Journal of Engineering and Applied Sciences 11 (11): 2495-2499, 2016
ISSN: 1816-949X

Verification of Moving Particle Pressure Mesh (MPPM) Method in Simulating Nonnewtonian Flow in a Square Cavity

Khai-Ching Ng, Tony Wen-Hann Sheu and Yao-Hsin Hwang
Center of Fluid Dynamics, Universiti Tenaga Nasional, Kajang, Malaysia

Abstract: The simulation tool based on particle-mesh method, namely the Moving Particle Pressure Mesh (MPPM) method has been previously developed by the researchers to simulate incompressible flow with constant viscosity. In this study, the MPPM method is extended to handle fluid with variable viscosity which is commonly found in a non-Newtonian flow case. A numerically consistent method which is derived based on the Generalized Finite Difference (GFD) technique is used to discretize the viscous term. The numerical results are verified against those from the benchmark finite-element solutions and good agreements are found.

Key words: Moving Particle Semi-implicit (MPS), Moving Particle Pressure Mesh (MPPM), non-Newtonian flow, incompressible flow, Computational Fluid Dynamics (CFD)

INTRODUCTION

There has been a notable growth of interest in the computation of fluid flow by using the so-called mesh-free (or mesh-less) methods since the past decade. As compared to the conventional mesh-based method (such as finite-volume, finite difference, etc.,) the mesh-free methods are attractive in such a way that the flow field is represented by a series of particles (or computational points which are freely moving in particle-based method). Since mesh is not required, this procedure is able to solve a lot of well-known issues in the mesh-based method such as the considerable time spent to ensure a so-called "good-quality" mesh. More importantly, the convective instability issue appeared in the mesh-based method can be circumvented by using the particle-based approach via the Lagrangian description of flow convection. To date, the two most popular particle-based method is the Smoothed Particle Hydrodynamics (SPH) and Moving Particle Semi-implicit (MPS) methods which have been used extensively in a wide range of engineering applications. Interested readers can refer to the excellent review papers reported by Liu and Liu (2010) and Koshizuka (2011) for SPH and MPS method, respectively.

Although, the particle method such as MPS has enjoyed intense popularity in the CFD community recentlyb (Natsui *et al*., 2014; Ng and Ng, 2013; Ng *et al*., 2013a, b), there are several numerical problems in MPS which must be fully addressed before it becomes a more practical simulation tool (Ng *et al*., 2014). In particular, the considerable amount of tuning efforts required in solving the Pressure Poisson Equation (PPE) in MPS has been eliminated in our recent work of Moving Particle Pressure Mesh (MPPM) method (Hwang, 2011). Very recently, we have witnessed its applicability in multiphase flow of high density ratio (Ng *et al*., 2015).

Based on our literature search, the applications of MPS in non-Newtonian fluid flow is rather limited. Only until recent years and Koshizuka (2011) have ventured into using the implicit MPS method to solve the highly viscous non-Newtonian flow. Also, Xiang and Chen (2015) have recently combined the SPH and MPS methods in their non-Newtonian fluid flow computation. Here, we intend to further extend our MPPM method to handle non-Newtonian fluid flow which is the main contribution of the current research.

In non-Newtonian fluid, the fluid viscosity is varying in accordance with the shear rate (a function of velocity gradient). Therefore, the derivatives of the velocity components must be carefully evaluated in order to yield a correct viscosity field. Here, we are motivated to discretize the velocity gradients in a numerically consistent manner, i.e., the Generalized Finite Difference (GFD) method recently proposed by Luo *et al*. (2015). The combined GFD-MPPM method will be verified against the benchmark solutions obtained from the well-known least-square based finite-element method reported by Bell and Surana (1994).

MATERIALS AND METHODS

Governing equations: The two-dimensional non-Newtonian, incompressible and isothermal lid-driven flow is considered in the current research, whereby the flow field can be expressed by the mass conservation (continuity) equation:

$$\nabla \bullet \vec{u} = 0 \quad (1)$$

Corresponding Author: Khai-Ching Ng, Center of Fluid Dynamics, Universiti Tenaga Nasional, Kajang, Malaysia

and the momentum conservation (Eq. 2 and 3):

$$\rho\frac{Du}{Dt}=-\frac{\partial P}{\partial x}+\frac{\partial}{\partial x}\left(2\eta\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\eta\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right) \quad (2)$$

$$\rho\frac{Dv}{Dt}=-\frac{\partial P}{\partial y}+\frac{\partial}{\partial x}\left(\eta\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right)+\frac{\partial}{\partial y}\left(2\eta\frac{\partial v}{\partial y}\right) \quad (3)$$

