

## Mathematical and Numerical Modelling of Gas-Solid Turbulent flows in complex geometries

Stella Rodrigues Ferreira, Laboratório de Mecânica dos Fluidos, stellarodrigues7@gmail.com  
João Marcelo Vedovoto, Laboratório de Mecânica dos Fluidos, jmvedovoto@mecanica.ufu.br  
Aristeu da Silveira Neto, Laboratório de Mecânica dos Fluidos, aristeus@mecanica.ufu.br

**Abstract:** *The presence of solid particles in industrial turbulent flows in complex geometries is widely evidenced in engineering. However the correct prediction of flow patterns, particles trajectories and distributions is a highly complex task. The analysis of the particle velocity as well as their trajectories in the gas-solid flow is of fundamental importance. We used an Euler-Lagrangian approach where the dispersed phase is treated in the Lagrangean referential and the governing equations for the continuous phase are solved in the Eulerian referential. An advantage of the application of the Immersed Boundary Method is that the immersed geometry is also represented in the Lagrangean referential. It is worthy to recall that one of the advantages of the Immersed boundary methodology is the capability of solving flows in the presence of complex geometries in Cartesian grids, which facilitates the particle tracking of solid particles in LES. By performing Large Eddy Simulations we demonstrate, in the present work, that it is possible to predict complex multiphase flows by combining promising methodologies like LES and Immersed Boundary Methodology. The fully parallel in-house code developed was validated face to experimental results of Sommerfeld and S.Lain (2001) with very good accuracy.*

**Palavras chave:** CFD, Complex geometries, Immersed Boundary Method, solid particles

### 1. INTRODUCTION

In many engineering applications multiphase flows are encountered. The computation of the continuous phase is often done using the Eulerian approach, in which partial differential equation for momentum, mass, and energy per unit volume are solved. In the simulation of the disperse phase, the Lagrangian approach was used. The equation of motion of a particle are solved by the ordinary differential equations for the velocity, position, mass, momentum. The solutions of these equations requires the evaluation of fluid properties at the particles locations.

The calculation of the positions and velocities of the particles present in fluid flow is of a considerable importance in studying the solid particles in turbulent channel flows, where the collisions of the solid particles with the walls play an essential role in the particle transport process.

Since the early 1990s, computation fluid dynamics (CFD) has been widely used for studying the physical effect of solid particles in curved piped or duct and in turbulent channel flows, with various numerical models for particle position and the cell contains a nearby position. In the present work, a numerical models involving the study of solid particles interaction with complex geometries was developed. In this context, the presence of turbulent flows in complex geometries is a rule, as in many engineering applications multiphase flows. The immerse boudary method allows the use of a cartesian mesh for modeling the flow over complex structures using lagrangian mesh.

In the present work was used the turbulent channel flow for validation of the one-way coupling according to experimental results by Sommerfeld and K.kussin (2001). For verification the algorithm implemented for particle-wall collision was formulation the shock between a particle and a inclined plane. They are purely geometrical exercises of intersection between Lagrangian particles and planes.

### 2. METHODOLOGY

In this section is presented the mathematical model used in the numerical model for the simulation particles in turbulent flows. Therefore, in this model, it is used the Euler-Lagrangian referential, which can be approached through the punctual force or resolved surface. Regardless of the approach, each computational particle is tracked in the computational domain.

## 2.1 Filtered transport equations for large eddy simulations

One of the most widely used methods is the large eddy simulation (LES), which filters the Navier-Stokes equations to segregate the different scales of the flow, in order to calculate the highest flow structures turbulent and model the minors.

As in the filtered Navier-Stokes equations, the turbulence scales are separated according to the group of large and small scales named sub-grid scales. The functions can be decomposed into a floating part in the form  $f = \bar{f} + f'$ , where  $f$  is a any function given by:

$$\bar{f}(\mathbf{x}) = \int f(\mathbf{x}')F(\mathbf{x} - \mathbf{x}')d\mathbf{x}', \quad (1)$$

wherein  $F$  is the filter, defined by:

$$F(\mathbf{x}) = F(x_1, x_2, x_3) = \begin{cases} 1/\Delta^3 & \text{se } |x_i| \leq \Delta, i = 1, 2, 3 \\ 0 & \end{cases}, \quad (2)$$

where  $(x_1, x_2, x_3)$  are the spatial coordinates of the vector  $\mathbf{x}$ ,  $\Delta = \sqrt[3]{V_C}$  is a sub domain in a finite volume of a mesh to discretize the transport equations in the Eulerian domain, and  $V_C$  is a volume of mesh element.

