XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

Numerical implementation of the Stokes-Couette flow using the SPH method

Charleston J. S. Roice Filho[1], José Fernandes Júnior[2], Joel R. G. Vasco[2]

[1]*School of Civil and Environmental Engineering, Federal University of Goiás*
Av. Universitária, 1488, 74605-000, Goiás/Goiânia, Brazil
cjsroice@gmail.com
[2]*Graduate Program in Environmental and Sanitary Engineering, Federal University of Goiás*
Av. Universitária, 1488, 74605-000, Goiás/Goiânia, Brazil
jfslmb@gmail.com, joelvasco@ufg.br

Abstract. This article uses the Lagrangian numerical method SPH (Smoothed Particle Hydrodynamics) to solve the Stokes-Couette flow. The laminar flow is created by horizontal oscillation of the lower plate while the upper plate remains stationary. Stokes-Couette flow can be used as a benchmark to test codes using the meshfree SPH method since this phenomenon is governed by the one-dimensional classical heat equation. The newtonian fluid domain between the flat plates is discretized by particles in a vertical column, which would be free to move according to the forces acting on the particles. Boundary conditions are treated with ghost particles, created outside the physical domain to keep the number of neighbors of each particle constant. The initial condition considers the fluid at rest. The heat equation is then solved for each particle inside the domain and the horizontal velocity profile is obtained from the integration over time. The variation in time of the velocity profile from the oscillatory point of view (non-zero oscillation frequency) and the evolution in time of the velocity profile (zero oscillation frequency) are evaluated. The numerical solution of the SPH method reproduces the Stokes-Couette flow, which can be used as a benchmark for more complex SPH-based codes.

Keywords: Numerical implementation, SPH method, Stokes-Couette flow

1 Introduction

Numerical methods are useful tools for researchers and scientists, solving problems for which analytical solution is difficult to obtain or unavailable. A part of traditional numerical methods applied to problems in fluid dynamics, such as the Finite Difference Method (FDM), is based on an Eulerian approach to the equations of mass, linear momentum and energy [1]. In addition, such numerical methods require a mesh to discretize the fluid domain.

The SPH (Smoothed Particle Hydrodynamics) method does not seem to be suitable for solving simple problems when compared to traditional numerical methods, even when the conservation equations are written in the Lagrangian approach. However, the SPH has several advantages, notably associated with the simulation of free surface flows, such as the representation of large displacement phenomena, such as breaking waves; the absence of numerical diffusion due to the discretization of the convective term of the linear momentum equation; and disregarding the boundary condition on the free surface, or even its detection. The SPH has negative points, such as the treatment of boundary conditions close to solid borders [2] and the efficient search of the neighborhood of the particles [3]. These difficulties may prevent new users from studying the SPH method.

Thus, this article elucidate the main points in the implementation of a SPH code using the Stokes-Couette flow as a study case.

2 Stokes-Couette flow

The one-dimensional Stokes-Couette flow is generated by the oscillation of a flat plate. A newtonian fluid is considered between two flat, parallel, horizontal and infinite plates, separated by a distance H, with the lower plate

oscillating at a constant frequency ω, while the upper plate remains at rest. Figure 1a illustrates the flow described.

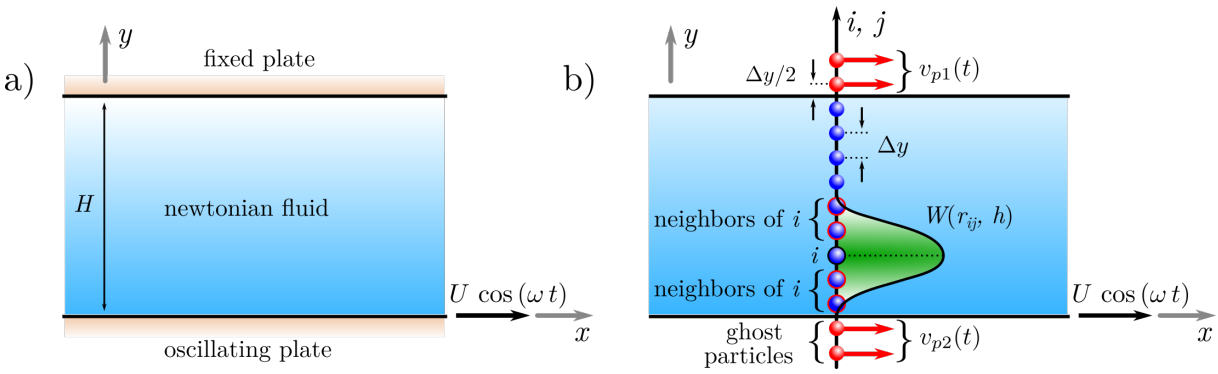


Figure 1. a) Description of the Stokes-Couette flow and b) distribuition of the particles in the y direction and illustration of the neighborhood of the particle i