The shear rate dependent viscosity η is defined from a potential-type rheological law:

$$\eta=\eta_0\dot{\gamma}^{n-1} \quad (4)$$

where, n is the power-law index. The typical Newtonian fluid flow can be recovered if n = 1. The shear rateis defined as:

$$\dot{\gamma}=\sqrt{2\left(\frac{\partial u}{\partial x}\right)^2+2\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2} \quad (5)$$

The Reynolds number Re for the lid-driven flow case is defined as:

$$Re=\frac{\rho U_{lid}L}{\eta_0}$$

Where:
L = The length of the square cavity
Re = Taken as 100 in this case

Figure 1 shows the schematic diagram of the square cavity.

Numerical discretization: The details of MPPM method can be found in our previous work (Hwang, 2011). The pressure and velocity terms appeared Eq. 1-3 are calculated by the projection method. First, Eq. 2 and 3 are solved without the pressure gradient term on the moving particles (Lagrangian steps). Then the face velocities on the pressure mesh are interpolated from the neighboring moving particles. The Pressure Poisson Equation (PPE) is then constructed on the pressure mesh level (based on Eq. 1 and the new pressure field is obtained. The velocities of the moving particles are subsequently corrected from the neighbouring pressure field. It is important to note that the coefficient matrix in our PPE is built only once (since the pressure mesh is fix) which is somehow different from that in the conventional particle methods. In conventional particle method such as MPS, the coefficient matrix is built at every time step due to the random motions of the fluid particles.

In the current research, we shall emphasize on the discretization procedures of the viscous terms which are directly related to the physics of non-Newtonian fluid

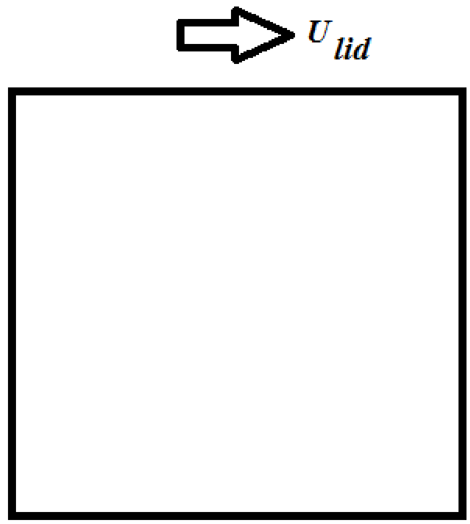

Fig. 1: Lid-driven flow in a square cavity. The top wall is subjected to a constant velocity

flow. In the current research, the viscous terms are discretized by using the Generalized Finite Difference (GFD) technique as detailed by Luo *et al.* (2015). To account for variable viscosity, a recent strategy proposed by Luo *et al.* (2015) is adopted. Accordingly, the derivatives in GFD can be computed as:

$$\eta\frac{\partial u}{\partial x}=\sum_{j\neq i}\left(\frac{1}{0.5(\eta_i^{-1}+\eta_j^{-1})}\right)(u_j-u_i)\,w_j^{\,2}(a_1h_j+a_2k_j0.5a_3h_j^2+a_4h_jk_j+0.5a_5k_j^2) \quad (6)$$

$$\eta\frac{\partial u}{\partial y}=\sum_{j\neq i}\left(\frac{1}{0.5(\eta_i^{-1}+\eta_j^{-1})}\right)(u_j-u_i)\,w_j^{\,2}(b_1h_j+b_2k_j0.5b_3h_j^2+b_4h_jk_j+0.5b_5k_j^2) \quad (7)$$

$$\frac{\partial}{\partial x}\left(\eta\frac{\partial u}{\partial x}\right)=\sum_{j\neq i}\left(\frac{1}{0.5(\eta_i^{-1}+\eta_j^{-1})}\right)(u_j-u_i)\,w_j^{\,2}(c_1h_j+c_2k_j0.5c_3h_j^2+c_4h_jk_j+0.5c_5k_j^2) \quad (8)$$

$$\frac{\partial}{\partial y}\left(\eta\frac{\partial u}{\partial y}\right)=\sum_{j\neq i}\left(\frac{1}{0.5(\eta_i^{-1}+\eta_j^{-1})}\right)(u_j-u_i)\,w_j^{\,2}(e_1h_j+e_2k_j0.5e_3h_j^2+e_4h_jk_j+0.5e_5k_j^2) \quad (9)$$

where, h_j and k_j are (x_j-x_i) and (y_j-y_i), respectively. w_j is the weight function, taken as $1/(r_j-r_i)^3$. The constants a_n, b_n, c_n and e_n are obtained from the inversion of a 5×5 matrix (Luo *et al.*, 2015). It is important to note that the harmonic mean viscosity is considered in the summation procedures (Eq. 6-9). The rationale of using the harmonic mean viscosity can be found in Luo *et al.* (2015).

