Boundary Conditions for Simulating Karman Vortices Using the MPS Method

Kazuya Shibata^{1*}, Seiichi Koshizuka¹, Kohei Murotani¹,
Mikio Sakai², Issei Masaie³

¹Department of Systems Innovation, School of Eng., The University of Tokyo

²Department of Nuclear Engineering and Management, School of Eng., The University of Tokyo

³Prometech Software Inc.

*shibata.kazuya@sys.t.u-tokyo.ac.jp

Received: November 12, 2014; Accepted: April 21, 2015; Published: November 6, 2015

Abstract. New boundary conditions to simulate the Karman vortices were developed for the Moving Particle Semi-implicit (MPS) method. These boundary conditions enable us to analyze the wake region of a cylinder without an unnatural cavity observed in past analyses. Karman vortices that were present behind a circular cylinder and a square cylinder in flows were successfully simulated in two dimensions using the MPS method with the developed boundary conditions. The simulated drag coefficients and Strouhal numbers were close to those of the finite difference method and experiments.

Keywords: Moving particle semi-implicit (MPS) method, Smoothed particle hydrodynamics (SPH), Outflow boundary, Curl model, Drag coefficient, Strouhal number

1. Introduction

Lagrangian approaches have been applied to various complex flows that involve large deformation, fragmentation, and coalescence of fluids. Smoothed particle hydrodynamics (SPH) [1] and the MPS method [2] are representative Lagrangian methods. Recently, there has been significant progress in this field. For example, Kajtar and Monaghan [3,4] simulated the motion of three linked ellipses through a viscous fluid in two dimensions using SPH. Khayyer et al. [5] proposed a corrected incompressible SPH for accurately tracking the water surface in breaking waves. They also performed enhanced wave impact simulations using the improved incompressible SPH method [6]. Antuono et al. [7,8] simulated free-surface flows using SPH schemes with numerical diffusive terms. They showed that the diffusive terms allow the reduction of the

high-frequency numerical acoustic noise and smoothing of the pressure field. Cleary [9] extended SPH to fluid phenomena in low pressure die casting, in which he simulated the solidification and freezing of an injected metal. Idelsohn et al. [10] simulated multifluid flows using the particle finite element method. They demonstrated the mixing of fluids with different densities.

The MPS method used in this study was developed by Koshizuka and Oka [2]. The main difference between the MPS method and the SPH is the approximation models for partial differential operators. Kondo and Koshizuka [11] and Hibi and Yabushita [12] proposed algorithms for suppressing the pressure oscillations in the MPS method. Tanaka and Masunaga [13] introduced quasi compressibility for stabilization and smoothening of the pressure field in the MPS method. Tanaka et al. [14] developed a multiresolution model for the MPS method. Khayyer and Gotoh [15-17] and Lee et al. [18] also proposed modified MPS methods. Suzuki et al. [19] developed the Hamiltonian moving particle semi-implicit method, and showed its excellent conservation for the mechanical energy.

The MPS method has been used in various fields. A sloshing problem was simulated by Hashimoto et al. [20]. The motion of a floating vessel in a tsunami was simulated by Masuda et al. [21]. The nonlinear motions of a two-dimensional floating body were simulated by Sueyoshi et al. [22]. Shipping water on a deck was simulated by Shibata and Koshizuka [23] and Shibata et al. [24]. Tsukamoto et al. [25] also used the MPS method to simulate the effects of an elastically linked body on sloshing.

Our study develops a boundary condition for simulating the Karman vortices using the MPS method. The Karman vortices have not been previously simulated using the MPS method because boundary conditions have not yet been developed for the phenomena. The fluid behavior and fluid force created by the Karman vortices are important for designing marine structures, chemical plants and other industrial products. Hence, the vortices should be estimated with sufficient accuracy. Although numerous studies involving Karman vortices have been carried out using both experiments [26-31] and computational fluid dynamics approaches based on grid methods [32-41], only a few studies [42-45] have used particle methods. In particular, there are few papers about Karman vortices simulation by the MPS method.

In previous studies [46] using the MPS method, unnatural cavities were observed at wake regions. An example is shown in Fig. 1. Existing outflow and free-surface boundary conditions were used in the simulation. As seen in the figure, two unnatural cavity regions occurred behind the fixed cylinder in a flow. Because of the cavities, Karman vortices were not simulated using the MPS method. In this study, we develop a set of boundary conditions which avoids the unnatural cavities and makes it possible to simulate Karman vortices. For the verification and validation, we compared our simulation results with those of the finite difference method and experiments obtained by other researchers.

The MPS method with the developed boundary condition will allow the simulation of the

Karman vortices without numerical diffusion because the convection term is not discretized. Moreover, it is believed that by using this method, the vortex behavior and fluid mixing can be easily understood by tracing the motion of particles.

Figure 1: Example of unnatural cavity at the wake region
(Left boundary: inflow, Right boundary: outflow, $Re = 100$)

2. Moving particle semi-implicit method^[2]

2.1. Governing equations

The governing equations for the MPS method are as follows:

$$\frac{D\rho}{Dt} = 0 \quad (1)$$

$$\rho \frac{D\vec{u}}{Dt} = -\nabla P + \mu \nabla^2 \vec{u} + \rho \vec{g} \quad (2)$$

Equations (1) and (2) are the conservation equations of the mass and momentum for the incompressible flows. Numerical diffusion due to the convection term does not occur because a fully Lagrangian description is employed and the convection term is not discretized. the semi-implicit algorithm of the MPS method[2] was employed.