The filtered quantities  $\bar{f}$  and  $\tilde{f}$  were claculated by numerical simulation. The fluctuation  $f = \bar{f} + f'$  are the tensor of subgrid scales not resolved quantity  $f$ .

$$\frac{\partial \bar{p}}{\partial t} + \frac{\partial \bar{p}\tilde{u}_i}{\partial x_i} = 0, \quad (3)$$

$$\frac{\partial \bar{p}\tilde{u}_i}{\partial t} + \frac{\partial \bar{p}\tilde{u}_i\tilde{u}_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j} - \frac{\partial \tau_{ij}^{SGS}}{\partial x_j}. \quad (4)$$

where,  $\tau_{ij}^{SGS} = (\bar{\rho}\tilde{u}_i\tilde{u}_j - \bar{\rho}\tilde{u}_i\tilde{u}_j)$  is the tensor of subgrid scale (SGS), which is the immediate consequence of the scales of decomposition process.

## 2.2 Subgrid dynamic modeling

The dynamic model is based also on the Boussinesq hypothesis Boussinesq (1877), where the dynamic procedure model coefficients are automatically computed using information contained in the resolved turbulent scales, thereby eliminating possible uncertainties associated with tunable parameters.

Similar to the Smagorinsky model Smagorinsky (1963) the subgrid viscosity, nevertheless in the dynamic model the  $C_s$  is evaluated using the dynamical procedure for turbulent tensor, as follows:

$$C_s = \frac{\mathcal{L}_{ij}\mathcal{M}_{ij}}{2\mathcal{M}_{ij}\mathcal{M}_{ij}}, \quad (5)$$

Where,

$$\mathcal{L}_{ij} = -\widehat{\bar{\rho}\tilde{u}_i\tilde{u}_j} + \bar{\rho}\tilde{u}_i\tilde{u}_j, \quad (6)$$

and

$$\mathcal{M}_{ij} = \widehat{\bar{\rho}\Delta^2\tilde{S}_{ij}|\tilde{S}|} - \bar{\rho}\Delta^2\tilde{S}_{ij}|\tilde{S}|. \quad (7)$$

where  $\mathcal{L}_{ij}$  is the Leonard tensor.

## 2.3 Equations of motion for the lagrangian approach

Using the Lagrangian approach to the particulate phase requires the solution of the equation of motion for each computational particle. This motion equation includes the particle's inertia, drag, gravity, slip-shear lift force and slip-rotational lift force. Besides, the change of angular velocity along particle trajectory results from the torque equation and wall collisions.

The equations of motion for the particles are given by :

$$\frac{dx_{pi}}{dt} = u_{pi} \quad (8)$$

$$m_p \frac{du_{pi}}{dt} = \frac{3}{4} \frac{\rho}{\rho_p D_p} m_p C_D (u_i - u_{pi}) |\vec{u} - \vec{u}_{pi}| + m_p + g_i \left(1 - \frac{\rho}{\rho_p}\right) + F_{lsi} + F_{lri} \quad (9)$$

$$I_p = \frac{dw_{pi}}{dt} = T_i \quad (10)$$

where,  $x_{pi}$  is the actual particle position,  $u_{pi}$  are the velocity components of instantaneous velocity of the fluid,  $m_p$  is the particle mass,  $\rho_p$  is the density of particle,  $D_p$  is the particle diameter, and  $I_p$  is the moment of inertia for a sphere.

