International Journal of Scientific Research & Engineering Trends
Volume 6, Issue 3, May-June-2020, ISSN (Online): 2395-566X

Confined Flow Study using Smoothed Particle Hydrodynamics

José Gustavo Coelho
Institute of Exact and Technological Sciences: Dept. of Mechanical Engineering
Federal University of Triângulo Mineiro: Uberaba, Brazil
jose.gustavo@uftm.edu.br

Artur Elias de Morais Bertoldi
Faculty of Gama; Aerospace Engineering
University of Brasília
Brasília, Brazil
artur.bertoldi@aerospace.unb.br

Abstract **– Smoothed Particle Hydrodynamics (SPH) is a Lagrangian meshless method that simulates flow as a set of fluid separation in motion. The forces acting on each particle are available using a spatial filter process. This filter plays an important role in the method. In this study, an analysis of this filter is made in a set confined to the low Reynolds number. The analyzed cases are an analytical equation and situations with Re = 1 and Re = 100 in a cavity. The results found numerically were satisfactory.**

Keywords **– SPH, Low Reynolds, Meshless, Numerical Simulation.**

I. INTRODUCTION

The SPH was created by Monagham [1] to simulate the dynamics of galaxies. Currently, it is used to simulate different physical cases such as fluid dynamics [2], [3], non-Newtonian fluid flows [4], multiphase flows [5], large deformation [6], granular material [7], explosions [8], and others.

The fluid dynamics equations are modelled by the SPH as discrete particles. These particles interact with their surroundings ones through hydrodynamic forces. These forces are analysed using a spatial average, which is the core of the model. The SPH is a model without mesh and therefore has an adaptive feature. Knowing the properties of the neighboring particle, one finds the state of the particle in question, such as velocity, pressure, density, etc.

With these characteristics, the model is appropriate to analyse flows with moving borders and free surface [8]. One of the difficulties that the method has is the determination of neighboring particles, since they move at each step of time, making it necessary to search for the next particle at each iteration. When working with incompressible flows, an additional problem is the determination of the pressure field, since the model was designed for an explicit approach, so for the calculation of pressure, the use of a state equation is necessary.

The model uses several parameters that are studied by the academic community, such as formulations for the equations of state [9], [2] and [10]. The kernel variation is done by [11].

The focus of this study is to use SPH for confined flows with low Reynolds number. Computer code was written in C++ and Stokes flow for Re = 1 is initially analysed and also the case of the cavity with Re = 1 and Re = 100. In both situations the results agree with the parameters used for validation.

II. EQUATIONS

In the Lagrangian form, the principle of conservation of mass and moment can be expressed by eq. 1 and eq. 2

$$\frac{D\rho}{Dt} = -\rho\nabla\cdot\boldsymbol{u} \qquad (1)$$

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{1}{\rho}\nabla P + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} + \boldsymbol{F} \qquad (2)$$

where ρ is the density of the fluid, u the velocity field (which is the velocity of the material particle), P the pressure field, F an external force by unity of mass and is the deviatoric part of the stress tensor of the fluid. The nabla operator, ∇, have the conventional meaning and the material derivative is defined by $D/Dt = \partial/\partial t + u\cdot\nabla$.

To determine the pressure field, Morris modeling is used, according to eq. 3

$$P = c^2\rho \qquad (3)$$

where c is the sound velocity throughout the medium. For the determination of c, we use the formulation of Liu [8], given in eq. 4.

$$c^2 \cong max\left(\frac{U_0}{\delta_*}, \frac{vU_0}{L\delta_*}, \frac{FL}{\delta_*}\right) \qquad (4)$$

where δ* = Δρ/ρ0, ρ0 is a reference density, U0 a characteristic velocity, L a characteristic length and ν the viscosity. A condition that needs to be satisfied to ensure incompressibility is that Δρ/ρ0 < 0.03.
In the SPH the fluid is represented by a set of discrete particles, where each particle has physical properties such as pressure, mass, volume, density and speed. The only parameter that does not vary is the mass of the particle.

