Title:
Simulation of Flow Induced Multi-Particle Motion Using Finite Element Method

Authors:
Xiaobo Peng, xipeng@pvamu.edu, Prairie View A&M University
Diwei Zhang, zhang.diwei@yahoo.com, Prairie View A&M University
Dongdong Zhang, peterzdd_2002@hotmail.com
Lai Jiang, laijiang@pvamu.edu, Prairie View A&M University

Keywords:
Flow Induced Multi-Particle Motion, Finite Element Method, Fluid-Structure Interaction

DOI: 10.14733/cadconfP.2020.91-95

Introduction:
Flow induced multi-particle motion is a common phenomenon which can be found in different areas. Particle immersed in the flow with low-Reynolds number is a main characteristic of this kind of problems. Motions of particles, such as translation, rotation, deformation, interaction between wall and particles, and interaction among particles are complicated problems. Studying motions of particles may reveal insights of multi-particle motion phenomenon and also benefit several different industries. To manufacture the short-fiber reinforced composite material using injection molding process, fibers immersed in the resin matrix are injected into the mold. Fiber's motions including transport, rotation, and deformation are subject to the motion of the fluid. The final distribution and pattern of fibers determines the performance of final products. Similarly, in the paper making industry, pulp with paper fiber in the processing of forming paper is a flow induced fiber motion problem. In the medical application, blood cell motion in the blood vessel also can be categorized into this field. Obtaining insight of physical meaning of blood cell motion may be helpful for understanding mechanism behind some diseases from different perspective.

Flow-induced multi-particle motion has been investigated theoretically, experimentally, and numerically. Research on theoretical methods established the foundations of understanding the single particle or fiber motion. Many experimental research work has been conducted to investigate the flow-induced particle motion problem in the about last seventy years. Main focuses in those experiments are the design of experiment setup, including creating a certain flow profile (such as the Couette flow and the Poiseuille flow), and the methods of observing the motion of particles (such as arrangement of cameras, recording methods and tracking methods). From the second half of twenty century, the numerical methods on solving multi-particle motion became a very powerful tool with the development of the computer. Especially, the finite element method has been developed on solving solid mechanics problems and, the finite volume method was developed on solving the computational fluid dynamics problems. Flow-induced particle motion is categorized into the fluid-structure interaction problem, which involves two domains, i.e., fluid domain and particle domain, and one interface, i.e., interface between fluid and particle. Therefore, solving flow-induced particle motion numerically is a complicated problem involving various solving schemes. There are different ways of modeling particles, such as using bead-chain or rod-chain to represent flexible fiber. There are various ways of solving the fluid domain, such as the lattice boltzmann method, the finite volume method, the finite element method. Coupling strategies are also very important and challenging. It involves the moving mesh and the re-mesh because of large translations and rotations of particles, meanwhile, data transferring among the fluid, the particle, and the interface. There are explicit and implicit methods can be applied in the coupling of fluid domain and particle domain [4].

Methods:
Flow-induced multi-particle motion was studied with numerical method in this paper. Problem was limited in the low Reynolds number flow. Amount of particle was limited in three. Problem was simplified in the two-dimension.

In this paper, the particle domain was modeled as a two-dimensional elastic problem. Since the expected large translation and rotation of particles, non-linear model is applied. To capture the motion of particles, a time dependent governing equation of the particle motion was chosen. The particle motion in this flow-induced multi particle motion solving strategy was described in the total Lagrangaian finite element method [2]. A second order ordinary differential equation was created. Then the Bath method was applied to integrate over time domain to solving that [1]. The damping coefficient of particles was the Rayleigh proportional damping. The final governing equation which participating the coupling strategy directly is shown in Eqn. (1), where a two particles case is presented as a general form of the derived solving scheme.

$$\begin{bmatrix} M^1 & 0 \\ 0 & M^2 \end{bmatrix} \begin{Bmatrix} \ddot{U}^1_{(i)} \\ \ddot{U}^2_{(i)} \end{Bmatrix} + \begin{bmatrix} C^1 & 0 \\ 0 & C^2 \end{bmatrix} \begin{Bmatrix} \dot{U}^1_{(i)} \\ \dot{U}^2_{(i)} \end{Bmatrix} + \begin{bmatrix} K^1_{t\ (i-1)} & 0 \\ 0 & K^1_{t\ (i-1)} \end{bmatrix} \begin{Bmatrix} U^1_{(i)} \\ U^2_{(i)} \end{Bmatrix} = \begin{Bmatrix} {}^2_0F^1_{external} - {}^1_0F^1_{internal}(U^1_{(i-1)}) \\ {}^2_0F^2_{external} - {}^1_0F^2_{internal}(U^2_{(i-1)}) \end{Bmatrix} \tag{1}$$

where M, C, and K_t are mass matrix, damping matrix, and tangential stiffness matrix, respectively. U represents displacement. Superscript 1 and 2 on the top right corner indicates the number of the particles. The 'dot' above the displacement U indicates the order of time derivative. Subscript i and $i-1$ on the bottom right corner of the term relating to U indicates the current configuration and the previous configuration of particles. ${}^2_0F_{external}$ and ${}^1_0F_{internal}$ represent external forces on the current configuration and the internal forces stored from the previous configuration.

