SIMULATION OF FREE FALL PARTICLES USING FOURIER PSEUDO-SPECTRAL METHOD COUPLED WITH IMMERSED BOUNDARY METHOD

Felipe Pamplona Mariano, fpmariano@mecanica.ufu.br
Leonardo de Queiroz Moreira, lqmoreira@mecanica.ufu.br
Aristeu da Silveira Neto, aristeus@mecanica.ufu.br
Universidade Federal de Uberlândia - UFU
Faculdade de Engenharia Mecânica - FEMEC
Laboratório de Mecânica dos Fluidos - MFLab
Av: João Naves de Ávila, 2121
Campus Santa Mônica, Bl: 5P
CEP: 38400-902 Uberlândia-MG-Brasil
Phone: (34) 3239-4040 / Fax: (34) 3239-4042

Adailton Silva Borges, adailton@utfpr.edu.br
Universidade Tecnológica Federal do Paraná – UTFPR
Departamento de Engenharia Mecânica
Av. Alberto Carazzai, 1640
Campus Cornélio Procópio
CEP 86300-000 - Cornélio Procópio – PR
Phone: +55 (43) 3520-4000 / Fax: +55 (43) 3520-4010

Abstract. *The present paper shows simulations of free fall particle in a fluid. These problems, that involve solid-liquid two phase flows are studied for interest of chemical, pharmaceutical and food industries as well as geophysical environments. The fluid structure interaction (FSI) problems is a challenge for researchers of Computational Fluids Dynamics (CFD) and a method very used to solve it is the Immersed Boundary Method (IBM). IBM models the boundary conditions through a force field imposed at Navier-Stokes equations. Such a methodology does not requires any procedure of remeshing the computational grid, therefore, a FSI simulation can be carried out most efficiently. Nevertheless it presents, low accuracy and low convergence order. Aiming to solve this restriction, a new methodology is proposed in the present work, which uses the Pseudo-Spectral Fourier Method. This method provides an excellent numerical accuracy with the development of the Fast Fourier Transform algorithm (FFT). It shows lower computational cost than others high-order methods. Furthermore another important issue, is the projection method of the pressure term. Which in the Fourier space, does not require a Poisson solver, that is usually the most onerous part of the classical computational methodologies. For validate the new methodology, it was proposed simulate two problems. The first one is the flow around a circular cylinder, that is a classical problem of CFD and drag and lift coefficients are avalable. The second problem is free fall particle in a fluid , which were compared the results with different authors showing good agreement and the potenciality of new method.*

Keywords*: Computational Fluids Dynamic, Fluid Structure Interaction, Free Fall Particle, Fourier Pseudo-Spectral Method, Immersed Boundary Method.*

1. INTRODUCTION

The present paper shows the potential of the method called IMERSPEC proposed by Mariano (2007) and Mariano et al. (2010a). This method works with two methodologies, Immersed Boundary and Fourier Pseudo-Spectral, in order to obtain the best qualities of them.

There are two advantages of working with the Fourier pseudo-spectral methodology (Canuto *et al.*, 2006). One of them is the high order numerical accuracy obtained in the solution of partial differential equations. The other is when working specifically with incompressible Navier-Stokes equations (Canuto et al., 2007), because when these equations are transformed to spectral space the pressure-velocity coupling, which is usually solved by the linear system (Poisson equation), is replaced by a matrix-vector product. Thus it is eliminated the solver of linear system and it is results in a low computational cost.

On the other hand, the disadvantage of this approach is using the Discrete Fourier Transform (DFT - Briggs e Henson, 1995) which requires periodic boundary conditions. By aiming to solve this problem Mariano et al. (2010a) proposed extend the calculus domain and impose the boundary conditions, using the IBM (Peskin, 2002), as can be seen in Fig. 1. And Fourier pseudo-spectral method can be used. Furthermore facilitates to study the flow over move and complex geometries.

In Mariano et al. (2010b) show two characteristics of method, one is the high order convergence rates (fourth order), and the other is the facility of to work with flow over complex geometries.

This work shows other step of methodology validation, and free fall particles are simulated. It is a problem of great industrial interest and a challenge for researchers in CFD, because it is a problem of Fluid Structure Interaction (FSI), *i.e.*, the particle (complex geometry) moves in the fluid flow.

The present paper is organized as follows; first, it presents the mathematical modeling and numerical approach of the proposed method. Next the results are analyzed and compared with other authors and, finally, the conclusions are presented.