2.2. Particle approximations

In the MPS method, the governing equations are discretized by replacing the differential operators with the following gradient and Laplacian models:

$$\langle \nabla \phi \rangle_i = \frac{d}{n^0} \sum_{j \neq i} \left[\frac{\phi_j - \hat{\phi}_i}{|\vec{r}_j - \vec{r}_i|^2} (\vec{r}_j - \vec{r}_i) w(|\vec{r}_j - \vec{r}_i|) \right] \quad (3)$$

$$\langle \nabla^2 \phi \rangle_i = \frac{2d}{n^0 \lambda} \sum_{j \neq i} (\phi_j - \phi_i) w(|\vec{r}_j - \vec{r}_i|) \quad (4)$$

where w is a weight function given as

$$w(r) = \begin{cases} \left(\frac{r_e}{r} \right) - 1 & (r < r_e) \\ 0 & (r \geq r_e) \end{cases} \quad (5)$$

We applied the radii, r_e , shown in Table 1 in this study.

Table 1: Radii of interaction domain used in this study

r_e for the particle number density	$2.1l_0$
r_e for the gradient model	$2.1l_0$
r_e for the Laplacian model	$3.1l_0$

l_0 : the distance between adjacent particles in the initial configuration

The particle number density is calculated with the above weight function, as follows:

$$n_i = \sum_{j \neq i} w(|\vec{r}_j - \vec{r}_i|) \quad (6)$$

The particle number density has two interpretations: one is the value proportional to the fluid density and the other is the normalization factor of the weighted average. The particle number density should be constant to satisfy the mass conservation equation (1). The parameter n^0 denotes the constant particle number density. The restraint condition for the particle number density is implicitly included in the pressure Poisson equation. The parameter λ in (4) is a constant given by

$$\lambda = \frac{\sum_{j \neq i} |\vec{r}_j - \vec{r}_i|^2 w(|\vec{r}_j - \vec{r}_i|)}{\sum_{j \neq i} w(|\vec{r}_j - \vec{r}_i|)} \quad (7)$$

This parameter corrects variance increase of the Laplacian model so that the variance agrees with that of the analytical solution. The standard models of the differential operator ((3) and (4)) were used in this study without any correction. This is because unnatural cavities behind a blunt object do not occur by using the boundary condition to be explained later and any special treatment is not necessary for the models of differential operators.

3. Boundary conditions for simulating Karman vortices

3.1. Outline

As mentioned in the introduction, unnatural cavities in a wake region prevent us from simulating Karman vortices. To avoid the unnatural cavities and simulate the Karman vortices, a new outflow and Dirichlet boundary conditions for pressure were devised. In this study, Karman vortices in the channel shown in Fig. 2 were simulated without free surfaces. In this simulation, we need to deal with two free-slip wall boundaries, an inflow boundary and a fixed cylinder boundary in addition to the above new boundary conditions. The all boundary conditions are explained in the following sections.

Figure 2: Schematic diagrams of the simulation domain

3.2. Outflow boundary

One of the reasons for the unnatural cavities is that outflow boundaries change the quantity of fluid in the entire simulation domain. The existing boundary condition discharges fluid particles at velocity of their own velocity without any special treatment. Therefore, the amount of fluid in the simulation domain decreases with time. As a result, unnatural cavities are generated. Permeable and nonreflecting boundary conditions for explicit SPH were developed by Lastiwka et al. [42]. However, it is difficult to directly adopt their boundary condition in the MPS method because of the use of the implicit algorithm for pressure calculation of the MPS method. In this study, a new outflow boundary condition was developed for the MPS method. The improved point is that the velocity on the outflow boundary is corrected to conserve the total quantity of fluid in a channel.

3.2.1 Outflow boundary

In the standard MPS simulations based on a fractional step scheme, the temporal velocity of each fluid particle is calculated as follows:

$$\vec{u}_i^* = \vec{u}_i^k + \left(\nu \langle \nabla^2 \vec{u} \rangle_i^k + \vec{g} \right) \Delta t, \quad (8)$$

where k is a time step number, ν is the kinematic viscosity coefficient of fluid, and $\vec{g}$ is the acceleration of gravity. To neglect the gravity effect, $\vec{g}$ was set to a null vector in this study. The total quantity of fluid in a channel must be conserved to avoid unnatural cavities and satisfy the incompressible condition. The temporal velocities of the fluid particles in the fixed velocity region on the outflow boundary were then corrected to keep the fluid amount in the simulation domain, as follows:

$$\vec{u}_i^* \cdot \vec{n} := \left(\vec{u}_i^{*upstream} \cdot \vec{n} - \Delta u \right), \quad (9)$$

$$\Delta u = \frac{(N^0 - N^*) l_0^3}{A} \frac{1}{\alpha \Delta t}. \quad (10)$$

The length of the fixed velocity region, $L_{velocity}$, was $2.1l_0$ in this study. The vector $\vec{u}_i^*$ is the temporary velocity of the i -th particle calculated by the time integration of the Navier-Stokes equation (2) (with the exception of the pressure gradient term). Each fluid particle is moved at

the velocity $\vec{u}_i^*$ to the temporal position $\vec{r}_i^*$ before the positions is corrected by the pressure gradient term. As mentioned above, the temporal velocity was corrected instead of $\vec{u}_i^{k+1}$ in this study because $\vec{r}_i^*$ should be corrected by the pressure gradient term in the Navier-Stokes equation (2) keeping the incompressible state of fluid. The vector $\vec{n}$ in (9) is the direction of the main stream at the outflow plane. The parameter N^0 is the initial number of fluid particles, while N^* is the current number of fluid particles in the entire domain. The parameter A is the cross-sectional area of the outflow boundary. The parameter α is a relaxation coefficient, and α was 1.0 in this study. The parameter Δu is the correction term for conserving the amount of fluid in the simulation domain. The vector $\vec{u}_i^{*upstream}$ is the average velocity of an upstream particle k_2 in the neighborhood of the i -th particle, as shown in Fig. 3. The upstream particle k_2 is found as follows:

$$l_{ij} = (\vec{r}_j - \vec{r}_i) \cdot (-\vec{n}), \quad (11)$$

$$k_1 = j \mid \max\{l_{ij}\} \mid \vec{r}_j - \vec{r}_i \mid \leq r_{e,search}, \quad (12)$$

$$l_{k_1j} = (\vec{r}_j - \vec{r}_{k_1}) \cdot (-\vec{n}), \quad (13)$$

$$k_2 = j \mid \max\{l_{k_1j}\} \mid \vec{r}_j - \vec{r}_{k_1} \mid \leq r_{e,search}, \quad (14)$$

where $\vec{r}_i$ and $\vec{r}_j$ are the position vectors of the particles i and j , respectively. The particle j is a neighboring particle located within a distance $r_{e,search}$ from the particle i or k_1 . In this study, $r_{e,search}$ was $3.1 l_0$; the radius $3.1 l_0$ was determined so that the radius becomes large enough to search upstream particles whose velocities are not fixed. Therefore, $r_{e,search}$ needs to be larger than $L_{velocity}$. The upstream particle k_1 was determined using (11) and (12). The same procedure was carried out using (13) and (14), and the upstream particle k_2 was determined. Carrying out the same procedure twice enables us to use a small $r_{e,search}$ and reduces the simulation cost for searching neighboring particles. Compared to the simulation cost for pressure, the cost for searching the upstream particles is small because the cost for searching upstream particle is proportional to the first power of the total particles while that for pressure is proportional to the 1.5 power of total particles.

Finally, the upstream velocity $\vec{u}_i^{*upstream}$ in (9) was calculated as the average velocity around particle k_2 as follows:

$$\vec{u}_i^{*upstream} = \frac{1}{\sum_{j \neq i} w^*(\vec{r}_j - \vec{r}_i)} \sum_{j \neq i} \{\vec{u}_j w^*(\vec{r}_j - \vec{r}_i)\}, \quad (15)$$

where $w^*(r)$ is a weight function expressed as follows:

$$w^*(r) = \begin{cases} 1 & (r \leq r_e^*) \\ 0 & (r > r_e^*) \end{cases}. \quad (16)$$

In the above equation, r_e^* is the radius for average and its value was determined as $1.5 l_0$ in this study. In (15), the wall and dummy particles were neglected.

Equation (9) implies that the normal component of the temporary velocity is corrected to that of the upstream velocity by considering the conservation of fluid mass in the entire simu-

lation domain. To prevent harmful effects of the outflow boundary on the upstream, the upstream velocity distribution was approximately imposed to the outflow boundary by giving the upstream particles' velocities to the boundary particles. The parameter Δu decreases the outflow velocity when the amount of fluid is reduced from the initial value. In contrast, Δu increases the fluid velocity when the amount of fluid is increased from the initial value. This boundary condition assumes that backflow does not occur. In the case where the corrected velocity was reversed on the outflow boundary, the velocity component in the $\vec{n}$ direction was set to zero as an exemption. Although there is the restriction about flow direction, this boundary condition can deal with vortices where relative velocity from the center of the vortex is backward as long as the absolute flow velocity is forward.

3.2.2 Deletion of particles

The fluid outflow is expressed by deleting the particles that exited the outflow boundary plane. In the actual implementation of the program, the particle type of each outflow particle was changed from fluid to "ghost," which does not affect the other particles. In the inflow boundary procedure, the ghost particles were reused as the inflow particles by first changing the particle type back to dummy, then to wall, and finally to fluid.

Figure 3:
Schematic diagram of searching upstream particles
 k_1 : Upstream particle selected in the first search
 k_2 : Upstream particle selected in the second search
 $L_{velocity}$: Length of fixed velocity region
($L_{velocity} = 2.1l_0$ in this study)

3.3. Dirichlet boundary condition for pressure

3.3.1. Existing boundary condition

A Dirichlet boundary condition is necessary to solve the simultaneous equations for the pressure Poisson equation. Usually, $P=0$ is given to the free-surface particles as the Dirichlet boundary condition in the MPS method. The free-surface particles are defined with the particle number density. The particle whose particle number density is below βn^0 is identified as a free-surface particle. The value 0.97 is generally used for β in MPS simulations. However, to simulate the Karman vortices in a channel without free surfaces, the usual way is not able to be adopted because there are no free-surface particles. Moreover, using the existing boundary condition, the Dirichlet boundary conditions are mistakenly applied to some inner particles in the wake region because the particle number density is sometimes slightly lower than the

threshold. Moreover, pressure that is lower than zero is reset to zero in the usual MPS simulations to avoid unnatural low pressure around free surfaces. For these reasons, it was difficult to express low pressure in the wake region. As a result, unnatural cavities were occurred and Karman vortices were not simulated in the past MPS simulations. Therefore, a new technique for giving Dirichlet boundary condition is developed in this study. The detail is explained in the next item.