All others change according to the time evolution and these properties represent a spatial average about a certain part of the domain. Considering a generic property α (which can be scalar, vector or tensorial) the value of a particle at a given position r is given by

$$\alpha(\boldsymbol{r}) = \int_{\Omega} \alpha(\boldsymbol{r}')W(\boldsymbol{r}-\boldsymbol{r}')d\boldsymbol{r}' \quad (5)$$

where W is an interpolation function, or kernel, that obeys all the properties of a typical probability density function, as explained in [12] and Ω is the domain. This integration is done only in a part of the domain, which defines the neighborhood of the particle. In this paper, we use the cubic spline kernel, defined by eq. 6,

$$W(s) = \begin{cases} 1 - \frac{3s^2}{2} + \frac{3s^3}{4} & if\ 0 \le s < 1 \\ \frac{2-s^3}{4} & if\ 1 \le s < 2 \\ 0 & if\ \ge 2 \end{cases} \quad (6)$$

where s = r/ h, h is the smoothed length and . The integration in the equation (6) actually is approximated by a summation over the discrete particles in the neighbourhood of the particle. So, the eq. 5 reduces toe eq. 7

$$\alpha(\boldsymbol{r}_a) \approx \sum_b \frac{m_b}{\rho_b}\alpha_b W_{(r_{ab})} \quad (7)$$

where b indicates that the summation is taken over the neighbourhood of the particle, m is the mass and rab is the distance between the particle a and the particle b.
The momentum equation in SPH method can be white as eq. 8

$$\frac{D\boldsymbol{u}_a}{Dt} = -\sum_b m_b\left(\frac{P_a}{\rho_a^2}+\frac{P_b}{\rho_b^2}\right)\nabla_a W_{(r_{ab})} + \sum_b \frac{m_b(\mu_a+\mu_b)\boldsymbol{u}_{ab}}{\rho_a\rho_b}\left(\frac{1}{r_{ab}}\frac{\partial W_{ab}}{\partial r_a}\right) + \boldsymbol{F}_a \quad (8)$$

where μ is the viscosity. Fa is the body force evaluated at the particle a and second instalment of the equation the is the viscous contribution. More information about this formulation is found in [9].
The continuity equation is calculated as show in eq. 9

$$\frac{D\rho_a}{Dt} = \sum_b m_b \boldsymbol{u}_{ab}\cdot\nabla_a W_{ab} \quad (9)$$

The time step is calculated by eq. 10

$$\Delta t = min\left\{\begin{matrix} 0.25\frac{h}{c}, \\ 0.25\min_a\sqrt{\frac{h}{f_a}}, 0.125\min_a\frac{h^2}{\nu} \end{matrix}\right\} \quad (10)$$

The boundary conditions used is based in the Leonard-Jones models, conform eq. 11

$$FP_{ab} = \begin{cases} D\left[\left(\frac{r_0}{r_{ab}}\right)^1 2 - \left(\frac{r_0}{r_{ab}}\right)^4\right]\frac{x_{ab}}{r_{ab}^2} & if\ \frac{r_0}{r_{ab}} \le 1 \\ 0 & if\ \frac{r_0}{r_{ab}} > 1 \end{cases} \quad (11)$$

This method uses the idea of repulsive force in the domain wall particles. In the eq. 11, D is the magnitude of the square of the maximum velocity in the domain and the distance r0 is approximately equal to the initial separation between the particles. More information about the boundary conditions can be found in [8] and Monaghan [10].

III. RESULTS

Initially, we started with an analytically constructed solution using the Stokes equation. The Reynolds number is defined and the velocity equation is obtained. From that velocity one arrives at the field force. The domain is a 2D square, with unit length. The calculated field force will be the boundary condition inserted in the numerical simulation together with the continuity equation. Thus, the result will be a velocity profile that will be compared with the analytically constructed profile. The field strength is shown in eq. 12 and the continuity equation is shown in eq. 13.

