Modeling unsteady flow characteristics using smoothed particle hydrodynamics

S. Shahriari, I.G. Hassan, L. Kadem *

Department of Mechanical and Industrial Engineering, Concordia University, Montreal, Canada

A R T I C L E   I N F O

Article history:
Received 26 January 2011
Received in revised form 24 March 2012
Accepted 2 April 2012
Available online 16 April 2012

Keywords:
Smoothed particle hydrodynamics
Transient and oscillating flows
Viscosity formulations
XSPH variant
Pressure symmetric formulation
Wall shear stress

A B S T R A C T

Smoothed particle hydrodynamics (SPH) method has been extensively used to simulate unsteady free surface flows. The works dedicated to simulation of unsteady internal flows have been generally performed to study the transient start up of steady flows under constant driving forces and for low Reynolds number regimes. However, most of the fluid flow phenomena are unsteady by nature and at moderate to high Reynolds numbers. In this study, first a benchmark case (transient Poiseuille flow) is simulated to evaluate the ability of SPH to simulate internal transient flows at low and moderate Reynolds numbers \( (Re = 0.05, 500 and 1500) \). For this benchmark case, the performance of the two most commonly used formulations for viscous term modeling is investigated, as well as the effect of using the XSPH variant. Some points regarding using the symmetric form for pressure gradient modeling are also briefly discussed. Then, the application of SPH is extended to oscillating flows imposed by oscillating body force (Womersley type flow) and oscillating moving boundary (Stokes' second problem) at different frequencies and amplitudes. There is a very good agreement between SPH results and exact solution even if there is a large phase lag between the oscillating pressure difference and moving boundary and the movement of the SPH particles generated. Finally, a modified formulation for wall shear stress calculations is suggested and verified against exact solutions. In all presented cases, the spatial convergence analysis is performed.

© 2012 Elsevier Inc. All rights reserved.

1. Introduction

Traditional mesh-based numerical methods have been widely applied in different fields of science and engineering. The growing interest for simulating more complex problems has however highlighted some limitations associated with these methods. The most significant limitation comes from their mesh dependent nature. To overcome this limitation, a new generation of numerical methods called meshfree methods has been introduced. A class of such methods, called particle methods, considers fluid elements as particles interacting with each other while not connected in a similar way as in mesh-based methods. This characteristic results in an independence from mesh generation and remeshing processes especially in cases with complex geometries and deformable boundaries. Almost all meshfree particle methods share a similar fundamental concept, but they employ different approximation formulations and require specific implementation techniques. These methods are still under development and several improvements are still ongoing. The most widely used particle methods are: molecular dynamics, Monte Carlo, lattice Boltzmann, particle in cell and smoothed particle hydrodynamics [1]. Molecular dynamics method is an atomistic method and is still not enough accurate to reproduce the realistic dynamic of mesoscopic and macroscopic systems [2]. Implementation of Monte Carlo method, a probabilistic meshfree method, needs...
an iterative process based on proper random sampling techniques [3]. Lattice Boltzmann method uses a series of imaginary particles to model the fluid domain. These particles move through a background lattice mesh while the viscous flow behavior is modeled by inter-particle interaction forces [4]. In particle in cell method fluid particles are tracked in the domain while their physical properties are interpolated on a stationary mesh [5].

This paper focuses on smoothed particle hydrodynamics (SPH), developed originally to simulate compressible flows in astrophysics [6,7]. In comparison with other mesh-free methods, the specific characteristic of SPH is its fully Lagrangian meshfree nature. Indeed, it does not require a kind of background mesh to integrate the physical governing equations locally or in the entire domain of simulation. Although this method has passed its preliminary development stage, several studies are still ongoing to improve the numerical algorithm, discover new implementation features and verify its accuracy in widespread applications.

The first application of SPH to terrestrial problems was to free surface flow simulations [8]. Its spectrum of applications grew very rapidly to include wider fluid mechanic problems such as modeling compressible viscous flows [9], low Reynolds number incompressible flows [10], multiphase flows [11,12], interfacial flows [13], flows through porous media [14] and viscoelastic flows [15]. Indeed, most of the reported unsteady incompressible flow simulations using SPH are limited particularly to free surface problems such as dam breaking, shallow water and wave flumes (see extra issue of Journal of Hydraulic Research published in 2010 [16]) or transient internal flows at low Reynolds number [10,15,17]. Steady internal flows at higher Reynolds numbers were also as of interest in literature [18,19]. SPH method has been widely used in computational solid dynamics in problems such as: development of fracture and elastic behavior of solids with large deformation [20-24]. New approaches have also emerged to overcome the time step constraint and compressibility effect existing in classical SPH method [18,25].

In this study, we extend the application of SPH to simulate a variety of unsteady internal flows at moderate Reynolds number. More specifically, we will discuss some features of SPH method with emphasis on its application to simulate transient start up of steady flows and oscillating flows induced by variable applied body forces and moving boundaries.

The SPH method has proven to be capable of successfully simulating transient start up of internal steady flows due to constant driving forces for low Reynolds numbers. This typically applies to compressible flows with Re < 55 [9], incompressible flows with Re < 1 [10,17] and viscoelastic flows with Re < 2.5 [15]. Furthermore, an attempt has been made to explain the transition to turbulence in a start up Poiseuille flow in the presence of a streamwise magnetic field [26], but the comparison with an exact solution was not reported and the flow patterns were presented qualitatively.

Indeed, most fluid flow phenomena in science and engineering vary with time at discrete locations and occur through a wide range of Reynolds numbers. For example: (1) physiological flows are characterized by moderate Reynolds number and unsteadiness due to the oscillating driven forces and boundary movements; (2) in industrial piping systems and intake/exhaust ducts, the unsteadiness of pulsed pressure waves is an important topic having a direct influence on the mass flow rate and mechanical performance of mechanical systems; and (3) unsteady flow modeling is also challenging in new applications such as microelectromechanical systems (MEMS) in which delivering a certain volume of fluid is an important issue. This is accomplished through the use of different technologies involving time dependent flow nature in microchannels due to periodic driving pressure.

In this paper, first the transient behavior of Poiseuille flow is simulated for a range of Reynolds numbers up to 1500. For this benchmark case, the two most referred standard formulations to model viscous force suggested by Morris et al. [10] and by Monaghan [8] are examined. In addition, the effect of the XSPH variant, suggested to keep the movement of each particle consistent with the average velocity of its neighboring particles on particle distribution is also investigated. Then, the ability of SPH to simulate unsteady flows induced by an oscillating pressure difference (Womersley type flow) and oscillating moving boundary (Stokes’ second problem) with various frequencies and amplitudes is examined. Finally, an applied formulation is presented to approximate the wall shear stress in SPH. This formulation can also be used with other particle methods. Indeed, an accurate determination of wall shear stress is of primary importance for several applications and in particular for physiological flows where it is well correlated with damage to cells lining blood vessels [27].