2. MATHEMATICAL MODELING

In this session will be presented the mathematical model of immersed boundary method, based in Multi-Direct Forcing proposed by Wang et al. (2008) and the coupling fluid-structure interaction. After that, the equations which govern the problem will be transformed for the Fourier spectral space, due to use the properties of discrete Fourier transformed and, finally the methodology proposed by this paper, which will be presented connecting the two methodologies.

2.1. Mathematic model for the fluid

The flow is governed by conservation momentum equation (Eq. 1) and the continuity equation (Eq. 2). The information of the fluid/solid interface (domain Γ) is passed to the eulerian domain (Ω) for addition of the term source to Navier-Stokes equations. The aims of this term are represent the boundary conditions of the immersed geometry as a body force (Goldstein et al. 1993). The equations that govern the problem are presented in theirs tensorial form:

$$\frac{\partial u_l}{\partial t} + \frac{\partial (u_l u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_l} + \nu \frac{\partial^2 u_l}{\partial x_j \partial x_j} + f_l, \tag{1}$$

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{2}$$

where $\frac{\partial p}{\partial x_l} = \frac{1}{\rho_f}\frac{\partial p^*}{\partial x_l}$; p^* is the static pressure in [N/m^2]; u_l is the velocity in l direction in [m/s]; $f_l = \frac{f_l^*}{\rho_f}$; f_l^* is the term source in [N/m^3]; ρ_f is the density; ν is the cinematic viscosity in [m^2/s]; x_l is the spatial component (x,y) in [m] and t is the time in [s]. The initial condition is any velocity field that satisfies the continuity equation.

The source term is defined in all domain Ω, but has different values from zeros only in the points which coincide with the immersed geometry, it enable that the eulerian field perceives the presence of solid interface (Enriquez-Remigio and Silveira Neto, 2007).

$$f_l(\vec{x},t) = \begin{cases} F_l(\vec{x}_k,t) & \text{if } \vec{x} = \vec{x}_k \\ 0 & \text{if } \vec{x} \neq \vec{x}_k \end{cases}, \tag{3}$$

where $\vec{x}$ is the position of the particle in the fluid and $\vec{x}_k$ is the position of a point in solid interface (Fig. 1).

The boundary conditions are periodic in all directions in eulerian domain, Ω_B, as showed in Fig. 1, it is necessary due to Fourier pseudo-spectral method properties. The boundary condition in the problem simulated is imposed by direct forcing methodology in Γ_{BC}, and also the boundary conditions of bodies immersed in flow, represented by Γ_i in Fig. 1.

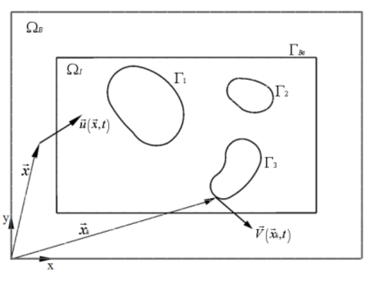


Figure 1. Schematically representation of eulerian and lagrangian domain.

Using Eq. (3) can be concluded that the field $f_l(\vec{x},t)$ is discontinuous, which can be numerically solved only when there are coincidence between the points that compose the interface domain with the compose the fluid domain. In cases there is no coincidence between these points, very frequently in the complex geometries, it is necessary to distribute the function $f_l(\vec{x},t)$ on its neighborhoods. Just by calculating the lagrangian force field $F_l(\vec{x}_k,t)$, it can be distributed and thus, transmitted the information geometry presence for eulerian domain, these functions can be found in Griffith and Peskin, (2005).

2.2. Mathematic model for the immersed interface

The lagrangian force field is calculated by direct forcing methodology, which was proposed by Uhlmann (2005). One of the characteristics of this model is that is not necessary using *ad-hoc* constants and allows the modeling non-slip condition on immersed interface. The lagrangian force $F_l(\vec{x}_k,t)$ is available by momentum conservation equation over a fluid particle that is joined in the fluid-solid interface:

$$F_l(\vec{x}_k,t) = \frac{\partial u_l}{\partial t}(\vec{x}_k,t) + \frac{\partial}{\partial x_j}(u_l u_j)(\vec{x}_k,t) + \frac{\partial p}{\partial x_l}(\vec{x}_k,t) - \nu \frac{\partial^2 u_l}{\partial x_j \partial x_j}(\vec{x}_k,t). \tag{4}$$