RESULTS AND DISCUSSION

The above methods have been implemented in the MPPM solver and the numerical results obtained from the lid-driven flow case is shown below. We have performed a grid-independent test for the case of n = 0.25. Obviously, the result computed on 40×40 pressure mesh deviates considerably from those obtained from finer mesh counts. Seemingly, the result is already grid-independent at the pressure mesh resolution of 80×80 as shown in Fig. 2.

Figure 3 and 4 show the velocity profiles at c the mid-sections of the square cavity. Here, the numerical results obtained at the finest particle resolution, i.e., 160×160 are compared with those from (Bell and Surana, 1994). Five cases of different power-law indices (n = 0.25,

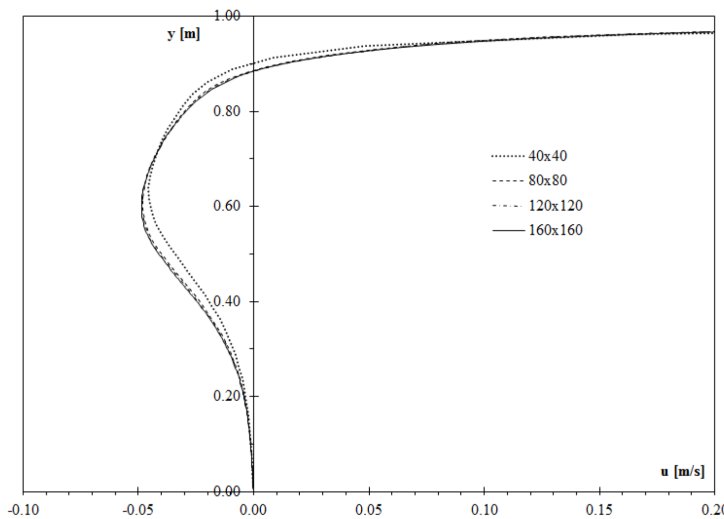

Fig. 2: The y-velocity profiles along the x-axis placed at the mid-section of the square cavity. Cases simulation by using different pressure mesh resolutions are considered

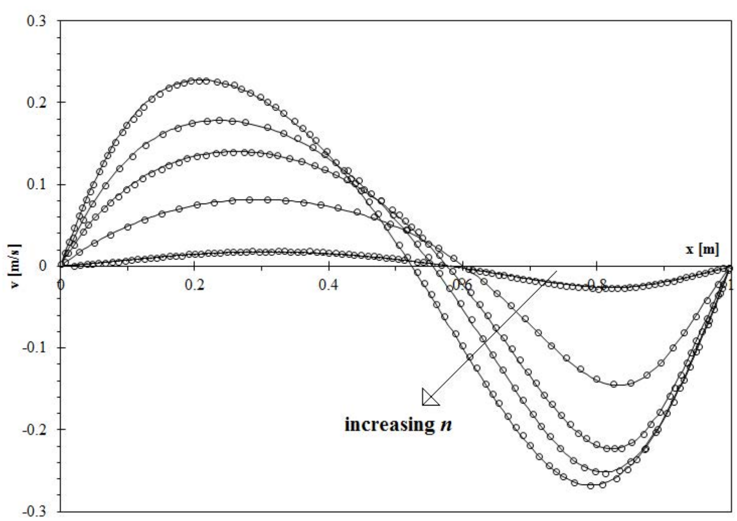

Fig. 3: The y-velocity profiles along the x-axis placed at the mid-section of the square cavity. The empty circles denote the finite-element solutions by Bell and Surana (1994). The 5 cases (n = 0.25, 0.50, 0.75, 1.0 and 1.5) are presented