The application of the Navier-Stokes equation for the Stokes-Couette flow in the x direction, disregarding the pressure gradient $\partial p/\partial x$, results in:

$$\frac{\partial v_x}{\partial t} = \nu \frac{\partial^2 v_x}{\partial y^2}, \tag{1}$$

where v_x is the linear velocity in the x direction [m/s] and ν is the kinematic viscosity of the fluid [m^2/s]. The boundary conditions of eq. (1) are $v_x(0,t) = U\ \cos(\omega\, t)$ and $v_x(H,t) = 0$ and the initial condition is $v_x(y,0) = 0$.

If $\omega \neq 0$, the analytical solution of eq. (1) is given by:

$$\begin{aligned} v_x(y,t) = \frac{U}{2\,(\cosh(2\,\lambda\,H) - \cos(2\,\lambda\,H))} \left[\exp\left(-\lambda\,(y-2\,H)\right)\cos(\omega\,t - \lambda\,y) + \ldots\right. \\ \cdots + \exp\left(\lambda\,(y-2\,H)\right)\cos(\omega\,t + \lambda\,y) - \exp\left(-\lambda\,y\right)\cos(\omega\,t - \lambda\,y + 2\,\lambda\,H) - \ldots \\ \left.\cdots - \exp\left(\lambda\,y\right)\cos(\omega\,t + \lambda\,y - 2\,\lambda\,H)\right], \end{aligned} \tag{2}$$

where U is the maximum amplitude of the velocity of the oscillating plate [m/s] and $\lambda = \sqrt{\omega/(2\,\nu)}$. The analytical solution of eq. (1) for $\omega = 0$ is:

$$v_x(y,t) = \frac{U\,(H-y)}{H} - \sum_{n=1}^{\infty} \frac{2\,U}{n\,\pi}\ \sin\left(\frac{n\,\pi\,y}{H}\right)\ \exp\left(-\frac{\nu\,n^2\,\pi^2\,t}{H^2}\right). \tag{3}$$

Equation (2) e (3) are used to compare the analytical solution with the numerical one, presented by the implemented SPH method, described in the section 3.

3 SPH

The SPH method approximates any field $\phi(\vec{r})$ by a convolution of $\phi(\vec{r})$ with the Dirac delta function. The SPH method replace the Dirac delta function with a kernel, which has some characteristics of the Dirac delta function such as the integral over the entire domain equal to one [4]. The kernel W is a scalar function and depends on the relative distance between particles and on the smoothing length h. Therefore, the notation $W = W(|y_i - y_j|, h) = W(r_{ij}, h) = W_{ij}$ is used, where y_i is the vertical position of the particle i.

Figure 1b illustrates the value of the kernel for the neighboring j particles of the i particle through the color gradient. The kernel used is given by the one-dimensional cubic spline [5], given by:

$$W(s) = \frac{2}{3\,h} \begin{cases} 1 - \frac{3}{2}s^2 + \frac{3}{4}s^3, & \text{if } s < 1, \\ \frac{1}{4}(2-s)^3, & \text{if } 1 \leq s < 2, \\ 0, & \text{if } s \geq 2, \end{cases} \tag{4}$$

with $s = r_{ij}/h$.

Written according to the SPH method, eq. (1) is:

$$\frac{\mathrm{d}\, v_{x,\,i}}{\mathrm{d}t} = \sum_{j \neq i} \frac{2\, m_j\, \nu}{\rho_j} \frac{v_{x,\,ij}\, y_{ij}}{r_{ij}^2 + (0,01\, h)^2} \frac{dW_{ij}}{dr}, \tag{5}$$

where m_j represents the mass of the particle j [kg], ρ_j is the density of the particle j [kg/m], $v_{x,\,ij} = v_{x,\,i} - v_{x,\,j}$, $y_{ij} = y_i - y_j$, dW_{ij}/dr is the derivative of the kernel (eq. (4)) with respect to r.