3.3.2. Modified boundary condition

In this study, the Dirichlet boundary condition ($P = 0$) was given to the particles satisfying the following two conditions. First, the particle position is within $4.1l_0$ of the outflow plane. This width $4.1l_0$ was determined considering the radius of interaction domain for particle number density, which was $2.1l_0$; the width should be larger than the radius. Second, the particle number density is lower than βn^0 . Large β increases the number of particles whose pressures are fixed. The parameter β was 0.97 in this study; the value 0.97 is generally used for β in MPS simulations [2, 22-24, 47, 48]. To simulate low pressure, the process of resetting it to zero (which is generally carried out in the MPS method) was not performed throughout the simulation domain, with the exception of the outflow region, i.e., within $4.1l_0$ of the outflow plane. This improvement allows us to express low pressure and avoid the unnatural cavity in the wake region. The width of $4.1l_0$ was determined considering the radius of interaction domain for particle number density, which was $2.1l_0$; the width should be larger than the radius although the width does not have a significant effect on the simulation. To avoid nonphysical low pressures due to the small particle number density on the outflow boundary, the temporary particle number density n^* in the pressure Poisson equation in the MPS method was corrected to n^0 when $\beta n^0 < n^* < n^0$ in the outflow region. This indicates that the source term of the pressure Poisson equation was corrected to zero.

3.4. Inflow boundary

Figure 4 shows a schematic diagram of the inflow boundary algorithm. Figure 4(a) shows the initial state, while Figure 4(b) shows that the particles on the inflow boundary are forced to move rightward at a designated inflow velocity. The fluid particles in the channel were pushed to the right by the left wall particles. The pushing force that acted on the fluid particles was based on the pressure gradient in the Navier-Stokes equation (2). When the distance of the particles moving on the inflow boundary is equal to or greater than the initial spacing between the particles (as shown in Fig. 4(c)), a single layer of dummy particles is generated at the left end at a distance of l_0 from the right particles, as shown in Fig. 4(d), where l_0 represents the initial spacing between particles. Furthermore, the wall particles on the inflow boundary change to fluid particles, while the inner dummy particles change to wall particles. Using this algorithm, we impose a zero pressure gradient condition on the inflow boundary.

3.5. Fixed cylinder boundary

A cylinder in a flow consists of particles that are fixed in the initial position. There are no

cavities in the cylinder. Pressure of the fixed particles is calculated in the same manner as that of the fluid particles. The velocity of the fixed particles is set to zero. The other properties of the rigid particles are the same as those of the fluid particles.

3.6. Free-slip wall boundary

In the usual simulation of the MPS method, wall boundaries consist of three layers of particles, as shown in Fig. 4. The inner layer, which is adjacent to fluid particles, consists of wall particles whose pressures are calculated in the fluid simulation. The outer two layers consist of dummy particles, which are used to calculate only the particle number density. Dummy particles do not have pressure variables. Using this algorithm, we impose a zero pressure gradient condition on the wall boundary. To express the free-slip wall boundary condition, the wall and dummy particles are neglected in the interaction for the viscosity term in (2).

Figure 4:
Schematic diagram of the inflow algorithm
Sequence is from (a) to (d).

4. Verification analysis

4.1. Simulation condition

Figure 2 shows the simulation domain. The left plane indicates the inflow boundary and the right plane indicates the outflow boundary. The other two side boundaries were free-slip wall boundaries. Either a circular or square cylinder was fixed in the channel. The simulation for square cylinder was carried out to confirm that the MPS method can express the shape difference of cylinders and the developed boundary condition is able to be applied to various flows.

The boundary conditions explained in Section 3 were adopted for these boundaries. The inflow velocity distribution was uniform and was kept constant. The radii of the particle interaction domain for the gradient model and the particle number density were $2.1l_0$. The radius for the Laplacian model was $3.1l_0$. The diameter of the cylinders D was 0.10 m. The fluid mass density was 1000 kg/m³ and the acceleration of gravity was zero. The simulations were carried out in two spatial resolutions: $l_0 = D/20$ and $D/40$. The total number of particles for the two spatial resolutions was approximately 83,000 and 326,000, respectively.

The force $\vec{F}$ acting on the cylinders was calculated as follows:

$$\vec{F} = -\int_V \nabla P dV + \vec{f}_{friction}, \quad (17)$$

where $\vec{f}_{friction}$ is the force vector due to viscosity and was obtained by the viscous term of the Navier-Stokes equation (2). In this study, the first term on the right-hand side of (17) is expressed by the volume integration of the pressure gradient using Gauss' divergence theorem, although in mesh methods, it is expressed by the surface integration of pressure on the cylinder. The reason is that volume integration is easier than surface integration for the MPS method because the normal vectors on the cylinder surface are not necessary for volume integration. The force $\vec{F}$ was evaluated for both the drag coefficient and the lift coefficient. The simulation was carried out for two Reynolds numbers, $Re = 100$ and 200, where the flow fields were considered to be laminar and turbulence effects could be neglected.

4.2. Simulation results

Figure 5 shows the simulated fluid behavior. The parameter $t' (= tU/D)$ represents dimensionless time. The upstream particles on the cylinder are colored in blue and red to visualize the vortices. The width of each colored region was $0.25D$ at the inflow boundary. From this figure, it was found that Karman vortices were alternatively generated behind the circular and square cylinders. The sparse region, which was problematic for the MPS method, did not occur behind the cylinder by using the developed boundary conditions. Moreover, the shape of the streak lines was similar to that obtained in the experiment by Perry et al. [26]. The computation time was 1,137 h in the longest case. The total time step number was 83,114. The average time step width $\Delta t'$ was 2.4×10^{-3} . The computer used in this study was equipped with the CPU: Quad-Core Intel Xeon 2.4 GHz and the main memory: 16 GB. A single core was used without parallel computing.