The Reynolds number for the particle ( $Re_p$ ) is define as :

$$Re_p = \frac{\rho_p d_p (u - u_p)}{\mu}, \quad (11)$$

where the  $F_D$  is the drag force, define as :

$$F_D = \frac{18\mu}{\rho_p (d_p)^2} + \frac{C_D Re_p}{24}, \quad (12)$$

where  $C_D$  is the drag coefficient is obtained based on the analytical result of Schiller e Naumann (1935):

$$C_D = \begin{cases} \frac{24(1+0.15Re_p^{0.687})}{Re_p} & \text{for, } Re_p \leq 1000, \\ 0.44 & \text{for, } Re_p > 1000, \end{cases} \quad (13)$$

The Slip-shear force is defined as:

$$\vec{F}_{ls} = 1.615\mu D_p Re_s^{1/2} c_{ls} \frac{[\vec{u} - \vec{u}_p] \times \vec{u}_p}{|\vec{w}|}, \quad (14)$$

where  $\vec{w} = 0.5\Delta \times \vec{u}$  is the fluid rotation, and  $Re_s = \rho D_p^2 \frac{|\vec{w}|}{\mu}$  is the particle Reynolds number of the shear flow, and  $c_{ls}$  represents the ratio of the extended lift force to the Saffman force are given by:

$$c_{ls} = \begin{cases} (1 - 0.3314\beta^{0.5})e^{\frac{-Re_p}{10}} + 0.3314\beta^{0.5} & \text{for } Re_p \leq 40, \\ 0.0524(\beta Re_p)^{0.5} & \text{for } Re_p > 40, \end{cases} \quad (15)$$

The following form of the slip-rotation lift force has been used:

$$\vec{F}_{lr} = \frac{\pi}{8} D_p^3 \rho \frac{Re_p}{Re_r} c_{lr} [\vec{\omega} \times (\vec{u} - \vec{u}_p)]. \quad (16)$$

Here,  $\vec{\omega} = \vec{w} - \vec{w}_p$ , and the Reynolds number of particle rotation is given by  $Re_r = \rho D_p^2 |\vec{\omega}| / \mu$ . The lift coefficient to may be used for  $Re_p = 2000$  by:

$$c_{lr} = 0.45 + \left(\frac{Re_r}{Re_p - 0.45}\right) * e^{-0.0568 Re_r^{0.4} Re_p^{0.3}}. \quad (17)$$

The torque force acting on a rotating particle and de expression of Rubinow and Keller (1961) was extended to account for the relative motion between fluid and particle, as shown below:

$$\vec{T} = \frac{\rho}{2} \left(\frac{D_p}{2}\right)^5 C_R |\vec{\omega}| \vec{\omega}. \quad (18)$$

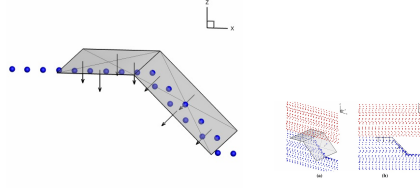


Figura 1. Inclined plane with particle reflection

## 2.4 Interpolation of the continuous phase into the particle course

In this study was used the hat function for the interpolation of eulerian velocity. This function ensure for eulerian velocity the second order of accuracy, as defined by the following:

$$D_h = \prod \frac{\varphi[(x_k - x_i)/\Delta x_i]}{\Delta x_i}, \quad (19)$$

$$\varphi = \begin{cases} |1 - r| & \text{se } r \leq 1, \\ 0 & \text{se } r > 1 \end{cases}. \quad (20)$$

Where  $x_k$  is the Lagrangian position,  $x_i$  is the fluid particle position, and  $\Delta x_i$  is the mesh dimension.

This interpolation method of one particle within a control volume is weighted by the inverse of the distance, squared between the particle and the nodes of the mesh.

## 2.5 Mathematical modeling of the immersed boundary method for particle wall collision model

The immersed boundary method makes use of two different grids: i) An Eulerian grid, where the balance equations are solved; ii) A Lagrangian representation of the immersed object, i.e., a set of Lagrangian points that discretize an object. There are different ways of creating such a set of points, first if the geometry is simple enough to be represented by an equation, like a cylinder for instance, and second when no other lagrangian entities, the presence of only the set lagrangian points that characterize the immersed object is deemed sufficient to the (multi) Direct Forcing method work.

### 2.5.1 Shock of a particle which a inclined plane

In the present work the particle reflection condition as well the respective impacted angle with the geometry wall. Such geometry is represented by the immersed boundary method, which is composed by triangles which normal are defined and pointing outwards, into the geometry.