2. Mathematical formulation and numerical aspects

2.1. SPH formulation

In SPH, a continuum medium is discretized as a set of number of particles. Each particle has its own physical properties such as pressure and velocity. Numerical discretization involves interpolating the value of a physical scalar or vector quantity (A) for a particle located at r, based on an interpolating kernel function (W), using the properties of its neighboring particles “b”, located at rₜ. This concept is interpreted numerically in standard form of [28]

\[ \langle A(r) \rangle = \sum_b m_b \frac{A_b}{\rho_b} W(r - r_b, h), \]  

where \( h \) is called smoothing length. Throughout this paper, “\( \langle \cdot \rangle \)” denotes SPH approximation of an arbitrary physical quantity. Using this equation and its derivatives, the governing equations of fluid flow (conservation of mass, momentum and energy) can be rewritten under the form of SPH formulation.
2.2. Conservation of mass

The conservation of mass for a fluid element, particle, in an incompressible flow is expressed as
\[ \mathbf{V} \cdot \mathbf{V} = 0. \]  

(2)

In SPH, the conservation of mass for a given particle “a” leads to the calculation of its density at its local position \((\mathbf{r}_a)\). The first SPH formulation for conservation of mass was derived directly from summation interpolant, Eq. (1), as
\[ \rho_a = \sum_b m_b \mathbf{W}_{ab}. \]  

(3)

here \(\rho_a = \rho(\mathbf{r}_a)\) and \(\mathbf{W}_{ab} = \mathbf{W}(\mathbf{r}_a - \mathbf{r}_b, h)\).

The time derivative form of the conservation of mass leads to [28]
\[ \frac{d\rho}{dt}_a = \sum_b m_b \mathbf{V}_{ab} \cdot \mathbf{V}_a \mathbf{W}_{ab}, \]  

(4)

here \(\mathbf{V}_{ab}\) is the gradient of the kernel function regarding the coordinates of given particle “a” and \(\mathbf{V}_{ab} = \mathbf{V}_a - \mathbf{V}_b\). Although Eq. (4) does not satisfy exactly the conservation of mass, it is employed in this study because some of its advantages compared with Eq. (3). Eq. (4) leads to variation in local density when neighboring particles move relative to each other [28] and ensures stability in density calculations. Also, Eq. (4) allows computation of density with other physical properties in the same subroutine [10,28],

2.3. Conservation of momentum

The equation of motion for a fluid element, particle, in an incompressible flow is
\[ \rho \frac{d\mathbf{V}}{dt} = \mathbf{f}_{body} - \nabla \mathbf{P} + (\mathbf{V} \cdot \nabla \mathbf{V}) \mathbf{V}, \]  

(5)

where \(\rho\) is fluid density, \(\mathbf{V}\) is velocity vector, \(\mathbf{f}_{body}\) is body force indicating the applied forces per unit volume of the fluid element due to the external fields, \(\mathbf{P}\) is thermodynamic pressure and \(\mu\) is dynamic viscosity.

To formulate the conservation of momentum under the SPH scheme, pressure and viscous terms need to be modeled. In literature, different forms for both terms have been suggested.

For the pressure, the symmetric form for the pressure gradient is the most frequently used as it satisfies the conservation of momentum. The most common type of symmetric form is expressed as [10,28]
\[ \left\langle \frac{\nabla \mathbf{P}}{\rho} \right\rangle_a = \sum_b m_b \left( \frac{p_b}{\rho_b^2} + \frac{p_a}{\rho_a^2} \right) \mathbf{V}_a \mathbf{W}_{ab}. \]  

(6)

In SPH, the pressure is related to the density by a quasi-incompressible equation of state. In this study, an equation of state is employed as [10]
\[ P = c^2 \rho, \]  

(7)

where, \(c\) is an artificial speed of sound. This equation has shown small sensitivity to density fluctuations. In terrestrial applications of fluid mechanics, the speed of sound is high compared to the bulk velocity of the fluid, therefore an artificial sound speed has to be employed to avoid very small computational time steps while keeping density fluctuations within an acceptable range (1–3%, to maintain incompressible flow behavior) [8,10].

At this point it should be mentioned that in experimental fluid mechanics, the thermodynamic pressure of a moving fluid is measured by a static pressure probe. By definition, “the static pressure is the pressure seen by the fluid particle as it moves (so it is something of a misnomer)” [29]. This definition is consistent with the Lagrangian motion of particles in SPH. However, in most of the SPH literature, \(P\) in Eq. (5) is referred to as “hydrostatic pressure” and in some works it is called the total pressure and expressed as the summation of dynamic and hydrostatic pressures in which the equation of state is mentioned to be used to model dynamic pressure [10,17,30]. It is then important to clarify that the pressure obtained through SPH simulations represents static pressure.

For the viscous term, the first expression proposed was based on the artificial viscosity used to limit non-physical post shock oscillations in the original SPH algorithm dealing with inviscid flows [28]. The resulting viscous force creates unrealistic high viscous effects. Then, this model will not be used in this study.

One of the standard forms used to model the viscous term is obtained from a combination of standard SPH and finite difference approximation of the first derivative. This formulation was originally presented to model the viscous term at very low Reynolds number flows [10]. This form is presented as
\[ \left\langle \frac{1}{\rho} \mathbf{V} \cdot \mu \mathbf{V} \right\rangle_a = \sum_b m_b \frac{\mu_a + \mu_b}{\rho_a \rho_b \mathbf{r}_{ab}} \mathbf{V}_a \mathbf{W}_{ab} \mathbf{V}_{ab}. \]  

(8)
This formulation (here called “Form I”) leads to conservation of linear momentum, while angular momentum is approximately conserved. In addition to being computationally efficient, it is also less influenced by computational errors in cases with low number of particles [10].

Another formulation used to model real viscous forces is [8,31]

\[
\left\langle \frac{1}{\rho} V \cdot \mu V \right\rangle \mathbf{V} = \sum_b m_b \frac{16}{\rho_a \rho_b} \frac{\mu_a \mu_b}{(\mu_a + \mu_b)} \frac{\mathbf{v}_{ab} \cdot \mathbf{r}_{ab}}{r_{ab}^2} \mathbf{v}_e W_{ab}.
\] (9)

In this paper, we refer to this standard formulation as “Form II”. This form was developed originally to simulate free surface flows and then has been widely used for other types of flow.

In this study, the accuracy of these two standard forms (“Form I” and “Form II”) is investigated under moderate Reynolds number flow conditions.