Figure 6 shows the vorticity contour of the simulated Karman vortices for the circular cylinder and $Re = 100$. From this figure, it was confirmed that the vortices were generated behind the cylinder and that they shed downstream. Because the differential operator for the curl, which is necessary to calculate vorticity, had not yet developed for the MPS method, we developed the following curl model and used it to calculate vorticity.

$$\langle \nabla \times \vec{u} \rangle_i = \frac{d}{n^0} \sum_{j \neq i} \frac{(\vec{r}_j - \vec{r}_i) \times (\vec{u}_j - \vec{u}_i)}{|\vec{r}_j - \vec{r}_i|^2} w(|\vec{r}_j - \vec{r}_i|). \quad (18)$$

The Appendix mentions the derivation of the model. In this study, the radius of the interaction domain for the curl model was $10.0l_0$.

Figure 7 shows an example of the simulated time history of the drag coefficient C_d and the lift coefficient C_L for the circular cylinder and $Re = 100$. The force acting on the cylinder in a flow fluctuated, although the fluctuation was suppressed to some degree by the volume integration of the pressure gradient. The reason for this is the pressure oscillation in the MPS method [11-13, 23, 24]. In this study, a low pass filter based on the fast Fourier transform, which is widely used in experiments, was adopted to eliminate the oscillating components in the time history. To determine the threshold frequency of the low pass filter, we investigated the power spectrum, which expresses the frequency components in the time history. Figure 8 shows an example of the power spectrum. This figure reveals that the power spectrum was separated into two parts, the high-frequency part and the low frequency part. In this study, we identified the components whose frequency was over 50 in the dimensionless frequency $f' (= fD/U)$ and eliminated them as high-frequency components, which were noise in the numerical simulation. Figure 9 shows the result of the inverse Fourier transform in which the high-frequency components were eliminated. It was found that the noise disappeared in the time history. In addition, the drag coefficient was almost constant although there were small fluctuations because of the shedding of the Karman vortices. It was also found that an alternating lift force acted on the cylinder.

Table 2: Drag coefficient and Strouhal number (Circular cylinder)

	Re	C_d	St
MPS ($l_0 = D/20$)	100	1.34	0.165
MPS ($l_0 = D/40$)	100	1.28	0.146
FDM by Liu et al.[34]	100	1.35	0.164
FDM by Calhoun [32]	100	1.35	0.175
Experiment by Tritton [29]	100	1.25	0.164(± 0.023)
Experiment by Roshko [30]	100		0.164
MPS ($l_0 = D/20$)	200	1.26	0.169
MPS ($l_0 = D/40$)	200	1.20	0.176
FDM by Liu et al.[34]	200	1.31	0.192
FDM by Calhoun [32]	200	1.17	0.202
FDM by Franke et al.[33]	200	1.31	0.194
Experiment by Kovaznay [31]	200		0.19

Table 3: Drag coefficient and Strouhal number (Square cylinder)

	Re	C_d	St
MPS ($l_0 = D/20$)	100	1.48	0.139
MPS ($l_0 = D/40$)	100	1.47	0.141
FDM by Franke et al.[33]	100	1.61	0.154
MPS ($l_0 = D/20$)	200	1.47	0.148
MPS ($l_0 = D/40$)	200	1.40	0.157
FDM by Franke et al.[33]	200	1.60	0.157

Tables 2 and 3 show the drag coefficients and the Strouhal numbers for the circular and square cylinders, respectively. The drag coefficient was calculated by averaging the time history in the absence of the high frequency components over 50 in the dimensionless frequency. The Strouhal number was also calculated with the zero-crossing frequency in the time history of the lift force without the components whose frequency was over 50 in the dimensionless frequency. The drag coefficient obtained by the MPS at the high spatial resolution ($l_0 = D/40$) was lower than that at the low spatial resolution ($l_0 = D/20$), and was close to the experimental value obtained by Torriton [29]. A possible reason for this difference is that enhanced spatial resolutions improved the expression of the surface shape of the cylinders. The drag coefficients at $Re = 200$ were lower than those at $Re = 100$ because the viscosity force was weakened. In the MPS results, the Strouhal numbers at $Re = 200$ were higher than those at $Re = 100$. These tendencies agreed with those of the finite difference method (FDM) and experiments. The drag coefficients of the square cylinder were higher than those of the circular cylinder. The Strouhal numbers of the square cylinder were lower than those of the circular cylinder. These tendencies were also in agreement with the FDM results. However, there were some differences between the MPS and FDM results for the drag coefficient and the Strouhal number. One possible reason for these differences is the boundary condition for viscosity calculation. As mentioned in Section 3.5, the velocity of the fixed particles on the cylinder was fixed to zero in the MPS simulation, while wall cells on the cylinder surface are usually given reverse velocity of neighboring fluid in the FDM. To more accurately express the no-slip boundary condition in the MPS method, we need to adopt the velocity condition used in the FDM for the MPS simulation, although the implementation is somewhat complicated. The second reason is that numerical diffusion in the convection term does not occur in the MPS method because the Lagrange description is adopted. The third possible reason is that dimensions of the simulation domain influenced the simulation results although the domain has a certain dimensions; the width of the simulation domain in x and y -directions were $20D$ and $10D$. These effects are subjects for

future studies.

(a) Circular cylinder, $Re = 100$

(b) Circular cylinder, $Re = 200$

(c) Square cylinder, $Re = 100$

(d) Square cylinder, $Re = 200$

Figure 5: Karman vortices by the MPS method

Figure 6: Vorticity contours (Circular cylinder, $Re = 100$)

Figure 7: Time history of the drag and the lift (Circular cylinder, $R_e = 100$)

Figure 8: Power spectrum of the drag and lift forces (Circular cylinder, $R_e = 100$)

Figure 9: Time history of the drag and lift using the low pass filter
(Circular cylinder, $R_e = 100$)

5. Conclusion

Boundary conditions of the MPS method were developed for the simulation of Karman vortices without unphysical cavities. The Dirichlet boundary condition for pressure was modified to express low pressure at the wake region. Moreover, the velocity on the outflow boundary was corrected to conserve the fluid mass in the entire simulation domain.