In this first test, a particle is assign with a constant velocity  $U_p$  and its trajectory is set in such a manner that it will pass close to a plane normal to the trajectory, then being reflected by an inclined plane. The plane is composed by eight large triangles.

As for the calculation of the particle reflection was certificated if the product value between the particle trajectory and the triangle face normal which makes up the plane geometry, if the product is negative the particle can chock in the plane. But if the internal product was higher or equal then zero, won't possibility the particles won't intercept the plane.

The figure 1 shows the evaluation of the particle position in relation to the planes, the particles recognize the triangle collides which the plane and are reflected. Despite the evaluation of the particle position, the velocity in this case is not changing, and carrying a new reflection model.

For the second test, a particle started with a constant velocity  $U_p$  allowing to calculate what was the particle vector trajectory angle does and its trajectory is set, in such a manner that it will pass close to a plane normal to the trajectory, and then it must be reflected by an inclined plane, but they changing your impact angle.

A view of the plane XZ, to giving details. About the reflection of the particles that proceeds to cross the immersed object. It can be noted in Fig. (2) that, as expected, the particles that intersect the inclined plane are deviated until no obstacle is present.

## 3. Results

In this section is presented the one-way and two-way are simulations. For the verification of the couplings, it was considered a pipe with immersed boundary method, these geometries being examples of those commonly found in practical engineering applications.

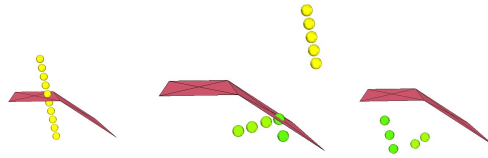


Figura 2. Inclined plane with change of impact angle

### 3.1 Confined flow in a duct

In order to investigate the suspension of dust particles in pipe flows is essential to understand the effect of turbulent transport of particles, as well as the Magnus effect induced by particle-wall interactions. For the analysis of the influence of particle motion in the continuous phase, in this work were considered the one-way and two-way coupling. In two-way coupling was used the transverse lift forces, due to particle rotation, and particle inertia.

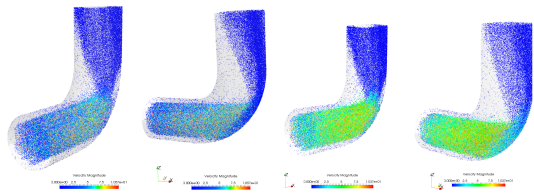


Figura 3. i) Particles in the pipe one-way coupling; ii) Particles in the pipe two-way coupling

The Figure 5 and 6 represents a the particle transport from a horizontal pipe through pipe bend into a connecting vertical pipe, the particle inertia result in an accumulation of particles at the outer wall of the bend, and this phenomenon is called to ‘dust rope’ . In the connecting vertical pipe, this rope will disintegrate due to the secondary flow induced by the bend and due to flow turbulence. An analysis of Figure 5 reveals that the one-way coupling in this case had lower speed then the two-way coupling, was expected, as in two-way coupling the disperse phase have influence in continuous phase.

## 4. CONCLUSIONS

The lagrangean approach for discrete phase allows one study the particle trajectory during the time, and the motion of particles is strongly governed by inertial effects In the pipe configuration pipe bend and horizontal pipe, the development of the particle concentration downstream of the bend is highly unsteady.

## 5. ACKNOWLEDGMENT

The authors would like to thank CAPES, PETROBRAS, CNPq, FAPEMIG and the School of Mechanical Engineering, Federal University of Uberlândia, for all the material and financial support granted.

## 6. REFERENCES

- Boussinesq, J., 1877. “Essai sur la Théorie des Eaux Courantes”. In *Mémoires Présentés par Divers Savants à l’Académie des Sciences*. Paris, France, pp. 1–660.
- Smagorinsky, J., 1963. “General circulation experiments with the primitive equations”. *Monthly Weather Review*, Vol. 91, No. 3, pp. 99–164.
- Sommerfeld, S. and K.kussin, 2001. “Analysis of transport effects of turbulent gas-particle flow in a horizontal channel”. *Proceedings 4th International Conference on Multiphase Flow*, Vol. 4, No. 10-11, pp. 1–12.

## 7. ACCOUNTABILITY OF INFORMATION

The authors are only responsible for the information included in this work.