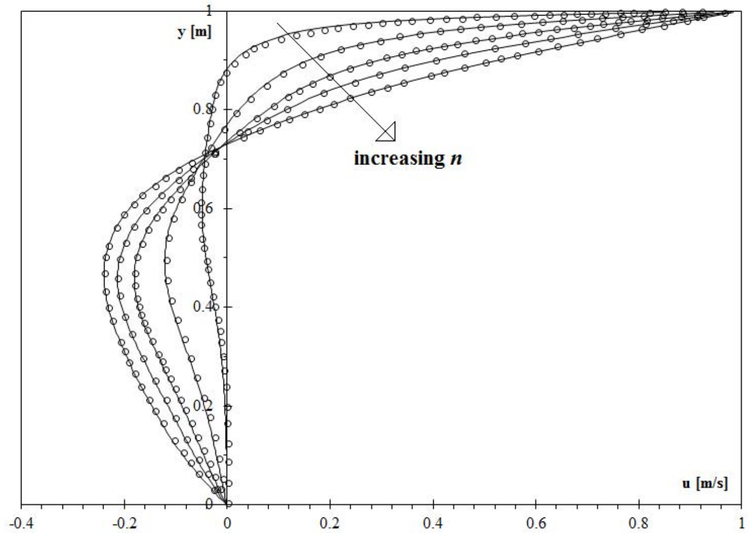

Fig. 4: The x-velocity profiles along the y-axis placed at the mid-section of the square cavity. The empty circles denote the finite-element solutions of Bell and Surana (1994). The 5 cases (n = 0.25, 0.50, 0.75, 1.0 and 1.5) are presented

0.50, 0.75, 1.0 and 1.5) are simulated and a very good agreement is found between both numerical solutions. For all the cases shown in the current work, Eq. 2 and 3 are solved in an implicit manner in order to allow larger time step size without encountering any numerical instability.

CONCLUSION

The Moving Particle Pressure Mesh (MPPM) method has been extended to handle non-Newtonian fluid flow. The viscosity which is expressed as a function of velocity gradients in non-Newtonian fluid is calculated based on the numerically consistent Generalized Finite Difference (GFD) method. The combined GFD and MPPM method has been implemented and used to solve the power-law fluid flow in a square cavity. Good agreement has been found between our numerical solutions and the benchmark finite-element solutions.

Upon verifying our numerical procedure in treating fluid with varying viscosity, our next phase of work is to apply this technique to handle fluid flow with many solid objects. The solid object will be modelled via a localized smoothing function of viscosity which is allowed to change smoothly across the interface.

ACKNOWLEDGEMENTS

Researchers are grateful for the financial sponsorship offered by the Ministry of Science, Technology and Innovation (MOSTI) Malaysia (Project No: 06-02-03-SF0258).

REFERENCES

Bell, B.C. and K.S. Surana, 1994. P-version least squares finite element formulation for two-dimensional, incompressible non-Newtonian isothermal and non-isothermal fluid flow. Int. J. Numer. Methods Fluids, 18: 127-162.

Hwang, Y.H., 2011. A moving particle Method with embedded Pressure Mesh (MPPM) for incompressible flow calculations. Numerical Heat Transfer Part B. Fundam., 60: 370-398.

Koshizuka, S., 2011. Current achievements and future perspectives on particle simulation technologies for fluid dynamics and heat transfer. J. Nuclear Sci. Technol., 48: 155-168.

Liu, M.B. and G.R. Liu, 2010. Smoothed Particle Hydrodynamics (SPH): An overview and recent developments. Arch. Comput. Meth. Eng., 17: 25-76.

Luo, M., C.G. Koh, M. Gao and W. Bai, 2015. A particle method for two-phase flows with large density difference. Int. J. Numer. Methods Eng., 103: 235-255.

Natsui, S., H. Takai, T. Kumagai, T. Kikuchi and R.O. Suzuki, 2014. Stable mesh-free moving particle semi-implicit method for direct analysis of gas-liquid two-phase flow. Chem. Eng. Sci., 111: 286-298.

Ng, K.C. and E.Y.K. Ng, 2013. Laminar mixing performances of baffling, shaft eccentricity and unsteady mixing in a cylindrical vessel. Chem. Eng. Sci., 104: 960-974.

Ng, K.C., Y.L. Ng and W.H. Lam, 2013a. Particle simulation and flow sequence on drainage of liquid particles. Comput. Math. Appl., 66: 1437-1451.

Ng, K.C., E.Y.K. Ng and W.H. Lam, 2013b. Lagrangian simulation of steady and unsteady laminar mixing by plate impeller in a cylindrical vessel. Ind. Eng. Chem. Res., 52: 10004-10014.

Ng, K.C., Y.H. Hwang and T.W.H. Sheu, 2014. On the accuracy of laplacian models in MPS. Comput. Physics Commun., 185: 2412-2426.

Ng, K.C., Y.H. Hwang, T.W.H. Sheu and C.H. Yu, 2015. Moving Particle Level-Set (MPLS) method for incompressible multiphase flow computation. Comput. Physics Commun., 196: 317-334.

Xiang, H. and B. Chen, 2015. Simulating non-newtonian flows with the moving particle semi-implicit method with an SPH Kernel. Fluid Dynamics Res., 47: 27-27.