Equation (5) is adapted from [6] and takes advantage of the fact that the convective term of eq. (1) is null, matching the total derivative with the partial derivative. In eq. (5) the density and kinematic viscosity are considered constant for all fluid particles. Finally, the summation term of eq. (5) is on the neighboring particles of particle i.

The particles allocated do not move vertically in the Stokes-Couette flow. The determination of the neighborhood becomes simpler and does not need to be updated every time. Therefore, it is verified if the relative distance between particles is less than $2\,h$, that is: $r_{ij} \leq 2\,h$.

The fluid particles are created with a distance Δy between them and a distance of $\Delta y/2$ between the plates and the nearest fluid particle. Ghost particles (Fig. 1b) are created to complete the domain and avoid problems associated with the lack of neighboring particles at the end of the domain. The number of ghost particles created depends on the kernel and h. The velocities of the ghost particles created are called $v_{pi}(t)$, where $i = 1$ for ghost particles on the upper plate and $i = 2$ for the ghost particles near the lower plate. The velocities of ghost particle are calculated ensuring the boundary conditions, which are written in terms of the SPH method:

$$0 = \frac{\sum_j m_j\, v_{x,j}\, W(y_j, h)/\rho_j}{\sum_j m_j\, W(y_j, h)/\rho_j} \quad \text{and} \quad U\, \cos(\omega\, t) = \frac{\sum_j m_j\, v_{x,j}\, W(y_j, h)/\rho_j}{\sum_j m_j\, W(y_j, h)/\rho_j}, \tag{6}$$

whereas zero order correction is applied to the W kernel, or Shepard correction [7]. The sums in the boundary conditions expressed in eq. (6) refer to the set of particles close to the fixed and oscillating plates. Assuming that the velocities of all ghost particles near the oscillating plate are equal and imposing $y = H$ on eq. (6), $v_{p1}(t)$ is:

$$v_{p1}(t) = -\frac{v_{x,\,N}\, W(0,5\,\Delta y, h) + v_{x,\,N-1}\, W(1,5\,\Delta y, h)}{W(0,5\,\Delta y, h) + W(1,5\,\Delta y, h)}, \tag{7}$$

where $v_{x,\,N}$ and $v_{x,\,N-1}$ are the velocities of the real particle closest to the fixed plate. On the other hand, if $y = 0$, $v_{p2}(t)$ becomes:

$$v_{p2}(t) = 2\,U\, \cos(\omega\, t) - \frac{v_{x,\,1}\, W(0,5\,\Delta y, h) + v_{x,\,2}\, W(1,5\,\Delta y, h)}{W(0,5\,\Delta y, h) + W(1,5\,\Delta y, h)}, \tag{8}$$

where $v_{x,\,1}$ and $v_{x,\,2}$ are the velocities of the first and second real particles, respectively. Equation 7 shows that the velocity of ghost particles $v_p(t)$ needs to be updated at each time step Δt, even if $\omega = 0$. The time step chosen for the simulations is $\Delta t = 0.1\, h^2/\nu$.