2.4. SPH particle movement and XSPH variant

Fluid particles move in a Lagrangian frame of reference based on

\[
\frac{d \mathbf{r}_a}{dt} = \mathbf{V}_a.
\] (10)

The new position of particles is derived by time integration of the velocity at each instant. In SPH literature, a variant, referred to as XSPH, is proposed with the goal of modifying and smoothing SPH particle movements based on the average velocity of their neighboring particles as [28]

\[
\frac{d \mathbf{r}_a}{dt} = \mathbf{V}_a + \epsilon \sum_b m_b \frac{(\mathbf{V}_b - \mathbf{V}_a)}{(\rho_a + \rho_b)^{1/2}} W_{ab}.
\] (11)

where \( \epsilon \) is a constant between 0 and 1.

The accuracy of both approaches for SPH particle movement (Eqs. (10) and (11)) will be investigated in this study.

2.5. Time integration and time step

A second order accurate time integration scheme, predictor–corrector algorithm, is used for time integration of flow governing equations. The predictor part consists of an explicit half time step integration followed by a corrector part to correct the approximated properties in the previous part. Finally, the properties at the new time step are calculated based on the values obtained at predictor and corrector steps.

For stability reasons, the time step should satisfy the criteria controlled by Courant–Friedrichs–Levy (CFL) condition, force and viscous diffusion terms [10].

2.6. The interpolating Kernel

A kernel function defines the interpolation form and determines the dimensional influence of the neighboring domain. The function should have several properties as being normalized and treated as a Dirac delta function when the smoothing length tends to zero [32]. The first applications of SPH used a Gaussian kernel. Since then, different forms of kernel functions have been proposed in the literature.

In this study, a quartic spline kernel for 2D simulations is used because of its high stability as [33],

\[
W_{ab} = \frac{96}{1199 \pi h^2} \begin{cases} 
(2.5 - q)^4 - 5(1.5 - q)^4 + 10(0.5 - q)^4 & 0 \leq q < 0.5, \\
(2.5 - q)^4 - 5(1.5 - q)^4 & 0.5 \leq q < 1.5, \\
(2.5 - q)^4 & 1.5 \leq q < 2.5, \\
0 & 2.5 \leq q,
\end{cases}
\] (12)

where \( q = \frac{r_{ab}}{h} \). In the quartic spline kernel, the influence dimension is 2.5 times the smoothing length.

2.7. Boundary condition

Wall boundary treatment in SPH requires special attention to prevent fluid particles from penetrating the walls and satisfying no slip condition. Different techniques have been developed to model wall boundaries in meshfree particle methods [9,10,25,34].

In our simulation, wall boundary conditions are implemented by placing a finite number of particles exactly on the wall boundaries and adding several layers of fixed imaginary particles parallel to the boundaries outside the domain. There is no need to create mirror particles at each time step as in Ref. [25]. These particles are placed in such a way to have the same spacing as the initial fluid particles. This ensures that fluid particles near the walls have an homogeneous distribution of
particles in their domain. This implementation is similar to the technique which uses a series of dummy particle layers on the walls with zero velocity [34]. In present study, the velocity of the wall boundary particles is set to zero. An extrapolated artificial velocity is allocated to the imaginary particles based on the velocity of fluid particles approaching the walls [10],

$$V_b = (1 - \beta)V_a$$  \hspace{1cm} (13)

and

$$\beta = \min \left( 1.5, 1 + \frac{d_b}{d_a} \right),$$  \hspace{1cm} (14)

where $V_b$ is the artificial velocity allocated to the imaginary particle “$b$”, $V_a$ is velocity of fluid particle “$a$” approaching the wall, $d_a$ is the normal distance of particle “$a$” to the wall and $d_b$ is the normal distance of particle “$b$” to the tangent line at the wall curvature in the direction of $d_a$.

The periodic boundary condition is applied at inlet and outlet boundaries to conserve the mass within the domain of interest.

In following, first the transient Poiseuille flow is simulated to investigate the accuracy of the two standard forms for viscous term modeling and the effect of the XSPH variant. The flexibility of SPH, as a particle meshfree method to handle the simulation of oscillating flows is then examined. For this purpose, two cases are investigated: internal flow driven by an oscillating pressure difference and flow above an oscillating plate. These cases will also be used to validate a suggested formulation for wall shear stress calculations.

3. Verification and validation: Poiseuille flow

The Poiseuille flow consists of the movement of a fluid between two infinite parallel plates similar to the flow in channels and pipes. There is no applied pressure difference in the vertical direction and the fluid is driven by an axial pressure difference resulting in a flow parallel to the plates. The fluid acceleration due to a directional pressure difference can be interpreted as the effect of an external body force. A body force causes acceleration, $a$, of a fluid element as

$$a = \frac{f_{\text{body}}}{\rho},$$  \hspace{1cm} (15)

here $f_{\text{body}}$ is the body force per unit volume of the fluid element. The force applied on a fluid element with the volume of $d\nu$ as a result of a pressure difference in $x$ direction is $-\frac{dP}{dx} d\nu$, so the force per unit of the mass becomes $-\frac{1}{\rho} \frac{dP}{dx}$.

In this study, the plates are located at $y = 0$ and $y = d$. The fluid starts to move from rest until it reaches steady state. Under such conditions, the analytical solution for the time dependent velocity is given by [10]

$$u(y, t) = \frac{f_x}{2\nu} y(y - d) + \sum_{n=0}^{\infty} \frac{4f_x d^2}{\pi^3 (2n + 1)^3} \sin \left( \frac{\pi y}{d} (2n + 1) \right) \exp \left( -\frac{(2n + 1)^2 \pi^2 \nu}{d^2} t \right),$$  \hspace{1cm} (16)

where $t$ stands for time, $u$ is the flow velocity in the $x$ direction ($V = u\hat{i}$) and $\nu$ is kinematic viscosity. The force applied per unit of the fluid mass, $f_x$, due to a pressure difference in $x$ direction is

$$f_x = -\frac{1}{\rho} \frac{dP}{dx} = \frac{8 \nu U_o}{d^2},$$  \hspace{1cm} (17)

here $U_o$ is the predefined maximum steady state velocity which occurs at a location equidistant from the plates. The physical properties are selected, for the simulations, as $d = 5 \times 10^{-3}$ (m), $\rho_o = 1000$ (kg/m$^3$) and $\nu = 1 \times 10^{-6}$ (m$^2$/s).

3.1. Viscous term formulation

We first examined the ability of the two standard formulations to model the viscous term (“Form I”, Eq. (8), and “Form II”, Eq. (9)) using this benchmark case. The maximum steady state velocity, $U_o$, is set to 0.1 (m/s) corresponding to a Reynolds number ($Re = U_o d / \nu$) equal to 500. The results for uniform particle spacing of $dx = dy = 1.0869565 \times 10^{-4}$ (m) (45 layers of particle in span of the plates) are shown in Fig. 1(a) and (b).