The values of $u_l(\vec{x}_k,t)$ and $p(\vec{x}_k,t)$ are done by interpolation of velocities and pressure, respectively, of eulerian points near the immersed interface. For lagrangian point x_k at the immersed boundary, we have:

$$F_l(\vec{X}_k,t) = \frac{u_l(\vec{x}_k,t+\Delta t) - u_l^*(\vec{x}_k,t) + u_l^*(\vec{x}_k,t) - u_l(\vec{x}_k,t)}{\Delta t} + RHS_l(\vec{x}_k,t), \tag{5}$$

where u^* is a temporary parameter (Wang, *et al.* 2008), Δt is the time step and $RHS_l(\vec{x}_k,t) = \frac{\partial}{\partial x_j}(u_l u_j)(\vec{x}_k,t) + \frac{\partial p}{\partial x_l}(\vec{x}_k,t) - \nu \frac{\partial^2 u_l}{\partial x_j \partial x_j}(\vec{x}_k,t)$. The Eq. (5) is solved by Eqs. (6) and (7) at same time step:

$$\frac{u_l^*(\vec{x}_k,t) - u_l(\vec{x}_k,t)}{\Delta t} + RHS_l(\vec{x}_k,t) = 0, \tag{6}$$

$$F_l(\vec{x}_k,t) = \frac{u(\vec{x}_k,t+\Delta t) - u_l^*(\vec{x}_k,t)}{\Delta t}, \tag{7}$$

where $u(\vec{x}_k,t+\Delta t) = U_{FI}$ is the immersed boundary velocity.

Eq. (6) is solved at eulerian domain at Fourier spectral space, *i.e.* the solution of Eq. (1) with $f_l=0$. $u_l^*(\vec{x},t)$ is interpolated for lagrangian domain, became $u_l^*(\vec{x}_k,t)$ and it is computed on Eq. (7). Then $F_l(\vec{x}_k,t)$ is smeared for eulerian mesh. Finally, the velocity is update by Eq. (8):

$$u_l(\vec{x},t+\Delta t) = u_l^*(\vec{x},t) + \Delta t. f_l. \tag{8}$$

2.3. Fluid-Structure Interaction (FSI)

The considered forces that acting on the particle drop (Wan and Turek, 2007) are the buoant (E), hydrodynamic (F_f) and particle weight (P), we can to see schematically in Figure 2. By comparing this figure with the Figure 1, we observe the eulerian domain (fluid - Ω) and the lagrangian domain (points that represent immersed particle - Γ). The particle moving is given by Newton-Euler equations:

$$M\frac{dU_l}{dt} = (\rho_p - \rho_f) g_l + F_l^f, \tag{9}$$

$$I \frac{d\omega_l}{dt} + \omega_l \times I\omega_l = T_l \, , \tag{10}$$

where M is the particle mass, I is the inertia momentum, ρ_p and ρ_f are particle and fluid density, respectively, g_l is the gravity force, U_l is the translational velocity, ω_l is the rotational velocity, F_l^f and T_l are hydrodynamic sum of forces and the torque, respectively, that act in the center of particle mass.

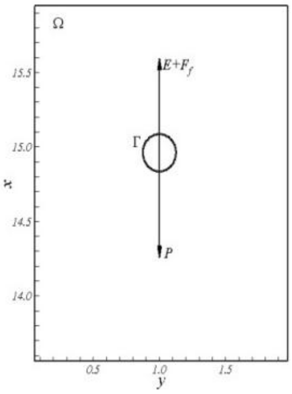


Figure 2. Forces acting over the free fall particle immersed into the fluid.

In these point it is important highlight that hydrodynamic forces and torque are calculate by sum of forces of the immersed boundary, given by Eq. 7:

$$\vec{F}_f = -\sum_{NL} \vec{F}_l = -\sum_{NL} V \rho_f \frac{\vec{U}_{FI} - \vec{U}_l^*}{\Delta t}, \tag{11}$$

$$\vec{M}_f = \sum_{NL} \vec{r} \times V \rho_f \vec{F}_l, \tag{12}$$

$$U_l^{FI} = U_l + \omega_l \times \left(XC_l - X_l \right). \tag{13}$$

where U_l and ω_l are given by Eqs. 15 and 16, respectively, XC_l is the particle center position and X_l are the positions of lagrangian points that define the particle.

