Proceeding Series of the Brazilian Society of Applied and Computational Mathematics, v. 6, n. 1, 2018.

Trabalho apresentado no XXXVII CNMAC, S.J. dos Campos - SP, 2017.

Proceeding Series of the Brazilian Society of Computational and Applied Mathematics

---

A General Approach for the 2D Lid-Driven Cavity Flow at Low Reynolds Number Using SPH

Thiago Vasconcellos Birro[1]
Technological Center, Department of Mechanical Engineering, UFES, Vitória, ES
Julio Tomás Aquije Chacaltana[2]
Technological Center, Environmental Engineering Post-Graduate Program, UFES, Vitória, ES
Carlos Friedrich Loeffler Neto[3]
Technological Center, Mechanical Engineering Post-Graduate Program, UFES, Vitória, ES
Antonio Manoel Ferreira Frasson[4]
Technological Center, Department of Electrical Engineering, UFES, Vitória, ES

1 Introduction

This work explores the advantages of one of the most known meshless methods, called Smoothed Particle Hydrodynamics (SPH), modeling the 2D lid-driven cavity flow at low Reynolds numbers. The SPH is a Lagrangian scheme and solves the continuity, momentum and energy equations by discretization of the domain of study in material points, in which no connectivity is required among them. Each particle has an associated mass m and field variables, being the latter terms calculated by a particle approximation defined in a compact support domain, as shown in equation (1). In addition, each particle has its own support domain, defined by a smooth length h. In a general way, the particle approximation of a function f is given as:

$$\langle f(x_i) \rangle = \sum_{j=1,}^{N} \frac{m_j}{\rho_j} f(x_j) W(x_i - x_j, h) \tag{1}$$

Where ρ is the density, W the Smoothing function or Kernel function and N the number of particles present in the compact domain. In addition, the index i means the actual particle and j the surrounding particles of i.

With the background developed by different studies, the objective of this production is to solve the 2D lid-driven cavity flow at low Reynolds with a code developed by Liu and his co-workers [2]. Two different boundary conditions are tested to model the presence of

---

[1] thiago.birro@hotmail.com

[2] julio.chacaltana@ufes.br

[3] carlosloeffler@bol.com.br

[4] antonio.frasson@ufes.br

solid walls: repulsive boundary force and new implementation using reflective boundary condition.

2 Cases of Study

A cavity of 1 m x 1 m, using water as fluid and Re from 1 to 100, is carried out with 1600 real particles in the domain and 320 virtual particles along the boundary. All results are compared in order to validate them with Ghia [1] and Pinto [3]. In this submission, only the results using Re=1 in figure 1 is shown.

3 Results

The first results for the both boundary conditions are in good agreement with the reference values found [1–3]. In addition, some improvements are being developed to achieve a higher order of accuracy, based on Kernel correction schemes. These improvements aim to correct the maximal deviation of Vx profile at $y=$ 0.5 m. However, even using a simpler analysis, the SPH shows to be a powerful numerical method to solve fluid dynamics problems.

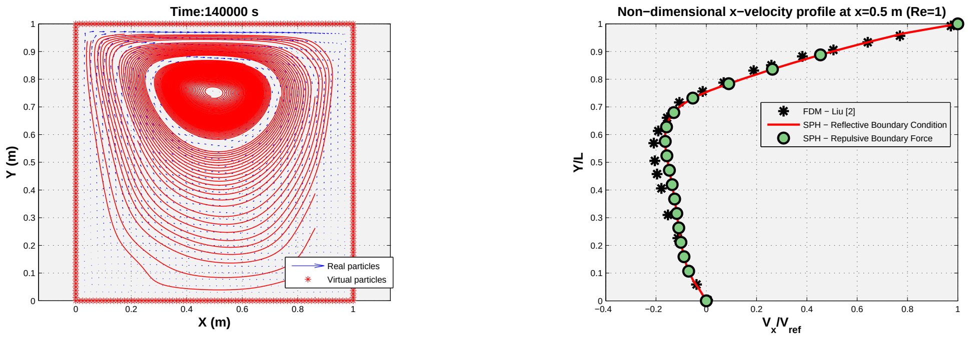

Figure 1: Results of the simulation using Re=1, water as fluid and repulsive boundary force.

References

[1] U. Ghia, K. N. Ghia, C. T. Shin. High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. *Journal of Computational Physics*, 48:387-411, 1982. DOI: 10.1016/0021-9991(82)90058-4.

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

[3] W. J. N. Pinto, Aplicação do método lagrangiano SPH para a solução do problema das cavidades, Master's Thesis, Federal University of Espírito Santo, 2013.