Using “Form I”, the results are more satisfactory, with a relative error of 0.47% for the peak velocity calculation at steady state. This error was 7.2% using “Form II”.

The spatial convergence rate in both simulation cases is evaluated based on $L_2$ relative error norm,

$$L_2 \text{ relative error norm} = \sqrt{\frac{\sum_{i=1}^{N}(u_{\text{SPH}}(i) - u_{\text{Exact}}(i))^2}{\sum_{i=1}^{N}u_{\text{Exact}}^2(i)}},$$  \hspace{1cm} (18)

where $u_{\text{SPH}}(i)$ and $u_{\text{Exact}}(i)$ are axial velocity of particle “$i$” simulated by SPH and predicted analytically, respectively. $N$ is the number of particles.
Fig. 2 presents the average $L_2$ relative error norm of the calculated velocity against the particle spacing. The error norm converges to a higher value when "Form II" is adopted.

The norm of velocity error converges with a rate close to one. The smoothing error in SPH has been proved to be second order [1], but several factors can affect the accuracy of SPH calculations [35-38]. The most important ones are the smoothing length, $h$, the ratio of particle spacing to smoothing length, $d_r h$, and the movement of particles with regards to their initial regular distribution [35].

In the literature, several approaches have been used to approximate the viscous term in momentum equation. Among them the "Form I" and "Form II" presented here are the most widely used formulations because of their robustness and simple implementation. In both formulations, the second derivative viscous fluxes are approximated based on combined finite difference and SPH derivatives.

The truncation error and the convergence properties of different schemes for first and second derivative discretization in SPH have been studied recently by Fatehi and Manzari [39] using theoretical analysis based on Taylor series expansion. A generalization of their work is used here to examine the discretization error of "Form I" and "Form II".

The general formulation of "Form I", Eq. (8), is simplified for constant $l$ across the fluid particles as follows,

$$
\left\langle \frac{\mu}{\rho} \mathbf{v} \cdot \mathbf{v} \right\rangle_a = \frac{\mu}{\rho_a} \sum_b 2 m_b \rho_b \mathbf{v}_{ab} \frac{\mathbf{r}_{ab} \cdot \mathbf{v}_{ab}}{r_{ab}^2},
$$

(19)
Considering the conservation of momentum in x direction (the flow under study is unidirectional: \( \mathbf{V} = U \hat{\mathbf{i}} \)) and substituting \( u_b \) with its Taylor series expansion around \( r_a \), results in
\[
\left( \frac{\mu}{\rho} \mathbf{V} \cdot \mathbf{V} \right) u_a = \left( \frac{\mu}{\rho} \mathbf{V} \cdot \mathbf{V} \right) u_a = \frac{\mu}{\rho_a} \sum b \frac{m_b}{\rho_b} \left( \mathbf{r}_{ab} \cdot \mathbf{V}_a u_a - \frac{1}{2} \mathbf{r}_{ab} \mathbf{r}_{ab} \cdot \mathbf{V}_a \mathbf{V}_a u_a + \frac{1}{6} \mathbf{r}_{ab} \mathbf{r}_{ab} \mathbf{r}_{ab} \cdot \mathbf{V}_a \mathbf{V}_a \mathbf{V}_a u_a + \cdots \right) \frac{\mathbf{r}_{ab} \cdot \mathbf{V}_a \mathbf{W}_{ab}}{r_{ab}^2},
\]
(20)
here, \( \mathbf{V}_a \) is the gradient regarding coordinates of particle \( a \) and ":" and "::" represent double and triple dot product operators.

The truncation error associated with SPH discretization of viscous term at each particle can be obtained as,
\[
E_a = \left( \frac{\mu}{\rho} \mathbf{V} \cdot \mathbf{V} \right) u_a - \left( \frac{\mu}{\rho} \mathbf{V} \cdot \mathbf{V} \right) a
\]
(21)
then the truncated error made in calculating viscous term by "Form I" is
\[
E_a = \frac{\mu}{\rho_a} \left( 2 \mathbf{V}_a u_a \sum b \frac{m_b}{\rho_b} \mathbf{V}_a \mathbf{W}_{ab} - \mathbf{V}_a \mathbf{W}_{ab} \right) - \left( \mathbf{r}_{ab} \cdot \mathbf{V}_a u_a \right) + \frac{1}{3} \mathbf{V}_a \mathbf{V}_a \mathbf{V}_a u_a \left( \sum b \frac{m_b}{\rho_b} \mathbf{r}_{ab} \mathbf{r}_{ab} \mathbf{V}_a \mathbf{W}_{ab} + \cdots \right),
\]
(22)
where \( I \) is unity tensor. The first and third terms inside of the parentheses are zero (refer to Eqs. (28) and (29) of Ref. [39]) and
\[
\sum b \frac{m_b}{\rho_b} \mathbf{r}_{ab} \mathbf{V}_a \mathbf{W}_{ab} + I \approx O \left( \left( \frac{dr}{h} \right)^{\beta+1} \right).
\]
(23)
Then, the order of magnitude of the leading term in the truncation error expression for "Form I" is
\[
\frac{\mu}{\rho} |\mathbf{V}^2 u_a| O \left( \left( \frac{dr}{h} \right)^{\beta+1} \right).
\]
(24)
The parameter \( \beta \) is the boundary smoothness of the kernel function and defined as the highest integer that the \( \beta \)th and all lower derivatives are zero at the edge of the kernel compact support [35]. The adopted spline kernel in the present study, Eq. (12), has \( \beta = 3 \).

The "Form II", Eq. (9), under assumption of constant \( \mu \) is presented as
\[
\left( \frac{\mu}{\rho} \mathbf{V} \cdot \mathbf{V} \right) = \frac{\mu}{\rho_a} \sum b \frac{m_b}{\rho_b} \mathbf{V}_a \mathbf{W}_{ab} \mathbf{V}_b \mathbf{V}_b.
\]
(25)
Applying the similar error analysis based on the Taylor series expansion of \( \mathbf{V}_b \) around \( r_a \) on this form leads to the order of magnitude of the leading term in the truncation error expression for "Form II" in x direction, as
\[
\frac{4 \mu}{\rho} |\mathbf{V}^2 u_a| O \left( \left( \frac{dr}{h} \right)^{\beta-1} \right).
\]
(26)
The assumption here is that the SPH particles have a regular distribution with constant viscosity and density. Both formulations have the same order of truncation error, but the error of "Form II" has a higher magnitude. The error is also dependent on the magnitude of the second derivative of the velocity at each particle, \( |\mathbf{V}^2 u_a| \). It means that the error grows in a nonlinear manner in transient start up state because of the accumulated errors in the calculation of the velocity (as can be seen in Fig. 1(b)). This magnifies the error especially for "Form II".