2.4. Fourier Pseudo-Spectral Method

By defining the equations that govern the flow through IBM, the next step is transforming them to the Fourier spectral space. It applies the Fourier transform in the continuity equation, Eq. (2), we obtain:

$$ik_j \hat{u}_j = 0. \tag{14}$$

According to analytic geometry the scalar product between two vectors is null, if both are just orthogonal. Therefore, from Eq. (14), the wave number vector k_j is orthogonal to transformed velocity $\hat{u}_j$. The plane of divergent free (plane π) is defining, perpendicular to wave number vector $\vec{k}$ and thus, transformed velocity vector $\hat{\vec{u}}\left(\vec{k}, t\right)$ belongs to plane π.

Now applying the Fourier transform in the momentum equation Eq. (2), we obtain:

$$\frac{\partial \hat{u}_l^*}{\partial t} + ik_j \widehat{u_l^* u_j^*} = -ik_l \hat{p} - \nu k^2 \hat{u}_l^*, \tag{15}$$

where k^2 is the square norm of wave number vector, *i.e.* $k^2 = k_j k_j$. and $i = \sqrt{-1}$

In agreement of plane π definition, each one of the terms of Eq. (10) assumes a position related to it: the transient term $\frac{\partial \hat{u}_l^*}{\partial t}$ and the viscous term $\nu k^2 \hat{u}_l^*$ belong to the plane π. The gradient pressure term is perpendicular to plane π, and non-linear, $ik_j \widehat{u_l^* u_j^*}$, a priori, it is not known in which position it can be found in relation to plane π. By joining the terms of Eq. (15) and observing the definition of plane π, we have found that:

$$\underbrace{\left[\frac{\partial \hat{u}_l^*}{\partial t} + \nu k^2 \hat{u}_l^*\right]}_{\in \pi} + \underbrace{\left[ik_j \widehat{u_l^* u_j^*} + ik_l \hat{p}\right]}_{\in \pi} = 0. \quad \Rightarrow \tag{16}$$

To close Eq. (16) is needed that the non-linear and the force field terms are over plane π. For that, it is utilized projection tensor definition (Canuto *et al.*, 2007), which projects any vector over it. Therefore, applying this definition on the right hand side of the sum done in Eq. (16):

$$\left[ik_j \widehat{u_j^* u_l^*} + ik_l \hat{p}\right] = \wp_{lm}\left[ik_j \widehat{u_m^* u_l^*}\right]. \tag{17}$$

The parcel of the gradient pressure field is orthogonal to plane π, then, it is zero after to be projected, disentailing from calculates of Navier-Stokes equations in the spectral space. The pressure field can be recovered at the pos-processing manipulating Eq. (17) (Mariano, 2007).

Other important point is non-linear term, in which appears the product of transformed functions, in agreement with Fourier transformed properties, this operation is a convolution product and its solution is given by convolution integral, this is solved by pseudo-spectral Fourier method (Canuto *et al*., 2007). Therefore the momentum equation in the Fourier space, using the method of the projection, assumes the following form:

$$\frac{\partial \hat{u}_l^*(\vec{k},t)}{\partial t} + \nu k^2 \hat{u}_l^*(\vec{k},t) = -ik_j \wp_{lm} \int_{\vec{k}=\vec{r}+\vec{s}} \hat{u}_m^*(\vec{r},t)\,\hat{u}_l^*(\vec{k}-\vec{r},t)\,d\vec{r}. \tag{18}$$

Non-linear term can be handed by different forms: advective, divergent, skew-symmetric, or rotational (Canuto *et al*., 2006), in spite of being the same mathematically, they present different properties when discretized. The skew-symmetric form is more stable and present best results, but is twice more onerous that the rotational form. However this inconvenience can be solved using the alternate skew-symmetric form, it is consisting in alternate between the advective and divergent forms in each time step (Souza, 2005), it is proceeding adopted for this paper.

For all types of handing the non-linear term is necessary solve the convolution integral, but its numerical solution is computational expensive, then the pseudo-spectral method is used, *i.e.* calculates the velocity product in the physical space and transforms this product for the spectral space.

When solved numerically the Navier-Stokes equations with the Fourier spectral method using the Discrete Fourier Transform (DFT), which is define by Briggs and Henson (1995):

$$\hat{f}_k = \sum_{n=-N/2+1}^{N/2} f_n e^{\frac{-i2\pi kn}{N}}, \tag{19}$$

when k is wave number, N is number of meshes points, n get the position x_n of collocation points ($x_n=n\Delta x$).