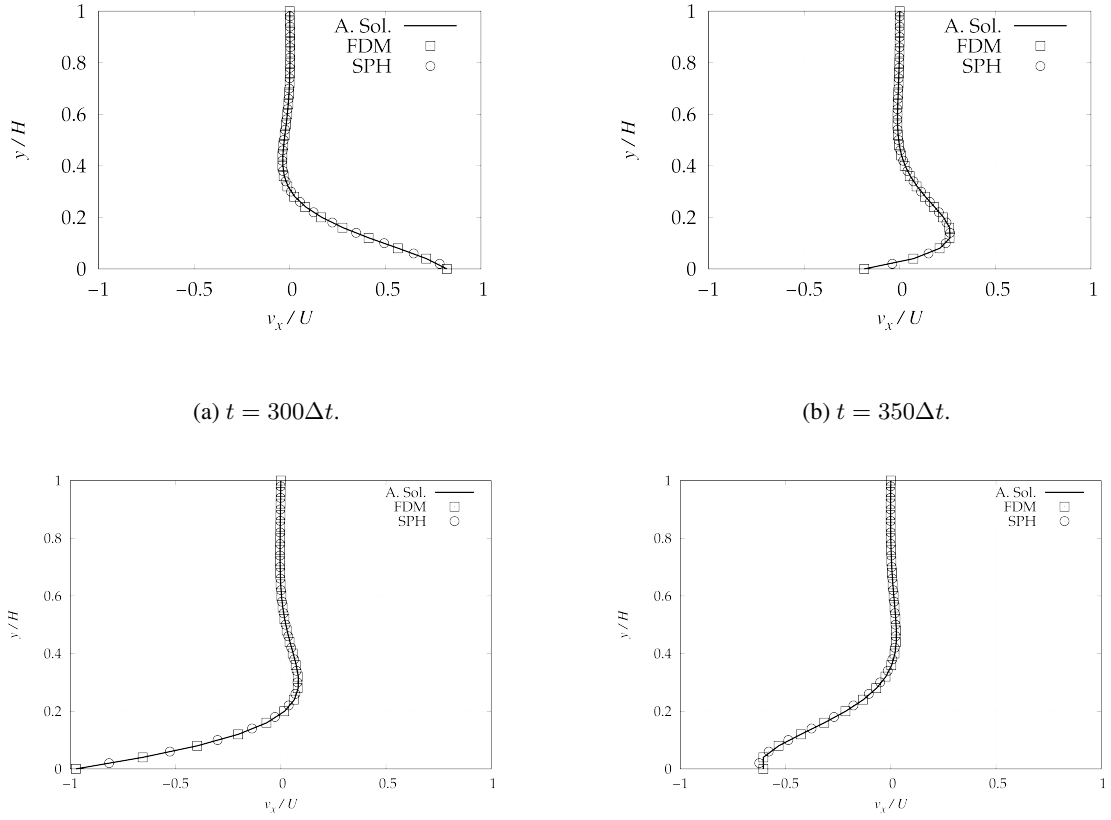
The 2nd order Runge-Kutta integration method is used to obtain the horizontal velocity of the particles. Finally, a simple explicit finite differences numerical method [1] is used to compare the velocity profiles, in addition to the eq. (2) and (3).

4 Results and discussion

The analysis of the Stokes-Couette flow starts from a discretization of the fluid domain of $N = 25$ particles distributed vertically, with a distance between the plates of $H = 0.1$ m, resulting in a space between particles of

$\Delta y = 0.004$ m. The smoothing length used (eq. (4)) is $h = 1.2\,\Delta y$ and the maximum search length is $2\,h$, which leads to the creation of four additional ghost particles, two external to each plate (Fig. 1b). The maximum speed is $U = 0.1$ m/s and the oscillation frequency of the oscillating plate is $\omega = 100$ rad/s. The Reynolds number of the chosen flow is $Rey = 1$, resulting in a fluid kinematic viscosity of $\nu = 0.01$ m^2/s, constant for all particles. The mass and density values of the particles result in $m/\rho = \Delta y = 0.004$ m. The time step is $\Delta t = 2.3 \cdot 10^{-4}$.

By starting from rest, the numerical solutions are compared with the analytical solution (eq. (2)) after completing an oscillation period. The analyzed times are then greater than $273\,\Delta t$. The Fig. 2 illustrate the speed profile obtained for four different times: $300\,\Delta t$, $350\,\Delta t$, $400\,\Delta t$, and $450\,\Delta t$.



(a) $t = 300\Delta t$. (b) $t = 350\Delta t$.

(c) $t = 400\Delta t$. (d) $t = 450\Delta t$.

Figure 2. Comparison between dimensionless velocity profiles for Stokes-Couette flow with $\omega = 100$ rad/s