From another point of view, returning to the origin of "Form II" can more clearly explain the limited accuracy inherent to this formulation. The "Form II", is originally constructed based the following definition of viscosity,
\[
\mu = \frac{1}{8} g h c \rho,
\]
(27)
here the value of \( \alpha \) is near to 1 [40]. This form for viscosity was derived based on a comparison between a simple form of artificial viscosity and the continuum viscosity. The form for artificial viscosity was introduced based on the diffusion theory of gases in simulating shock waves [41]. Interestingly, in a series of numerical tests, we found that the accuracy of the results based on "Form II" is highly dependent on the smoothing length, \( h \), and the Reynolds number of the flow having direct effect on the adopted sound speed, \( c \), to control the compressibility effects. These results are consistent with the definition of viscosity in this form (Eq. (27)). However, this was not the case for "Form I" and the accuracy of the results did not show significant dependency on the value of the smoothing length and flow characteristics.

It should be noted that Cleary [31] proposed and applied a calibration coefficient for "Form II" based on a series of numerical tests performed on the case of time dependent Couette flow. Interestingly, the proposed coefficient needs to be adjusted in relation to the flow characteristics. This is also the case for a new suggested formulation for modeling viscous force in shear flows which possesses a dependent coefficient [42]. The detailed investigation of these issues is beyond the scope of this paper.

The test conditions are then extended to different Reynolds numbers (\( Re = 0.05 \) and \( Re = 1500 \)) using "Form I" for viscosity formulation. The results are shown on Fig. 3. For all cases, there is a very good agreement between the numerical simulations
and the analytical solution. For the case of $Re = 1500$, the maximum relative error in calculation of peak velocity in transient state is 0.49%. The average relative error over all the particles in the domain at steady state is 0.68%.

It should be noted that the viscosity formulation in "Form I" was presented to simulate very low Reynolds number flows [10], however, it seems capable also to accurately simulate a wider range of Reynolds numbers for such unsteady flow configuration.

Fig. 4 presents the evolution of the maximum flow velocity (at the centerline between the two plates) and the maximum changing in density of the particles with respect to the initial reference density. The density remained constant and did not show large fluctuations for both Reynolds number cases ($Re = 0.05$ and $Re = 1500$). The maximum relative variations in density are 0.000056% and 0.000126% during the simulation for $Re=0.5$ and $Re = 1500$ respectively, which is within the acceptable range for assuming incompressible flow behavior.

3.2. XSPH variant

The second test using this benchmark case is to investigate the effect of applying XSPH variant to particle movements (Eq. (11)). This variant was proposed with the goal of preventing particles from penetrating each other when simulating impinging streams of fluid [43]. It has also been reported to be capable of keeping the particles orderly during simulations of nearly incompressible fluids in the absence of viscosity [28]. This variant has been employed in literature for different simulation cases, without a prior validation. Fig. 5 shows the effect of XSPH variant ($\varepsilon = 0.3$) on particle order beyond steady state, compared to the classical formulation for particle movement (Eq. (10)). Particles are initially positioned with a spacing of $2.08333 \times 10^{-4}$ (m). The fluid properties are the same as previously, Reynolds number is equal to 500 and the standard viscosity formulation in "Form I" is employed. In our simulation cases, the XSPH variant creates disorder in particle movements especially in the regions subjected to high velocity gradients.

---

**Fig. 3.** Velocity profiles for Poiseuille flow at different start up instants to reach steady state: (a) $Re = 0.05$; and (b) $Re = 1500$.

**Fig. 4.** The Poiseuille flow characteristics: (a) maximum flow velocity with respect to time for $Re = 1500$ case; and (b) maximum particle density changes versus time for $Re = 1500$ and $Re = 0.05$ cases.
The oscillation in inter-particle spacing along the axis of the plates and $L_2$ relative error norm of velocity for the first layer of the particles near to the plates are demonstrated in Fig. 6 at steady state with respect to the initial particle spacing. The rate of spatial convergence is close to the first order.

As it is shown in Fig. 6(a), the undesirable effect of XSPH variant can be minimized by increasing the number of particles, but this will lead to higher computational cost. In comparison, not using the XSPH variant leaded to accurate results and more ordered particle distribution.

The effect of different numerical parameters such as precision of calculations, time step, smoothing length and sound speed were also examined and the main consequence was just delaying or speeding up the onset of particle disordered distribution. Based on this study, XSPH should be used with caution in the presence of real viscosity, especially when the flow is subjected to high velocity gradients and particle resolution is limited. These results regarding the XSPH are consistent with the observation of Imaeda and Inutsuka [44] on the effect of this variant on shear flows in astrophysical systems.

3.3. Pressure gradient term formulation

In our study, the standard symmetric formulation (Eq. (6)) for pressure gradient is always employed as it conserves exactly the linear and angular momentum (the calculated pressure force on particle “a” due to its neighboring particle, “b”, is equal to the applied force on particle “b” caused by particle “a”).

Ma and Ge [30] examined the computational performance of standard symmetric formulation for the pressure gradient term compared to asymmetric one. They reported that the symmetric formulation is less accurate, more sensitive to the selected value for sound speed, and leads to larger errors. Their conclusion was based on simulation results with large fluctuations in flow variables. In the case of Poiseuille flow, they reported a noisy mean flow velocity and pressure terms for values of $c^2 = 1 \times 10^{-2}$, $2 \times 10^{-5}$, $5 \times 10^{-5}$ (m$^2$/s$^2$). These values correspond to sound speeds of 25, 358, and 566 times of the peak velocity magnitude at steady state ($1.25 \times 10^{-5}$ (m/s) for $Re = 1.25 \times 10^{-2}$), respectively. The error in their simulations was
about 3% at the peak of the velocity profile at steady state for both symmetric and asymmetric pressure formulations. In our present simulation, this error was 0.49% for \( Re = 1500 \) (the worst case in our simulations) using the same formulation for the viscous term modeling.

This discrepancy between our results and Ma and Ge results can be explained by the fact that the time step \( (10^{-4} \text{ s}) \) reported in their study is not appropriate for simulations with very high sound speeds due to the CFL stability conditions \[10\].

Fig. 7 shows the mean flow velocity calculated in our study, using the standard symmetric formulation for pressure gradient, considering all moving particles in the domain. The geometry and flow characteristics are selected the same as \[30\]: \( d = 1 \times 10^{-3} \text{ (m)}, \rho_o = 1000 \text{ (kg/m}^3\text{)}, \nu = 1 \times 10^{-6} \text{ (m}^2\text{/s)}, \text{Re} = 1.25 \times 10^{-2}, 50 \) particles in span of the plates and the worst value for sound speed \( c^2 = 5 \times 10^{-5} \text{ (m}^2\text{/s}^2\text{)}, \) a value 566 times greater than the peak velocity magnitude. Our results did not show large fluctuations as it is reported in Ref. \[30\].