The DFT restriction is periodic boundary conditions, by limiting the use of Fourier spectral transformed for CFD problems. The advantage is low computational cost gives by Fast Fourier Transform (FFT) (Cooley and Tukey, 1965), which solves the DFT (Eq. 19) of a way very efficiently, order $O(N\log_2 N)$. For systems with many collocation points, *e.g.* three-dimensional problems, the spectral method is very cheap when compared with another conventional high order methodology. Two examples of use this method are simulations of periodic temporal jets and turbulence isotropic.

2.5. Proposed Methodology: IMERSPEC

The algorithm of IMERSPEC method purposed is:

1) Solve the Eq. (18) in Fourier spectral space and obtain the temporal parameter $\hat{u}^*\left(\vec{k},t\right)$, using the low dispersion and low storage Runge-Kutta method proposed by Berland *et al.*, (2006) is used;
2) Use the Inverse Fast Fourier Transformer in $\hat{u}^*\left(\vec{k},t\right)$ and obtain $u^*(\vec{x},t)$ at physic space in the domain *Ω*;
3) Interpolate $u^*(\vec{x},t)$ for the lagrangian domain by cubic function proposed by Griffith and Peskin *et al.* 2005, and obtain $u^*(\vec{x}_k,t)$;
4) Calculate the lagrangian force, $F_l(\vec{x}_k,t)$, by Eq. (7);
5) We calculate the sum Eqs. (12), (13) and (14).
6) Update particles velocity and position;
7) Distribute the $F_l(\vec{x}_k,t)$ by cubic function proposed by Griffith and Peskin 2005, and obtain $f(\vec{x}_k,t)$ in eulerian domain;
8) Update the eulerian velocity, $u(\vec{x},t)$ by Eq. (8) and transformed it using FFT for spectral space, $\hat{u}^*\left(\vec{k},t\right)$, returned by step 1.

3. RESULTS

To validate the proposed methodology and developed code, two classical problems used in CFD were chosen. The first one is the flow over a cylinder, which is a benchmark of CFD. This case allowed shows the solution of incompressible two-dimensional Navier-Stokes equations using Fourier pseudo-spectral method with non-periodic boundary conditions imposed by immersed boundary. The second one, is the free fall particle, for to validate and to show the capability of the method for fluid-structure problems.

3.1. Flow over a cylinder

It is generate an inlet profile flow with velocity U_∞ in [*m/s*], the flow cross the section of a circular cylinder (Fig. 3) and verify the drag (*Cd*) (Eq. 20) and lift (*Cl*) (Eq. 21) coefficients, these variables determine the forces that act on bodies immersed in flow, the drag coefficient determines the resistance force of the fluid on the immersed body, while the lift coefficient determines the force that there is in perpendicular direction to incoming flow, a interesting problem in aeronautical engineering is the optimization of airfoils, that consist in maximize the lift and minimized the drag of the airfoil profiles. Other parameter analyzed is the Strouhal number (*St*) (Eq. 22) which determines the non-dimensional vortex shedding, it is important to solve problems of fluid-structure, for example, pillars of bridges or aircraft wings, submitted to a flow, if the frequency of vortex shedding is close to the natural frequency is extremely damaging to this structures.

$$Cd = \frac{-2\sum F_x}{\rho A_y U_\infty^2}, \tag{20}$$

$$Cl = \frac{-2\sum F_y}{\rho A_x U_\infty^2}, \tag{21}$$

$$St = \frac{freq.D}{U_\infty}, \tag{22}$$

where: *Fx* and *Fy* are the forces estimated at each lagrangian point with Eq. (7) in [*N*]; *Ax* and *Ay* are projected frontal area in direction *x* and *y*, respectively . In bidimensional case these areas are given in [m^2] considered the perpendicular dimension of surface equal a unity, *D* is the characteristic diameter and *freq* is the vortex shedding frequency downstream of cylinder. The domain of all cases have been simulated is $6\pi \times 2\pi$ [m^2] and has been discretized with *384 x 128* collocation points. The cylinder has a diameter of *D=0,785* [*m*], with *64* lagrangian collocation points. The cylinder position in domain is shown in Fig. 3.