Since most of flow-induced particle motion problems are studied in the condition with low-Reynolds number flow, applying zero-Reynolds number Stokes equation is reasonable mathematically. Energy conservation was not considered. The mass conservation and the momentum conservation applied in this work are presented in Eqn. (2) and Eqn. (3), respectively.

$$\boldsymbol{\nabla} \cdot \mathbf{V} = 0 \tag{2}$$

$$-\boldsymbol{\nabla} \cdot \{-p\mathbf{I} + [(\boldsymbol{\nabla}\mathbf{V}) + (\boldsymbol{\nabla}\mathbf{V})^T]\} = 0 \tag{3}$$

where $\mathbf{V}, p,$ t, and $\boldsymbol{\nabla}$ represent velocity vector, pressure, time, and the gradient operator, respectively. Those equations are time-independent, which means the fluid domain was modeled as a quasi-static problem. It is appropriate in this case since the transient dynamic doesn't influence the result significantly under the zero Reynolds number assumption. It is also worth to mention that Eqn. (2) and Eqn. (3) are dimensionless equations. Governing equation of the fluid in this study was solved by the mixed finite element method which is a finite element method. It is able to solve terms of different order together. In this case, the velocity is one order higher than the pressure mathematically. Since the advection term doesn't appear in the Stokes equation, pure finite element discretization without upwind treatment is able to avoid losing fidelity. The advantage of using finite element based method for solving the fluid domain is that this method is the same method used in solving the particle motion, which makes it easy to address issues related to coupling of mesh and data. After using mixed finite element method, Eqn. (2) and Eqn. (3) can be written into a matrix form equation as in Eqn. (4), which was directly involved in the coupling calculation with the particle motion governing Eqn. (1) [3].

$$\mathbf{K}_f \cdot \mathbf{X}(\mathrm{V_x}, \mathrm{V_y}, \mathrm{P}) = \mathbf{F}(\mathrm{F_x}, \mathrm{F_y}) \tag{4}$$

where $\mathbf{K_f}$ is the stiffness matrix of the fluid, $\mathbf{X}$ is the state including the fluid velocity in the x-direction V_x and in the y-direction V_y, and the pressure P. The term $\mathbf{F}$ represents the forces exerting on the fluid domain in the x and y direction, $\mathrm{F_x}$ and $\mathrm{F_y}$, respectively.

To illustrate the setup of boundary condition on both fluid and particle domain, a single particle immersed in the Poiseuille flow is applied as shown in Fig. 1. Fig. 1. shows that the Poiseuille flow profile equation is incorporated on the left side of the fluid domain. The do-nothing boundary condition is applied on the output, in this case on the right side of the fluid domain. No-slip boundary condition is incorporated on the top and the bottom wall of the fluid domain, as well as the surface of the particle which are highlighted with bold blue point in the Fig. 1 (b). A pressure node highlighted as a “star”, as shown in Fig. 1 (b), was incorporated on the left side of the fluid domain which was required to ensure Eqn. (4) to be solvable.

(a) (b)

Fig. 1: Boundary conditions in the case of a single particle immersed in the Poiseuille flow. (a) boundary conditions of the fluid domain; (b) Mesh of the fluid domain with highlighted boundary condition.

An explicit partitioned coupling strategy was applied to couple the particle and the fluid domain. After incorporating the boundary condition, the fluid domain was calculated first. Forces on the surface of the particle was extracted from the fluid solver. Then, the extracted forces were incorporated into the particle motion solver as boundary condition, which satisfies the dynamic compatibility. Under the loading of the fluid, particle solver was operated and matched forward a short time step to obtain a new state of the particle. The new state of the particle includes positions and velocity of each nodes. Re-meshing the fluid domain is to match the new position of the particle, which satisfies the geometrical compatibility. The new velocities of the particle were incorporated back to the fluid solver as a new boundary condition for the next coupling step, which satisfies the kinematic compatibility. The coupling scheme involving Eqn. (1) and Eqn. (4) is illustrated in

Fig. 2.