$$\boldsymbol{F} = \frac{1}{\rho}\nabla P - \nu\nabla^2\boldsymbol{u} \quad (12)$$

$$\nabla\cdot\boldsymbol{u} = 0 \quad (13)$$

where An analytical solution of eq. (12) and (13) is found by choosing a velocity field u = (u1,u2) that respects the continuity equation and the wall conditions is non-slip. Thus, we arrive at the formulation for u, shown in eq. 14.

$$u_1 = x_1^2(1-x_1)^2(2x_2 - 6x_2^2 + 4x_2^3) \quad (14)$$
$$u_2 = -x_2^2(1-x_1)^2(2x_1 - 6x_1^2 + 4x_1^3)$$

The pressure is constant throughout the control volume. Thus, you arrive at the field force, F, shown by eq. 15.

$$
\begin{aligned}
F_1 = {} & (-24x_2 + 12)x_1^4 + (48x_2 - 24)x_1^3 + \\
& (-48x_2^3 + 72x_2^2 - 48x_2 + 24)x_1^2 + \\
& (48x_2^3 - 72x_2^2 + 24x_2 - 2)x_1 + \\
& (-8x_2^3 + 12x_2^2 - 4x_2 + 1)
\end{aligned}
\tag{15}
$$

$$
\begin{aligned}
F_2 = {} & (48x_2^2 - 48x_2 + 8)x_1^3 + (-72x_2^2 + \\
& 72x_2 - 12)x_1^2 + (24x_2^4 - 48x_2^3 + \\
& 48x_2^2 - 24x_2 + 4)x_1 + (48x_2^3 - \\
& 72x_2^2 + 24x_2 - 2)x_1 + (-12x_2^4 + \\
& 12x_2^3 - 12x_2^2)
\end{aligned}
$$

With the boundary conditions defined, it is possible to engender the numerical code. A first simulation was performed with Re = 10−6. This number is defined from the maximum velocity across the domain. We used 1600 particles of internal fluid equally spaced at the beginning of the simulation. For the wall conditions it is used 320 fixed particles. The smoothing length, h, used was h=1.3Δx0. The result of the velocity profile is showed in Fig. 1, where the horizontal central line is compared with analytical solution. Fig. 2 shows the velocity vectors in all domain.

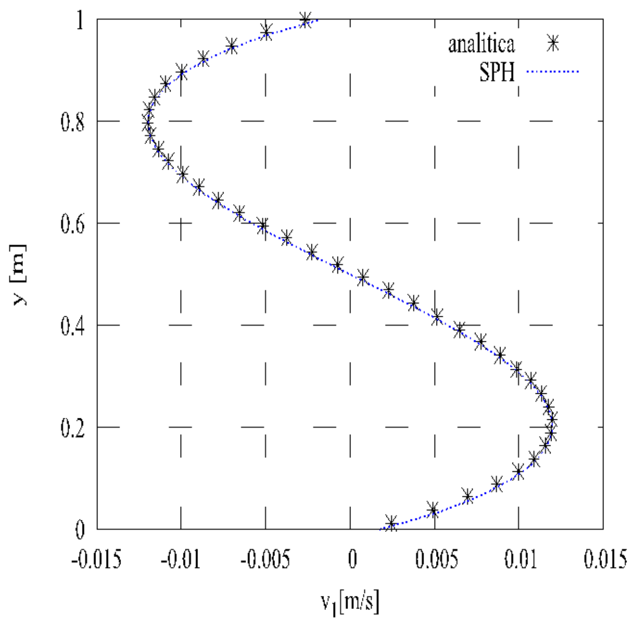

Fig.1. Velocity profile at the horizontal central line.

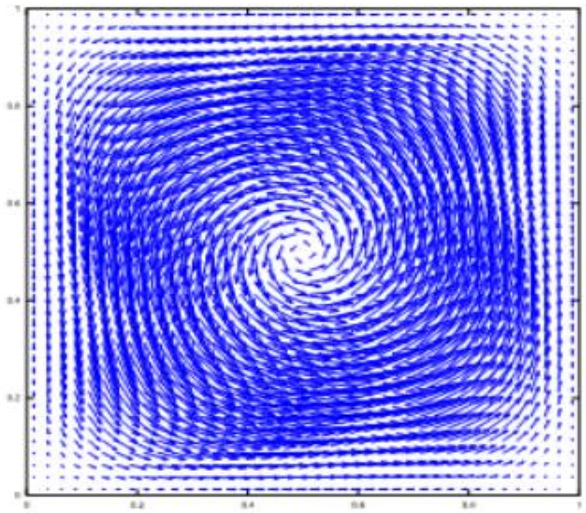

Fig.2. Velocity vectors in all domain.