The performance of the two formulations for viscous term modeling and the effect of using the XSPH variant were also examined using the benchmark case of plane Couette flow. Similar conclusions were obtained.

In the following sections, the flexibility of SPH as a meshfree particle method to handle simulation of oscillating flows is examined. Here, two cases: internal flow driven by an oscillating pressure difference and flow above an oscillating plate are studied. As the performance of the viscosity formulation in “Form I” was verified in the previous simulation case for a variety of Reynolds numbers, this form is employed here and its performance is examined in simulating cases where there is no simple relationship between the viscous and body forces applied. In addition, the XSPH variant is not considered since as shown, this variant might not be adequate for these types of shear flows with elevated velocity gradients.

4. Oscillating flow cases

4.1. Internal flow driven by oscillating pressure difference

This case is similar to Poiseuille flow except that the pressure difference does not remain constant but instead varies with time. This type of flow is commonly found in large arteries and in industrial piping systems due to changes in direction of pressure waves.

The flow between two plates oscillates due to an axial oscillating pressure difference as,

\[
\frac{dP}{dx} = -A \cos(\omega t),
\]

(28)

here \( A \) is the amplitude of the imposed pressure difference, \( \omega \) is the oscillation frequency \( (\omega = 2\pi/T, T \) is the period of oscillation) and \( t \) stands for time. The analytical solution for velocity in this case is found as \[45\]

\[
u(y, t) = \frac{A}{\omega \rho \gamma} \left( [\sin \Phi_1(y) \cdot \sin \Phi_2(y) + \sinh \Phi_2(y) \cdot \sin \Phi_1(y)] \cos(\omega t) + [\gamma - \cosh \Phi_1(y) \cdot \cos \Phi_2(y) - \cosh \Phi_2(y)] \cdot \cos \Phi_1(y) \sin(\omega t) \right),
\]

(29)

where

\[
\Phi_1(y) = \frac{W_o}{\sqrt{2}} \left( 1 + \frac{2y}{d} \right), \quad \Phi_2(y) = \frac{W_o}{\sqrt{2}} \left( 1 - \frac{2y}{d} \right)
\]

(30a)
and
\[ \gamma = \cosh(\sqrt{2}W_o) + \cos(\sqrt{2}W_o), \]  
(30b)

here \( d \) is the distance between plates and \( W_o \), called the Womersley number, is an important dimensionless number in oscillating internal flows that represents the ratio of oscillating inertial effects to viscous effects and is defined as
\[ W_o = \frac{d}{T} \left( \frac{\rho}{\mu} \right)^{1/2}. \]  
(31)

Fig. 8(a) and (b) shows the simulation results for a Womersley number of \( W_o = 1 \) and oscillating pressure difference amplitudes of 0.09 (N/m²) and 90 (N/m²). The fluid physical properties are selected similar to those in the previous cases. The particle initial spacing is \( 1.086956 \times 10^{-4} \) (m) and smoothing length is selected 1.25 times of the initial particle spacing. The relative error in calculated velocities at the centerline between the plates has an average value of 0.31% and a maximum value of 0.69% during computation.

For a Womersley number equal to 1, viscous forces dominate oscillating inertial forces and the velocity field shows Poiseuille type profiles. The highest velocity occurs at the centerline between the two plates.

The simulation results for a larger Womersley number of \( W_o = 10 \) and oscillating pressure difference amplitudes of 0.3 (N/m³) and 3000 (N/m³) are shown in Fig. 9(a) and (b). The particle initial spacing is \( 1.086956 \times 10^{-4} \) (m) and the same physical properties as before are employed for the fluid. The relative error in calculated velocity at the centerline has an average value of 0.31% and a maximum value of about 0.7%. For Womersley number equal to 10, the inertial forces are stronger than viscous forces and the velocity profiles become flat in the central region and the highest velocity occurs in the regions within the vicinity of the plate.

The spatial convergence is analyzed using \( L_2 \) relative error norm criteria and depicted in Fig. 10. The results show a rate of convergence between first and second order.

The maximum variation in particle density occurred in cases with high Womersley number and amplitude of pressure difference. The maximum relative variation in density for the cases with \( W_o = 1 \) is 0.0029%, occurs when \( A = 90 \) (N/m³), and for the cases with \( W_o = 10 \) is 0.034%, occurs when \( A = 3000 \) (N/m³). These values are perfectly within the acceptable range.

In cases with high Womersley number, the simulations using viscous formulation in “Form II”, which was less accurate for Poiseuille flow, shows a good agreement with the analytical solution in the central region. This is, however, not surprising since, as shown in Fig. 9, the velocity gradient is almost zero in this region.

Another important characteristic to investigate for such oscillatory flows is the phase shift between the applied pressure difference and the resulting velocity. Figs. 11(a) and (b) shows the applied oscillating pressure difference and the resulting oscillating velocity at the centerline between the plates.

SPH particles were able to accurately follow the rapid changes in the acceleration and deceleration phases. The simulations then accurately reproduce the phase shift between the applied pressure difference and the velocity. At instants where the driving pressure difference becomes zero, SPH particles still move properly dependent upon their acquired energy from previous instants.

In practice, the total volume flow rate is a more important parameter than velocity, because it determines the time that fluid resides in the region of interest. It also can be a valuable indicator to examine the accuracy of the SPH incompressibility assumption and velocity calculations with respect to particle resolution. When using particle methods, we can approximate the volume flow rate by a summation over the particles located in a strip across the flow cross section with the width of particle spacing. In the present simulation, this gives

\[ Q = \sum \frac{4}{\pi} \frac{1}{n} \rho \nu \sin \theta \left( u \cos \theta - \frac{\partial w}{\partial y} \right), \]
where $m$ indicates the number of particle layers extended across two plates and $dy$ is the width of the stream channel that a particle flows through (particle spacing).

The time variation of the calculated volume flow rate is compared with the exact solution in Fig. 12 for a case with $W_o = 10$ and $A = 3000$ (N/m^3). The results are in a very good agreement with analytical solution with an average relative error of 0.6% over one period.

4.2. Flow above an oscillating plate

In literature, this problem is referred to as Stokes’ second problem. A stationary fluid above a plate starts to move due to oscillation of a plate. The numerical simulation of this problem is of great interest and there is a wide practical application, such as the study of the mechanical behavior of blood cells under oscillating shear stress (microscale), controlling the coating thickness in oscillating film flows and the investigation of the hydrodynamic loads produced by structural vibration which is an important design factor in mechanical systems (macroscale).

