1996). Moving-Particle Semi-implicit method for fragmentation of incompressible fluid. *Nuclear Science and Engineering*, *123*(3), 421-434.

Koshizuka, S., Nobe, A., & Oka, Y. (1998). Numerical analysis of breaking waves using the moving particle semi-implicit method. *International Journal for Numerical Methods in Fluids*, *26*(7), 751-769.

Lee, B. H., Park, J. C., Kim, M. H., & Hwang, S. C. (2011). Step-by-step improvement of MPS method in simulating violent free-surface motions and impact loads. *Computer Methods in Applied Mechanics and Engineering, 200*(9), 1113-1125.

Ng, K. C., Yusoff, M. Z., & Ng, E. Y. K. (2006a). Multigrid solution of Euler equations using high-resolution NVD differencing scheme for unstructured meshes. *Progress in Computational Fluid Dynamics, an International Journal, 6*(7), 389-401.

Ng, K. C., Yusoff, M. Z., & Ng, E. Y. K. (2006b). Parametric study of an improved GAMMA differencing scheme based on normalized-variable formulation for low-speed flow with artificial compressibility technique. *Numerical Heat Transfer, Part B: Fundamentals, 50*(6), 561-584.

Ng, K. C., Yusoff, M. Z., & Ng, E. Y. K. (2007). Higher-order bounded differencing schemes for compressible and incompressible flows. *International Journal for Numerical Methods in Fluid*s, *53*(1), 57-80.

Ng, K. C., Ng, E. Y. K., Yusoff, M. Z., & Lim, T. K. (2008). Applications of high-resolution schemes based on normalized variable formulation for 3D indoor airflow simulations. *International Journal for Numerical Methods in Engineering*, *73*(7), 948-981.

Ng, K. C. (2009). A collocated finite volume embedding method for simulation of flow past stationary and moving body. *Computers and Fluids, 38*(2), 347-357.

Ng, K. C., & Ng, E. Y. K. (2013). Laminar mixing performances of baffling, shaft eccentricity and unsteady mixing in a cylindrical vessel. *Chemical Engineering Science*, *104*, 960-974.

Ng, K. C., Ng, E. Y. K., & Lam, W. H. (2013). Lagrangian simulation of steady and unsteady laminar mixing by plate impeller in a cylindrical vessel. *Industrial and Engineering Chemistry Research*, *52*(29), 10004-10014.

Ng, K. C., Hwang, Y. H., & Sheu, T. W. H. (2014). On the accuracy assessment of Laplacian models in MPS. *Computer Physics Communications*, *185*(10), 2412-2426, 2014.

Ng, K. C., Hwang, Y. H., Sheu, T. W. H., & Yu, C. H. (2015). Moving Particle Level-Set (MPLS) method for incompressible multiphase flow computation. *Computer Physics Communications, 196*, 317-334.

Zovatto, L., & Pedrizzetti, G. (2001). Flow about a circular cylinder between parallel walls. *Journal of Fluid Mechanics*, *440*, 1-25.