Once that the method is built and validated, the study focuses on the analysis of movable wall. In this way, a square geometry cavity is chosen in which one of the walls moves in order to generate flow in the fluid. This analysis are compared with experimental data, found in Liu (2003). The Reynolds number used was equal to 1 and the side of the cavity was L = 10 -3 m long. To secure Re = 1, the velocity of the upper wall U = 10-2 m has to be reached. In order to impose this boundary condition, a line of fixed particles is created and this velocity is imposed on them. We use 1600 fluid particles and 320 wall particles, and the smoothing length, h, is equal to initial separation between the particles.

Figure 3 shows the velocity field found after the convergence. Figure 4 and Fig. 5 show comparison between experimental and numerical data. It is noticed that both results agree. The horizontal velocity component was analyzed on a central line of the domain.

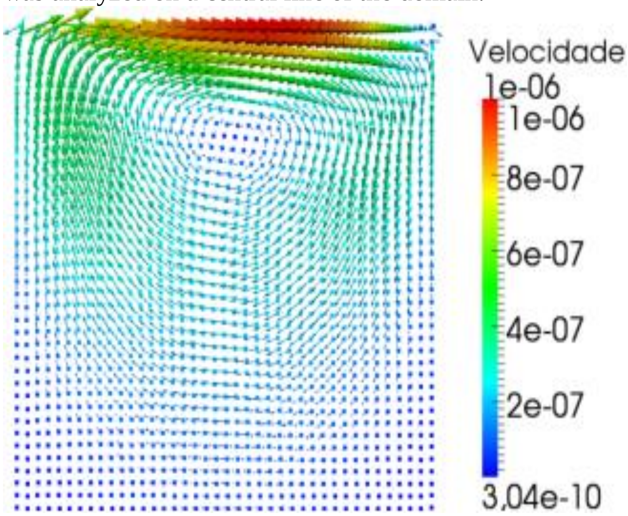

Fig.3. Shear-driven cavity flow after the convergence.

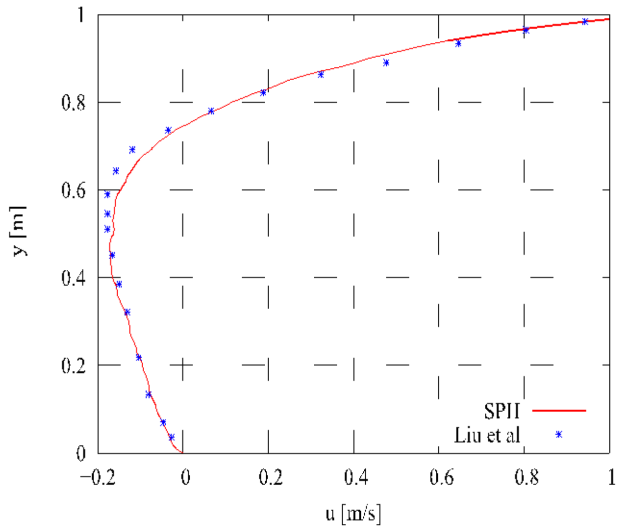

Fig.4. Vertical component of the velocity at the central line in the cavity for Re = 1. The experimental data is found in [8].