To verify the developed boundary conditions, the Karman vortices behind the circular and square cylinders in a flow were simulated by the MPS method using the developed boundary conditions. The drag and lift coefficients that act on cylinders were compared with those of the FDM and experiments at two Reynolds numbers. The result showed that the effects of the Reynolds number for the drag coefficients and the Strouhal number obtained by the MPS method were close to those of the FDM and experiments. The drag coefficients for the square cylinder were higher than those for the circular cylinder. The Strouhal numbers for the square cylinder were lower than those for the circular cylinder. These tendencies for the cylindrical shapes were in agreement with the FDM results. From these results, it was verified and validated that the presented boundary condition enables the MPS method to simulate the Karman vortices.

Acknowledgement

This work was supported by the Japan Society for the Promotion of Science (Grants-in-Aid for Scientific Research, 25420860).

Abbreviations

A : cross-sectional area of outflow boundary

C_d : drag coefficient, $(C_d = F_x / (0.5\rho U^2 S))$

C_L : lift coefficient, $(C_L = F_y / (0.5\rho U^2 S))$

d : number of space dimensions

D : diameter of cylinder

F_x : force in x -direction

F_y : force in y -direction

f' : dimensionless frequency of lift force, $(f' = fD/U)$

$\bar{g}$: acceleration of gravity

$L_{velocity}$: length of fixed velocity region

l_0 : initial spacing between particles

n_i : particle number density of i -th particle

n^0 : constant of particle number density

n^* : temporary particle number density after the explicit phase

N : number of particles

P : pressure

$\vec{r}$: position vector of particle

r : distance between particles

r_e : radius of the interaction domain

Re : Reynolds number, ($Re = UD / \nu$)

S : projected area of cylinder

S_i : Strouhal number, ($S_i = fD / U$)

t : time

t' : dimensionless time, ($t' = tU / D$)

U : inflow velocity

$\vec{u}$: velocity vector

u : flow velocity in x -direction

v : flow velocity in y -direction

w : weight function

α : relaxation coefficient

β : parameter for free surface detection

λ : parameter for Laplacian model

μ : viscosity coefficient

ν : kinematic viscosity coefficient

ρ : fluid density

ϕ : arbitrary quantity

$\hat{\phi}_i$: minimum quantity around i -th particle, ($\hat{\phi}_i = \min \{\phi_j, \phi_i\}, |\vec{r}_j - \vec{r}_i| \leq r_e$)

Appendix A. Derivation of the curl model for the MPS method

The differential operator for the curl is written in the following expression:

$$\nabla \times \vec{u} = \varepsilon^{\alpha\beta\gamma} \frac{\partial u^\gamma}{\partial x^\beta} \vec{e}^\alpha, \quad (\text{A.1})$$

where the superscripts indicate the coordinate directions. The gradient model of the MPS method (Eq. (3)) is also expressed in the following expression:

$$\left\langle \frac{\partial \phi}{\partial x^\beta} \right\rangle_i = \frac{d}{n^0} \sum_{j \neq i} \frac{(\phi_j - \phi_i)}{|\vec{r}_j - \vec{r}_i|^2} x_{ij}^\beta w(|\vec{r}_j - \vec{r}_i|), \quad (\text{A.2})$$

$$x_{ij}^\beta = x_j^\beta - x_i^\beta, \quad (\text{A.3})$$

where the bracket $\langle \rangle_i$ implies that the derived function is discretized by a particle approximation. The parameter x_i^β indicates the i -th particles' positional coordinate in the β direction. By substituting (A.2) for the derived function in (A.1), we obtain the following equation:

$$\langle \nabla \times \vec{u} \rangle_i = \varepsilon^{\alpha\beta\gamma} \left\langle \frac{\partial u^\gamma}{\partial x^\beta} \right\rangle_i \vec{e}^\alpha = \varepsilon^{\alpha\beta\gamma} \frac{d}{n^0} \sum_{j \neq i} \frac{(u_j^\gamma - u_i^\gamma)}{|\vec{r}_j - \vec{r}_i|^2} x_{ij}^\beta w(|\vec{r}_j - \vec{r}_i|) \vec{e}^\alpha = \frac{d}{n^0} \sum_{j \neq i} \frac{\varepsilon^{\alpha\beta\gamma} x_{ij}^\beta (u_j^\gamma - u_i^\gamma) \vec{e}^\alpha}{|\vec{r}_j - \vec{r}_i|^2} w(|\vec{r}_j - \vec{r}_i|) \quad (\text{A.4})$$

Using a vector expression, we obtain the following curl model for the MPS method.

$$\langle \nabla \times \vec{u} \rangle_i = \frac{d}{n^0} \sum_{j \neq i} \frac{(\vec{r}_j - \vec{r}_i) \times (\vec{u}_j - \vec{u}_i)}{|\vec{r}_j - \vec{r}_i|^2} w(|\vec{r}_j - \vec{r}_i|). \quad (\text{A.5})$$