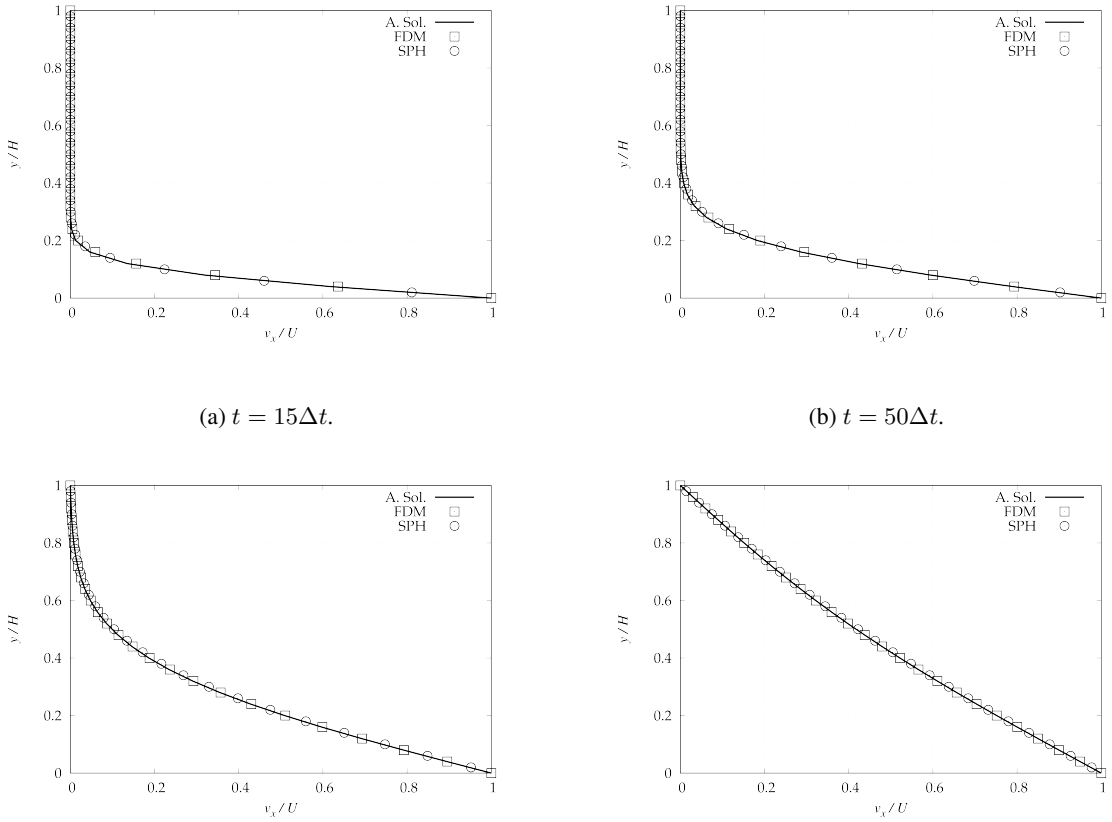
It can be seen in Fig. 2 that the boundary conditions for the velocity of the ghost particles (eq. (7) and (8)) correctly reproduce the expected values when compared with the analytical and FDM solutions.

Changing the oscillation frequency to $\omega = 0$ and considering 20 terms of the analytical solution, four different simulation times are analyzed: $15\,\Delta t$, $50\,\Delta t$, $200\,\Delta t$, and $900\,\Delta t$. The results can be seen at Fig. 3.

The velocity profile given by the SPH method reproduces the analytical one at all times analyzed and tends towards the permanent solution, as expected.

5 Conclusion

The Stokes-Couette flow solution using the SPH is detailed in this article. The SPH numerical method starts directly from the discretization of the heat equation (eq. (1)), using an approximation of the second derivative in SPH (adapted from [6]). The boundary conditions for the horizontal velocity, which follow the no-slip condition, must update the velocities of the ghost particles in each time step, even when $\omega = 0$. However, the implementation of the SPH method is simple and can be used in other 1D problems.



(a) $t = 15\Delta t$.

(b) $t = 50\Delta t$.

(c) $t = 200\Delta t$.

(d) $t = 900\Delta t$.

Figure 3. Comparison between dimensionless velocity profiles for Stokes-Couette flow with $\omega = 0$

References

[1] Fortuna, A. O., 2012. *Técnicas Computacionais para Dinâmica dos Fluidos: Conceitos Básicos e Aplicações*. EDUSP, São Paulo, 2nd edition.

[2] Liu, G. R. & Liu, M. B., 2003. *Smoothed Particle Hydrodynamics: a meshfree particle method*. World Scientific, Singapura.

[3] Gan, B. S. & Kawada, N., 2012. Fast calculation for particle interactions in SPH simulations: Outlined subdomain technique. *International Journal of Civil and Environmental Engineering*, vol. 6, pp. 28–34.

[4] Monaghan, J. J., 1982. Why particle methods work. *SIAM Journal on Scientific Computing*, vol. 3, n. 4, pp. 422–433.

[5] Fulk, D. A. & Quinn, D. W., 1995. An analysis of 1-D smoothed particle hydrodynamics kernels. *Journal of Computational Physics*, vol. 126, pp. 165–180.

[6] Cleary, P. W., 1998. Modelling confined multi-material heat and mass flows using SPH. *Applied Mathematical Modelling*, vol. 22, pp. 981–993.

[7] Bonet, J. & Lok, T. S. L., 1999. Variational and momentum preservation aspects of smooth particle hydrodynamic formulations. *Computer Methods in Applied Mechanics and Engineering*, vol. 180, pp. 97–115.