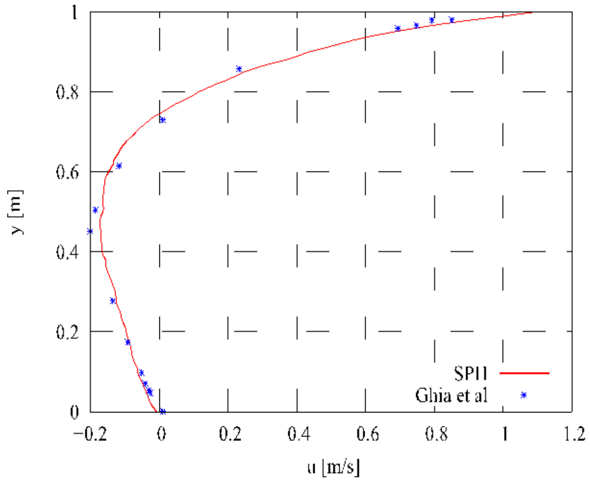

Fig. 6. Vertical component of the velocity at the central line in the cavity for Re = 100. Red thick line is SPH results and black thin line is the experimental data from Ghia (1982).

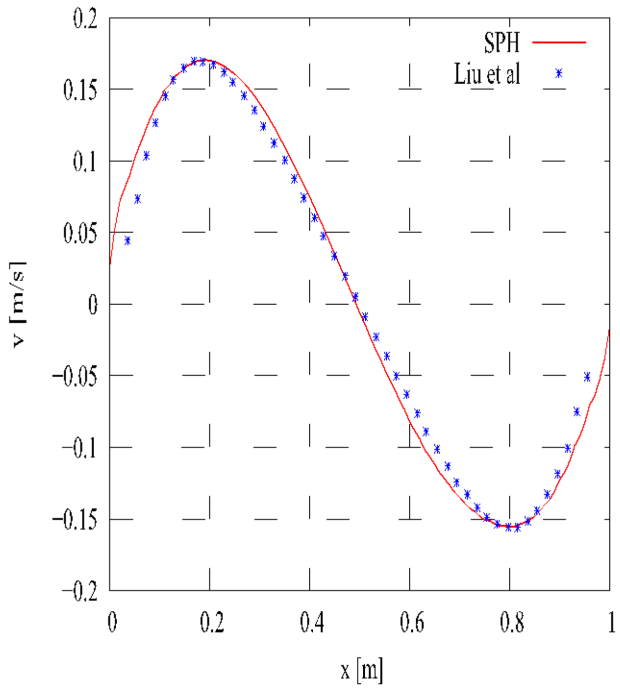

Fig.5. Horizontal component of the velocity at the central line in the cavity for Re = 1. The experimental data is found in [8].

Another simulation carried out was increasing the number of Reynolds. It is done with Re = 100. To do it, the speed of the wall is also increased. Thus, it was necessary to increase the particles, going to 64000 particles of fluid and 320 wall particles. It is used as length for h, 1.3 Δxo. The numerical results found were compared with Ghia [8]. Figure 6 and Fig. 7 present the comparison between the results. As expected, the results were satisfactory.

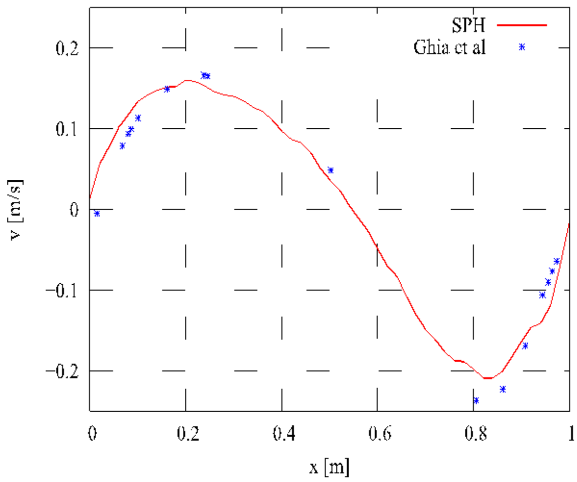

Fig. 7. Horizontal component of the velocity at the central line in the cavity for Re = 100. The experimental data is found in [8].

IV. CONCLUSION

An SPH numerical code was implemented to perform simulations in confined flow for low Reynolds numbers. A solution was created based on Stokes' equations. The code performance in flows confined with one of the moving walls was analysed. Simulations with Re = 1 and Re = 100 were studied. The results were satisfactory and with that, we intend to move on a new phase of the study, which is to implement new boundary conditions in the code, and also to analysing situations with free surface.