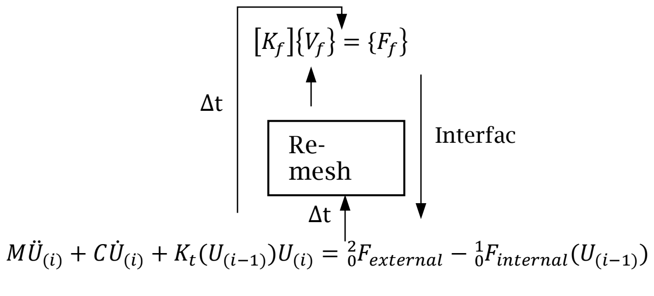


Fig. 2: Illustration of the coupling scheme between the particle and the fluid domain.

<u>Results and Discussion:</u>

To simulate how the existence of particle influences each other, a setup of three particles diagonally immersed in the double Couette flow was simulated. A schematic diagram of the simulation is shown in Fig. 3. Parameters of the simulation are listed in Tab. 1. Four cases were simulated, where three particles are initially placed in different distances (i.e., 3.5mm, 4mm, 4.5mm, 5mm) between each other in the vertical direction. Only the result of case of each particle keeping 5 mm distance is presented to illustrate the flow-induced multi-particle motion as shown in Fig. 4. From 22 sec to 42 sec, flow induced three particles drift closer and then apart away. The changing of displacement in the vertical direction of the particle on the top and the particle on the bottom shows that the existence of other particles affecting the motion of each other, as shown in Fig. 5.

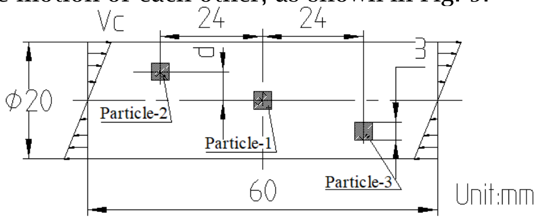
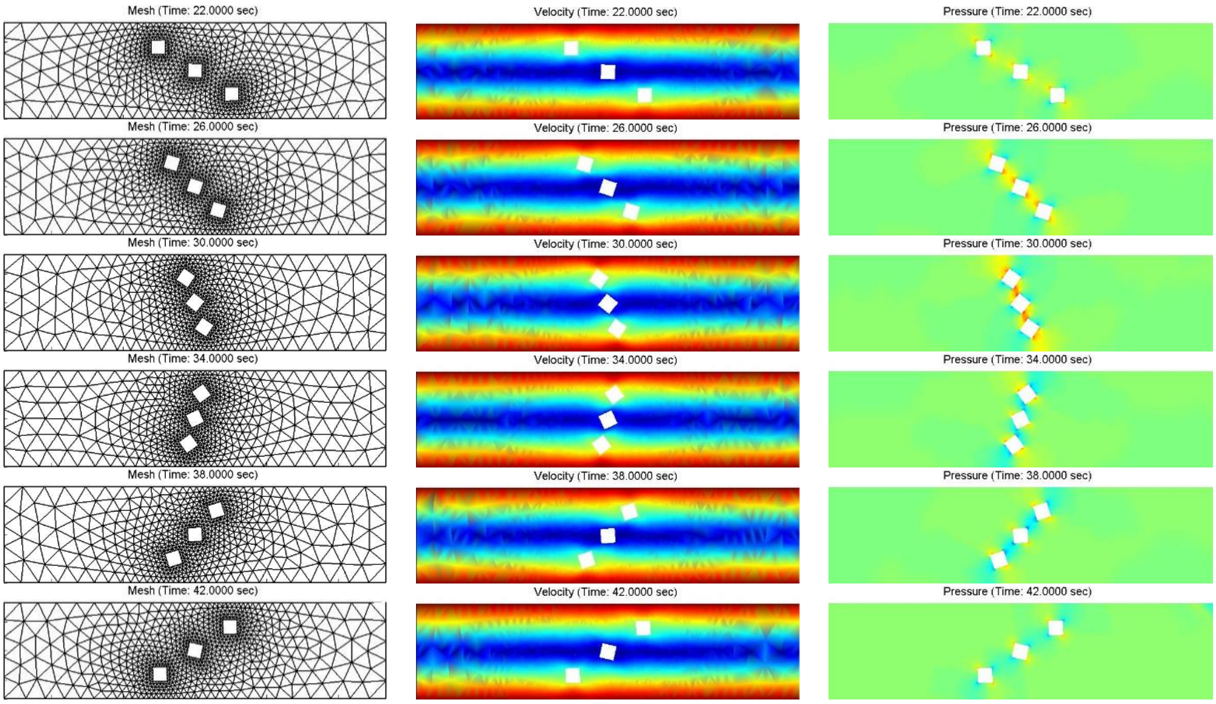


Fig. 3: Simulation of three particles immersed in the double Couette flow.