By replacing d/n^0 with the following relational expression derived by Suzuki [49]:

$$\frac{d}{n^0} \cong \left[\sum_{j \neq i} w(|\vec{r}_j - \vec{r}_i|) \left\{ \frac{(\vec{r}_j - \vec{r}_i)}{|\vec{r}_j - \vec{r}_i|} \otimes \frac{(\vec{r}_j - \vec{r}_i)}{|\vec{r}_j - \vec{r}_i|} \right\} \right]^{-1}, \quad (\text{A.6})$$

we obtain the following first order curl model:

$$\langle \nabla \times \vec{u} \rangle_i = \left[\sum_{j \neq i} w(|\vec{r}_j - \vec{r}_i|) \left\{ \frac{(\vec{r}_j - \vec{r}_i)}{|\vec{r}_j - \vec{r}_i|} \otimes \frac{(\vec{r}_j - \vec{r}_i)}{|\vec{r}_j - \vec{r}_i|} \right\} \right]^{-1} \sum_{j \neq i} \frac{(\vec{r}_j - \vec{r}_i) \times (\vec{u}_j - \vec{u}_i)}{|\vec{r}_j - \vec{r}_i|^2} w(|\vec{r}_j - \vec{r}_i|). \quad (\text{A.7})$$

References

- [1] J.J. Monaghan: Smoothed particle hydrodynamics, *Annu Rev Astron Astrophys*, 30 (1992) 543-574.
- [2] S. Koshizuka, Y. Oka: Moving-particle semi-implicit method for fragmentation of incompressible fluid, *Nucl Sci Eng*, 123 (1996) 421-434.
- [3] J. Kajtar, J.J. Monaghan: SPH simulations of swimming linked bodies, *J Comput Phys*, 227:19 (2008) 8568-8587.
- [4] J.B. Kajtar, J.J. Monaghan: On the dynamics of swimming linked bodies, *Eur J Mech B Fluids*, 29:5 (2011) 377-386.
- [5] A. Khayyer, H. Gotoh, S.D. Shao: Corrected incompressible SPH method for accurate water-surface tracking in breaking waves, *Coastal Engineering*, 55:3 (2008) 236-250.
- [6] A. Khayyer, H. Gotoh, S. Shao: Enhanced predictions of wave impact pressure by improved incompressible SPH methods, *Applied Ocean Research*, 31:2 (2009) 111-131.
- [7] M. Antuono, A. Colagrossi, S. Marrone, D. Molteni: Free-surface flows solved by means of SPH schemes with numerical diffusive terms, *Comput. Phys. Commun.*, 181:3 (2010) 532-549.
- [8] M. Antuono, A. Colagrossi, S. Marrone, C. Lugni: Propagation of gravity waves through an SPH scheme with numerical diffusive terms, *Comput. Phys. Commun.*, 182:4 (2011) 866-877.
- [9] P.W. Cleary: Extension of SPH to predict feeding, freezing and defect creation in low

- [10] S. Idelsohn, M. Mier-Torrecilla, E. Oñate: Multi-fluid flows with the particle finite element method, *Comput Methods Appl Mech Eng*, 198:33-36 (2009) 2750-2767.
- [11] M. Kondo, S. Koshizuka: Improvement of stability in moving particle semi-implicit method, *Int. J. Numer. Meth. Fluids*, 65:6 (2011) 638-654.
- [12] S. Hibi, K. Yabushita: A study on reduction of unusual pressure fluctuation of MPS method, *J. of the Kansai Society of Naval Architects*, 241 (2004) 125-131. (in Japanese)
- [13] M. Tanaka, T. Masunaga: Stabilization and smoothing of pressure in MPS method by quasi-compressibility, *J. Comput. Phys.*, 229:11 (2010) 4279-4290.
- [14] M. Tanaka, T. Masunaga, Y. Nakagawa: Multi-resolution MPS method, *Transactions of JSCEs*, (2009) Paper No.20090001. (in Japanese)
- [15] A. Khayyer, H. Gotoh: Modified moving particle semi-implicit methods for the prediction of 2D wave impact pressure, *Coastal Engineering*, 56:4 (2009) 419-440.
- [16] A. Khayyer, H. Gotoh: A higher order Laplacian model for enhancement and stabilization of pressure calculation by the MPS method, *Applied Ocean Research*, 32:1 (2010) 124-131.
- [17] A. Khayyer, H. Gotoh: Enhancement of stability and accuracy of the moving particle semi-implicit method, *J. Comput. Phys.*, 230:8 (2011) 3093-3118.
- [18] B. H. Lee, J. C. Park, M. H. Kim, S. C. Hwang: Step-by-step improvement of MPS method in simulating violent free-surface motions and impact-loads, *Comput. Methods Appl. Mech. Eng.*, 200:9-12 (2011) 1113-1125.
- [19] Y. Suzuki, S. Koshizuka, Y. Oka: Hamiltonian moving particle semi-implicit (HMPS) method for incompressible fluid flows, *Comput. Methods Appl. Mech. Eng.*, 196:29-30 (2007) 2876-2894.
- [20] H. Hashimoto, M. Sueyoshi, S. Minegaki: An estimation of the anti-rolling tank performance for parametric rolling prevention, *The Japan Society of Naval Architects and Ocean Engineers*, 6 (2007) 305-311. (in Japanese)
- [21] M. Masuda, K. Masuda, T. Ikoma, H. Maeda, A. Kobayashi: A study on analysis of tsunami-induced behaviors of a floating structure using the 2-D MPS method, *The Japan Society of Naval Architects and Ocean Engineers*, 9 (2009) 37-44. (in Japanese)
- [22] M. Sueyoshi, M. Kashiwagi, S. Naito: Numerical simulation of wave-induced non-linear motions of a two-dimensional floating body by the moving particle semi-implicit method, *J. Mar. Sci Technol.*, 13:2 (2008) 85-94
- [23] K. Shibata, S. Koshizuka: Numerical analysis of shipping water impact on a deck using a particle method, *Ocean Engineering*, 34:3-4 (2007) 585-593.
- [24] K. Shibata, S. Koshizuka, K. Tanizawa: Three-dimensional numerical analysis of shipping water onto a moving ship using a particle method, *Journal of Marine Science and Technology*, 14:2 (2009) 214-227.