This case is characterized by a varying phase lag between the velocities of different layers of particles above the plate. The plate moves in $x$ direction with a velocity of $U_0(t) = U_{0,m} \cos \omega t$, where $U_{0,m}$ is the amplitude of the velocity oscillation and $\omega$ is the oscillation or angular frequency ($\omega = 2\pi/T$, $T$ is the period of oscillation). The analytical solution for this problem is obtained as, [46]
The oscillating velocity of the flow is damped as \( y \) increases (\( \eta \rightarrow \infty \) leads to \( u \rightarrow 0 \)).

The instantaneous velocity profiles over one period for the plate velocity amplitude of \( U_{0,m} = 2 \times 10^{-5} \) (m/s) and 0.3 (m/s) and oscillation frequency of \( \omega = \pi \) (rad/s) and 10\( \pi \) (rad/s) are shown in Figs. 13 and 14. The fluid physical properties are selected similar to previous cases. Particles are initially spaced \( 1 \times 10^{-4} \) (m) and \( 5 \times 10^{-5} \) (m) apart for \( \omega = \pi \) (rad/s) and 10\( \pi \) (rad/s) cases, respectively.

The maximum relative variation in density is 0.078% occurring when \( \omega = \pi \) (rad/s) for cases with a velocity amplitude of \( 2 \times 10^{-5} \) (m/s). For cases with velocity amplitude of 0.3 (m/s) the maximum variation in density is 0.000047%, also occurring when \( \omega = \pi \) (rad/s). These results are perfectly within the acceptable range. The maximum density variation occurred for cases with low oscillation frequency.

SPH was capable to capture the oscillating boundary layer region. This case has the most complex oscillating flow features since different layers of fluid experience different phase lags with regard to plate oscillations. SPH particles were able to follow these oscillations having a good agreement with the analytical solution. The particles far from the plate remained at rest as predicted by the analytical solution even for high magnitude and frequency of the oscillations.

The highest error in velocity calculations for the layer of particles near to the oscillating plate (subjected to the most elevated velocity gradient) occurs at \( t = \pi T/4 \) and \( t = 3\pi T/4 \). These instants correspond to a change in direction of the oscillating
plate. For higher oscillation frequency cases (Fig. 14), the average relative error in velocity calculations for the layer of particles near to the plate is 1.54% (excluding $t = T/4$ and $3T/4$) and the largest error is 5.5% (at $t = T/4$ and $t = 3T/4$), however, for lower oscillation frequency cases (Fig. 13), the average error is 0.97% and the largest one is 4.9%. Although the cases with higher oscillation frequency (Fig. 14) have particle spacing two times smaller than those with lower frequency (Fig. 13), there is a higher average relative error in comparison with the theoretical solution.

The convergence examined using $L_2$ relative error norm is shown in Fig. 15. The velocity profiles converge with a rate more similar to first order. Interestingly, the cases with the same oscillation frequency show quite similar spatial convergence behavior.

A higher oscillation frequency limits the wave propagation and results in a thinner oscillating boundary layer subject to a higher velocity gradient. As a consequence, we suggest for high oscillation frequency cases to select a high particle resolution with respect to the thickness of the oscillating layer.

A good estimate for the particle resolution in the oscillating layer can be obtained knowing the thickness of this layer which is approximated as [46]

$$\delta \approx 6.5 \sqrt{\frac{v}{\omega}} \quad (\text{where } h = 0.01 \times U_{o,m}).$$

(34)

We tried to determine the optimal number of particle layers in the oscillating layer to minimize the error in velocity calculations. Fig. 16 shows the variation of the relative error in the velocity calculations for the layer of particles near to the oscillating plate as a function of number of particle layers, $N_\delta$, across the oscillating thickness.
To reduce the average relative error to 1% throughout the period, the optimal number of particles in the oscillating layer should be 40–50. However, it maintains the maximal relative error at 4–5% for the most critical instants \( t = T/4 \) and \( 3T/4 \) while having reasonable computational time step. Consequently, in simulating oscillating flow problems, we suggest selecting the particle spacing with regards to the thickness of the oscillating layer, not to the length scale of the flow domain. This means that using the same particle spacing for simulations with different frequencies is not recommended. A good initial value for particle spacing close to the wall boundaries can be obtained by dividing the estimated oscillating layer thickness, Eq. (34), by 40. To reduce the computational cost, a variable particle mass distribution technique can be accomplished with fine resolution in the oscillating layer thickness and coarse resolution far away from this region [47].

4.3. Oscillating wall shear stress

An important parameter to take into consideration for oscillating flows is the wall shear stress. Wall shear stress plays a significant role in natural and industrial phenomena including, the development of cardiovascular disease, mainly cholesterol deposition in arteries [48], erosion in mechanical systems [49] and membrane based filtration processes [50].

In laminar Newtonian flows, the wall shear stress, \( \tau_w \), is proportional to the local tangential velocity gradient with respect to normal distance from the wall, irrespective of wall shape. This is expressed as
\[ \tau_w = -\mu \left( \frac{dV_t}{dx_n} \right)_w, \]  

where \( x_n \) is the local coordinate normal to the wall and \( V_t \) is the tangential local velocity with respect to the wall.

For flat wall boundaries, this expression can simply be determined using a finite difference scheme, which requires knowing the velocity of one or two layers of particles adjacent to the wall. However, in meshfree particle methods using this scheme is limited and inappropriate. In this section, we examine the possibility of developing a general formulation for the calculation of wall shear stress.

There are several possible ways in SPH to approximate the gradient of a quantity. Using the summation approximation of Eq. (1), the gradient of the tangential velocity with respect to the normal coordinate of the wall can be derived and the wall shear stress becomes

\[ \langle \tau_w \rangle_{\text{wall}} = -\mu \sum_b m_b \frac{V_{t,b}}{\rho_b} \frac{\partial W(r_a - r_b, h)}{\partial x_n}, \]  

here

\[ \frac{\partial W(r_a - r_b, h)}{\partial x_n} = \frac{(x_{n,a} - x_{n,b})}{|r_a - r_b|} \frac{dW}{dr}. \]

Based on the general expression of

\[ \rho V A = \nabla (\rho A) - A \nabla \rho \]

and using the SPH standard approximation of Eq. (1), we can get

\[ \langle \tau_w \rangle_{\text{wall}} = -\frac{\mu}{\rho_n} \sum_b m_b (V_{t,b} - V_{t,a}) \frac{\partial W(r_a - r_b, h)}{\partial x_n}. \]

For oscillatory flows, analyzing instantaneous variations, rather than averaged values, of wall shear stress is of primary importance. In the SPH algorithm, it is difficult to allocate the appropriate velocity, based on Eq. (13), to the imaginary particles in the neighboring domain of a wall particle, so to be consistent with the exact velocity gradient corresponding to the wall particle. Although higher order extrapolation techniques can be adopted to determine the artificial velocity for imaginary particles in the neighboring domain of a wall particle, so to be consistent with the exact velocity gradient corresponding to the wall particle, these techniques are computationally expensive. Therefore, this constraint should be taken into account when proposing a general formulation.