	Particle-1	Particle-2	Particl-3
Type of flow	Double Couette		
Viscosity $(N\cdot s)/m^2$	17		
Fluid domain height:	2h = 0.02 m		
V_c (m/s)	V_c = 1.5 mm/s		
Shear rate: $\dot{\gamma}$ (1/s)	$\dot{\gamma} = 0.15\ (1/s)$		
Fiber density ρ_s	$\rho_s = 1300\ kg/m^3$		
Long axis length	Square particle: a = 3 mm		
Fiber ratio: re	re = 1		
Time step Δt (s)	Δt = 0.025		
Young's Modulus: E	37e9 (Pa)		
Damping ratio: ζ	ζ = 0		
Distance: d (mm)	3.5, 4.0, 4.5, 5.0		

Tab. 1: Parameters of simulations of three particles diagonally immersed in the double Couette flow.



(a) (b) (c)

Fig. 4: Results of simulation of three particles diagonally initialized arranged in the double Couette

flow. (a) Mesh; (b) Velocity distribution; (c) Pressure distribution.

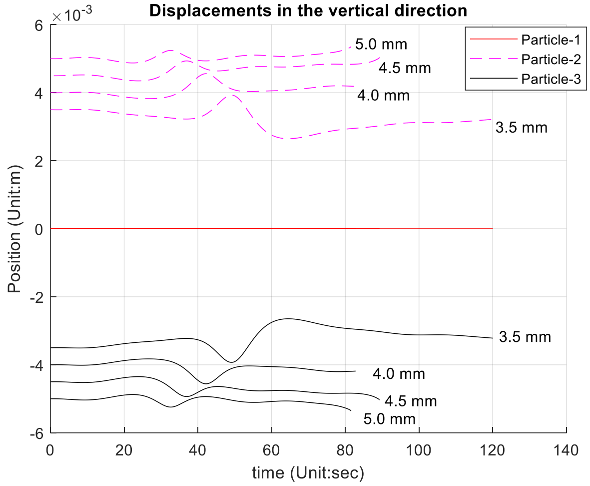


Fig. 5: Displacements in the vertical direction of three particle of cases of different distances between each particle.

Fig. 5. shows that particle-1 which was placed in the center at the beginning did not displace in the vertical direction obviously. Curves of motions of particle-2 and particle-3 in the vertical direction show symmetry in all four cases with different initial distances between particles. Particle-2 and particle-3 move slightly close to the particle-1 in the vertical direction when they approximate to the particle-1 in the horizontal direction. Then, they repel each other to the largest distance in the vertical direction when three particles aligned vertically. After that, particles tend to move closer again vertically when they depart horizontally. Finally, particle-2 and particle-3 recover to their initial vertical position after departing far away from each other. This numerical experiment shows flow-induced multi-particle motion, and reveal how particles influence the motions of each other in the direction vertical to the flow shear direction.

Conclusion:

For solving flow-induced multi-particle motion problem, a finite element based partitioned coupling method is presented in this paper. Results of simulation of three particle initially diagonally immersed in the double Couette flow is also discussed. The simulation revealed how the motions of of particles are influenced by each other.

References:

[1] Bathe, K. J.: Conserving energy and momentum in nonlinear dynamics: a simple implicit time integration scheme, Computers and Structures, 85(7–8), 2007, 437–445. https://doi.org/10.1016/j.compstruc.2006.09.004
[2] Reddy, J. N.: An Introduction to Nonlinear Finite Element Analysis, Oxford University Press, Oxford, 2004. https://dx.doi.org/10.1093/acprof:oso/9780198525295.001.0001
[3] Reddy, J. N.; Gartling, D. K.: The Finite Element Method in Heat Transfer and Fluid Dynamics, 3rd Edition, CRC Press, New York, 2010. https://doi.org/10.1201/9781439882573
[4] Zhang, D.; Peng, X.; Zhang, D.: Flexible fiber motion in fiber-reinforced composite material processing, in the Proceedings of the ASME International Mechanical Engineering Congress and Exposition, Pittsburgh, Pennsylvania, November 9-15, 2018. https://doi.org/10.1115/IMECE2018-86440