-
- [25] M.M. Tsukamoto, L. Y. Cheng, K. Nishimoto: Analytical and numerical study of the effects of an elastically-linked body on sloshing, *Computers & Fluids*, 49:1 (2011) 1-21.
- [26] A.E. Perry, M.S. Chong, T.T. Lim: The vortex-shedding process behind two-dimensional bluff bodies, *J. Fluid Mech.*, 116 (1982) 77-90.
- [27] C.H.K. Williamson: Three-dimensional wake transition, *J. Fluid Mech.*, 328 (1996) 345-407.
- [28] C.H.K. Williamson: Vortex dynamics in the cylinder wake, *Annu. Rev. Fluid. Mech.*, 28 (1996) 477-539.
- [29] D.J. Tritton: Experiments on the flow past a circular cylinder at low Reynolds numbers, *J. Fluid. Mech.*, 6:4 (1959) 547-567.
- [30] A. Roshko: On the development of turbulent wakes from vortex streets, *National Advisory Committee for Aeronautics*, 1191 (1954) 1-25
- [31] L.S.G. Kovasznay: Hot-wire investigation of the wake behind cylinders at low Reynolds numbers, *Proc. R. Soc. Lond. A.*, 198:1053 (1949) 174-190.
- [32] D. Calhoun: A Cartesian grid method for solving the two-dimensional streamfunction-vorticity equations in irregular regions, *J. Comput. Phys.*, 176:2 (2002) 231-275.
- [33] R. Franke, W. Rodi, B. Shönung: Numerical calculation of laminar vortex-shedding flow past cylinders, *J. Wind Eng. and Industrial Aerodynamics*, 35 (1990) 237-257.
- [34] C. Liu, X. Zheng, C.H. Sung: Preconditioned multigrid methods for unsteady incompressible flows, *J. Comput. Phys.*, 139:1 (1998) 35-57.
- [35] M. Cheng, G.R. Liu, K.Y. Lam: Numerical simulation of flow past a rotationally oscillation cylinder. *Computers & Fluids*, 30:3 (2001) 365-392.
- [36] D. H. Kim, K. S. Yang, M. Senda: Large eddy simulation of turbulent flow past a square cylinder confined in a channel, *Computers & Fluids*, 33:1 (2004) 81-96.
- [37] Z. Huang, J.A. Olson, R.J. Kerekes, S.I. Green: Numerical simulation of the flow around rows of cylinders, *Computers & Fluids*: 35:5 (2006) 485-491.
- [38] M.R.H. Nobari, J. Ghazanfarian: A numerical investigation of fluid flow over a rotating cylinder with cross flow oscillation, *Computers & Fluids*, 38:10 (2009) 2026-2036.
- [39] J. Schwing, D.E. Wiley, D.F. Fletcher: Simulation of the flow around spacer filaments between narrow channel walls. 1. Hydrodynamics, *Ind. Eng. Chem. Res.*, 41 (2002) 2977-2987.
- [40] J. Schwing, D.E. Wiley, D.F. Fletcher: Simulation of unsteady flow and vortex shedding for narrow spacer-filled channels. *Ind. Eng. Chem. Res.*, 42:20 (2003) 4962-4977.
- [41] J.B. Joshi, M.V. Tabib, S.S. Deshpande, C.S. Mathpati: Dynamics of flow structures and transport phenomena, 1. experimental and numerical techniques for identification and energy content of flow structures, *Ind. Eng. Chem. Res.*, 48:17 (2009) 8244-8284.
- [42] M. Lastiwka, M. Basa, N.J. Quinlan: Permeable and non-reflecting boundary conditions in SPH, *Int. J. Numer. Meth. Fluids*, 61:7 (2009) 709-724.

- [43] M. Yildiz, R.A. Rook, A. Suleman: SPH with the multiple boundary tangent method. *Int J Numer Meth Eng*, 77:10 (2009) 1416-1438.
- [44] M.S. Shadloo, A. Zainali, S.H. Sadek, M. Yildiz: Improved Incompressible Smoothed Particle Hydrodynamics method for simulating flow around bluff bodies, *Comput Method Appl M*, 200:9-12 (2011) 1008-1020.
- [45] S. Marrone, A. Colagrossi, M. Antuono, G. Colicchio, G. Graziani: An accurate SPH modeling of viscous flows around bodies at low and moderate Reynolds numbers, *J Comput Phys*, 245 (2013) 456-475.
- [46] K. Nomura, S. Koshizuka, Y. Oka, H. Obata: Numerical analysis of droplet breakup behavior using particle method, *J. Nucl. Sci. Technol. (Tokyo, Jpn.)*, 38:12 (2001) 1057-1064.
- [47] K. Shibata, S. Koshizuka, M. Sakai, K. Tanizawa: Transparent boundary condition for simulating nonlinear water waves by a particle method, *Ocean Engineering*, 38:16 (2011) 1839-1848.
- [48] K. Shibata, S. Koshizuka, M. Sakai, K. Tanizawa: Lagrangian simulations of ship-wave interactions in rough seas, *Ocean Engineering*, 42 (2012) 13-25.
- [49] Yukihiro Suzuki: Studies on improvement in particle method and multiphysics simulator using particle method, *Doctoral thesis, The University of Tokyo* (2008). (in Japanese)