At the top and bottom boundary conditions are periodicity. The inflow condition is a uniform profile of velocity ($U_\infty=1,0$ *m/s*). Other important parameter is the Reynolds number, that is *Re=100*, with the Reynolds number is possible to determine the viscosity of the fluid:

$$\nu = \frac{U_\infty D}{\mathrm{Re}} \tag{23}$$

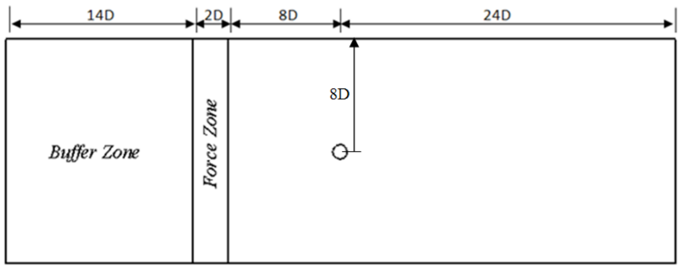


Figure 3. Calculus domain – circular cylinder.

It also imposed a buffer zone:

$$ZA = \phi(Q_l - Qt_l) \tag{24}$$

where Q is the problem solution, that is, u and v, Qt is the target solution, i.e. the solution is required in the final buffer zone, in this case, the target solution is an uniform profile U_∞, and Φ is a parameter of stretching vortex, and it is calculated by Eq. (25):

$$\phi_\eta = \beta\left(\frac{x_\eta - x_{za}}{x_f - x_{za}}\right)^\alpha , \tag{25}$$

where $\alpha=3.0$ and $\beta=1.0$ (Uzun, 2003), x_{za} and x_f are the beginning and the ending of buffer zone, respectively, x_η is the generic position.

The force zone or porous medium is a range of thickness 2D, where is imposed the inflow profile by direct forcing, in order to aligned the streamlines. The Fig. 4 shows vorticity isocontours (-*1,0<w<1,0*) at time *t*=250.*

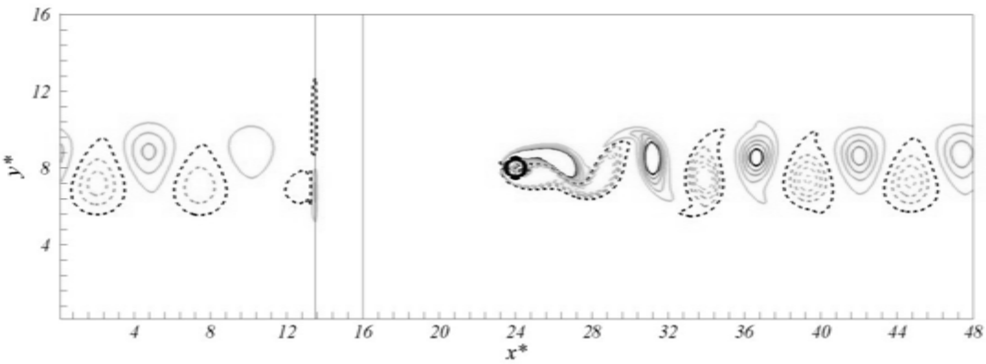


Figure 4. Isocontours of vorticity (-1,0<w<1,0) at *Re=100* in *t*=250*. - negative vortice; -- positive vorticity.

Table 1. Comparison of drag coefficient and Strouhal number.

	Lima e Silva *et al.*(2003)			**Lai and Peskin (2000)**			**Xu and Wang (2005)**			**Le *et al.* (2007)**			**Present work**		
Re	*Cd*	*Cl*	*St*	*Cd*	*Cl*	*St*	*Cd*	*Cl*	*St*	*Cd*	*Cl*	*St*	*Cd*	*Cl*	*St*
100	1,39	0,20	0,160	1,44	0,33	0,165	1,42	0,34	0,171	1,39	0,34	0,160	1,45	0,35	0,175
150	1,37	0,25	0,175	1,47	0,58	0,184							1,37	0,49	0,200
200							1,42	0,66	0,202	1,38	0,68	0,192	1,27	0,47	0,213
300	1,22	0,27	0,190										1,08	0,39	0,221

Fig. 6 show the vorticity field for different time steps.