V. ACKNOWLEDGMENT

This work is supported by FAPEMIG project APQ-00620-14.

REFERENCES

[1]. Monaghan J. J. "Smoothed Particle Hydrodynamics", Annual Review of Astronomy and Astrophysics 30, 543 – 574, 1992.

[2]. Monaghan J. J. "SPH compressible turbulence", Royal Astronomical Society, 335, 843-852, 2002.

[3]. Hu X. Y., Adams N. A. "An incoompressible multi-phase SPH method", Journal of Computational Physics 227, 264-278, 2007.

[4]. Lee E. S., Moulinec C., Xu R., Dioveau D. Laurence D. Stansby P."Comparisons of weakly compressible and trully incompressible algorithms for the SPH mesh free particle method", Journal of Computational Physic 227, 8417 – 8436, 2008.

[5]. Jian C., Ognjen O., Kenneth W., Jingijing M., Chegzhi L. “A coupled DEM-SPH model for moisture migration in unsaturated granular material under oscillation”, International Journal of Mechanical Sciences, 169, 105313, 2020.

[6]. Serroukh H. K., Mabssout M., Herreros M. I. “Updated Lagrangian Taylor-SPH method for large deformation in dynamic problems”, Applied Mathematical Modelling, 80, 238-256, 2020.

[7]. Meng Z., Wang P., Zhang A, Ming F., Sun P., “A multiphase SPH model based on Roe’s approximate Riemann solver for hydraulic flows with complex interface”, Computer Methods in Applied Mechanics and Engineering, 365, 112999, 2020.

[8]. Liu G. R., Liu M. B. "Smoothed Particle Hydrodynamics", Ed. World Scientif Publishing, Singapore, China, 473 p, 2003.

[9]. Morris J. P., Fox P.J., Zhu Y. "Modelling Low Reynolds Number Incompressible Flows Using SPH", Journal of Computational Physics 136, 214-226, 1997.

[10]. Monaghan J. J. "Smoothed Particle Hydrodynamics", Institute of Physics Publishing, Reports on Progress in Physics 68, 1703 – 1759, 2005.

[11]. Hongbin J., Xin D. "On criterios for smoothed particle hydrodynamics kernels in stable field", Journal of Computational Physiscs 202, 699-709, 2005.

[12]. Libersky L. D., Petschek A. G., Carney T. C. Hipp J. R., Allahdadi F. A., 1993, "High strain Lagrangian Hydrodynamics", Journal Computational Physics 109, 67, 1993.

AUTHOR’S PROFILE

José Gustavo Coelho
Graduated in Mathematics, Master in Mechanical Engineering and Doctorate (sandwich) in Mechanical Engineering at the University of Brasília (UnB) and Ecole Polytechnique Fédérale de Lausanne (EPFL / Switzerland), at the Laboratory for Hydraulic Machines. He has experience in the area of numerical flow simulation, using commercial software and also using his own program, called XSPH-UnB. Currently has work in progress in the areas of fluid mechanics, flow machines and aerospace simulation, using Eulerian (Finite Volumes and Finite Elements) and Lagrangian models, in particular the Smoothed Particle Hydrodynamics (SPH) and Discrete Element Method (DEM).

Artur Elias de Morais Bertoldi
Professor Artur E. M. Bertoldi, graduated in Physics at Federal University of São Carlos – UFSCar (São Paulo, Brazil). He has Master and PhD degrees in Mechanical Sciences at University of Brasília - UnB (Brazil) with International Certificate in doctoral research training (CIRD) from the Aero-Thermo-Mechanics Department at Université Libre de Bruxelles (Belgium). He has professional experience in public and private institutions and is currently professor for the Aerospace Engineering undergraduate course at University of Brasília. His work focuses specifically on the following themes: hybrid rockets design, regression rate studies, liquid injection system design and testing, combustion instabilities and experimental techniques for rocket propulsion.