Moreover, some SPH implementations do not use imaginary particles for simulation of wall boundaries. In fact, in this study, imaginary and wall particles are not used to calculate the summation of wall shear stress, even though the wall particles are solved at each time step and their density is updated.

Consequently only a limited number of particles will contribute to the summation in Eqs. (36) and (39) (kernel is truncated). One can employ a methodology based on the kernel function modification [51]. However, this can be computationally expensive and we prefer to implement an approach based on a general correction coefficient.

The proposed approach shows its strength when computing the wall shear stress on the surface of thin structural bodies immersed in a fluid. The neighboring domain of a particle on the surface consists of particles on other sides of the immersed body, however only the fluid particles on the same side of the surface should contribute to the summation of the particle’s properties.

Eqs. (36) and (39) can be modified by multiplying their right hand side by a correction coefficient, \( C_n \), without modification to the kernel function, resulting in

\[ \langle \tau_w \rangle_{\text{wall}} = -\mu \sum_b m_b C_n \frac{V_{t,b}}{\rho_b} \frac{\partial W(r_a - r_b, h)}{\partial x_n}, \]  

and

\[ \langle \tau_w \rangle_{\text{wall}} = -\frac{\mu}{\rho_n} \sum_b m_b C_n (V_{t,b} - V_{t,a}) \frac{\partial W(r_a - r_b, h)}{\partial x_n}. \]

By employing a quartic spline kernel (influence domain of \( 2.5 \times h \)) with \( h \) equals to 1.25 times the initial particle spacing, each fluid particle would have nearly 29 particles (including itself) in its neighboring domain (Fig. 17(a)). However, to calculate wall shear stress, a particle on the wall will only have about 12 particles in its neighboring domain (Fig. 17(b)). To compensate for the kernel truncation on the wall boundary, the correction coefficient of \( C_n = 2.4 \) is calculated based on the area (volume in three dimensional simulations) covered by the kernel function divided by the area of continuum domain represented by the particles in a truncated kernel.

Employing different kernel functions and smoothing lengths will result in different numbers of particles in the neighboring domain of wall particles. However, the correction coefficient will remain almost constant. Using a kernel function with an influence domain of \( 2.5 \times h \) (quartic spline kernel) and having smoothing length of \( h = 1 \) and 1.5 times the initial particle spacing results in a correction coefficient of \( C_n = 2.5 \) and \( C_n = 2.3 \), respectively. The values of correction coefficient for
different kernel functions and smoothing lengths are provided in Fig. 18. To generalize the correction coefficient, the averaged value of 2.4 can be used for all cases.
Fig. 19 shows the wall shear stress variation on the lower plate for flow driven by an oscillating pressure difference (see Section 4.1) using general Eqs. (36) and (39) and modified Eqs. (40) and (41) with the same particle resolution \((dy = 3.20513 \times 10^{-5} \text{ m})\). The results are compared with the analytical solution and show the capability of each equation to reach the accurate analytical prediction.

The spatial convergence of different formulations to calculate wall shear stress is depicted in Fig. 20 using \(L_2\)-norm of relative error. The error norm converges to a high value when Eqs. (36) and (39) are used. The error norm of wall shear stress converges with a rate between first and second order for Eqs. (40) and (41).

Using the same particle resolution as used to plot velocity profiles in Fig. 9(b), \(1.086956 \times 10^{-4} \text{ (m)}\), does not give accurate results for the calculation of wall shear stress. Indeed, it is a common problem in mesh-based and meshfree computational methods to accurately compute the gradients of physical properties near the boundaries. In such cases, a high resolution mesh or particle distribution is needed.

In the above simulation, the wall boundary is stationary \((V_{ta} = 0)\) and both Eqs. (40) and (41) gave the same value for the wall shear stress. The potential of the proposed formulations (Eqs. (40) and (41)) to accurately predict the wall shear stress can be investigated more precisely in the case of an oscillating wall boundary. Fig. 21 shows the variation of wall shear stress over one period using different formulations compared to the analytical solution for the case of a flow over an oscillating plate with a particle spacing of \(dy = 2.2388 \times 10^{-5} \text{ (m)}\).

Eqs. (39) and (41) are much more compatible with no-slip boundary condition on the walls, a condition which results in a relative velocity of flow with respect to the wall equals zero. In addition, a constant tangential velocity results in a zero gradient of this parameter. This condition is also satisfied by Eqs. (39) and (41), but not by Eqs. (36) and (40).

Eqs. (36) and (40) failed completely to provide correct results and using Eq. (39) did not result in accurate estimates for wall shear stress calculations. In Fig. 22, the \(L_2\) error norm of wall shear stress is plotted against the particle spacing for Eqs.
(39) and (41). Similar to previous simulations, the error norm converges to a higher value when Eq. (39) is used. The norm of wall shear stress error converges at a rate close to unity for Eq. (41).

The proposed formulation (Eq. (41)) can be employed regardless of the curvature of the wall at the local region of interest. However, to achieve the desired accuracy for a larger curvature of the wall, a smaller smoothing length, $h$, or higher particle resolution is required in order to make the curved wall resemble a flat wall region. These constraints are similar for all computational methods.

5. Conclusion

This study investigated the ability of the smoothed particle hydrodynamics (SPH) method to simulate internal transient and oscillating flows at different flow characteristics. The selected test cases were chosen to ensure the possibility of precise comparison between SPH results and analytical solutions.

The main conclusions of this work can be summarized as: (1) “Form I” for modeling the viscous term in momentum equation showed more accuracy in our simulation cases, and unlike “Form II” it is not dependent on the type of kernel function and flow characteristics; (2) using the XSPH variant, especially when particle resolution is not high and flow is subjected to high velocity gradients, can generate undesirable particle disorders; (3) the SPH method is able to correctly simulate internal oscillating flows even when there is a large phase lag between the oscillation of the derived velocity and applied pressure difference and moving boundary; (4) all the oscillating flow simulations using SPH showed the same convergence rate as for transient Poiseuille flow type simulations; and (5) a modified formulation for wall shear stress calculations was presented and verified against exact solutions.

All simulated cases did not show unacceptable compressibility effects. This is despite that in some cases fluid particles experienced rapid changes in acceleration magnitude and direction, and were subjected to higher inertial effects (higher Reynolds numbers) when compared to previous studies.

Finally, the two oscillating flow cases studied in this paper can be of interest for research communities in meshfree particle methods to be considered as benchmark cases, as they include complex flow behavior.

References