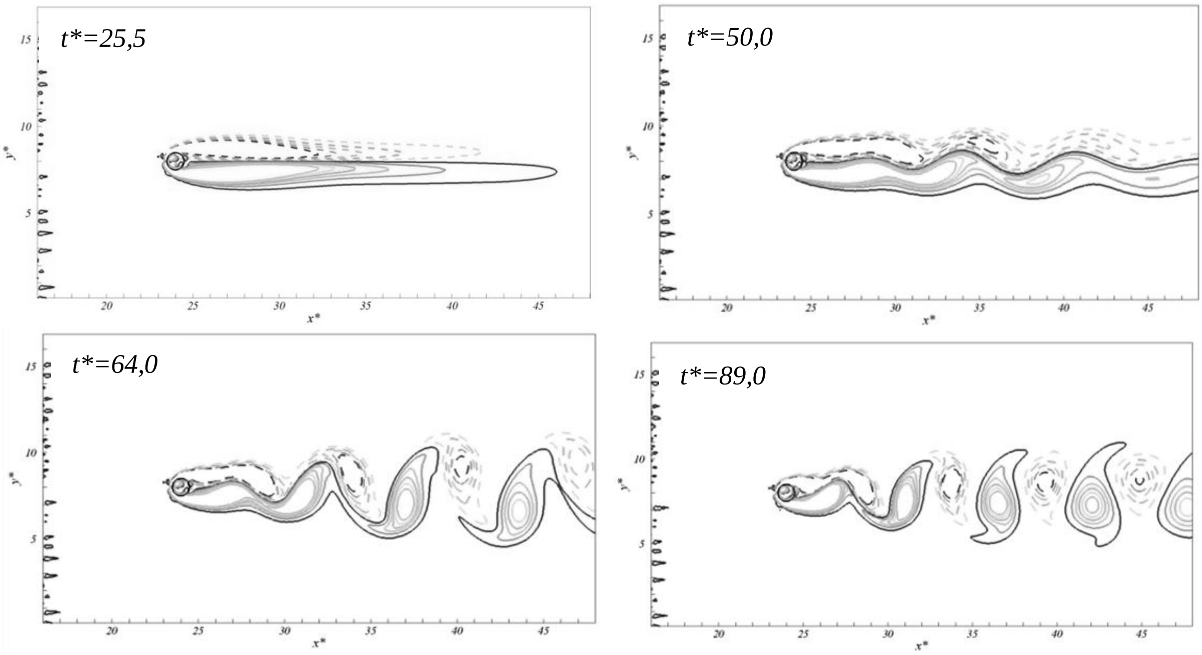


Figure 6. Temporal evolution of vorticity field at *Re=100*. **-** negative vortice; **--** positive vorticity.

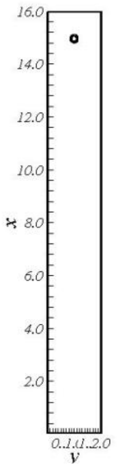
The first time at Fig. (6), in the beginning of simulation, arise two recirculation bubbles, in *t*=50* there is a formation of instability, and in the sequence appears the vortex shedding, *t*=64,0* and *t*=89,0*. Tab. 1, shows the comparison of *Cd*, *Cl* and *St* for different Reynolds numbers and among different authors.

3.2. Free fall particles

The simulations performed in this section have objective to validate the modeling of fluid-structure interaction, as well as to show the potentiality of IBM. The problem consist in analyze the free fall of a solid and indeformable particle, that to Begin at rest, into fluid attain a terminal velocity, that is expressed in function of Reynolds number:

$$\mathrm{Re} = \frac{\rho_p D_p \sqrt{U^2(t) + V^2(t)}}{\mu} \tag{26}$$

The calculus domain and the fluid and particle properties are show at Fig. 9 and are exactly to works of the Wan and Turek (2007), Ulhmann (2005) and Wang *et al.* (2008), where D_p is the particle diameter.



$D_p = 0,25\,[m]$

$\rho_p = 1,25 \; e \; 1,50 \; [kg/m^3]$

$\rho_f = 1,00 \; [kg/m^3]$

$\mu_f = 0,1 \; e \; 0,01 \; [kg/(ms)]$

$g = 9,81 \; [m/s^2]$

Figure 8. Calculus Domain of free fall particles simulations.

All the simulations of this section use the forth order low-dispersion and low-diffusion Runge-Kutta purposed by Berland *et al.* (2006) and a criterion CFL=0.25. For temporal evolution of particle we use the second order Adams-Bashfort method. The Fig. 9 shows the free fall particle in different times.

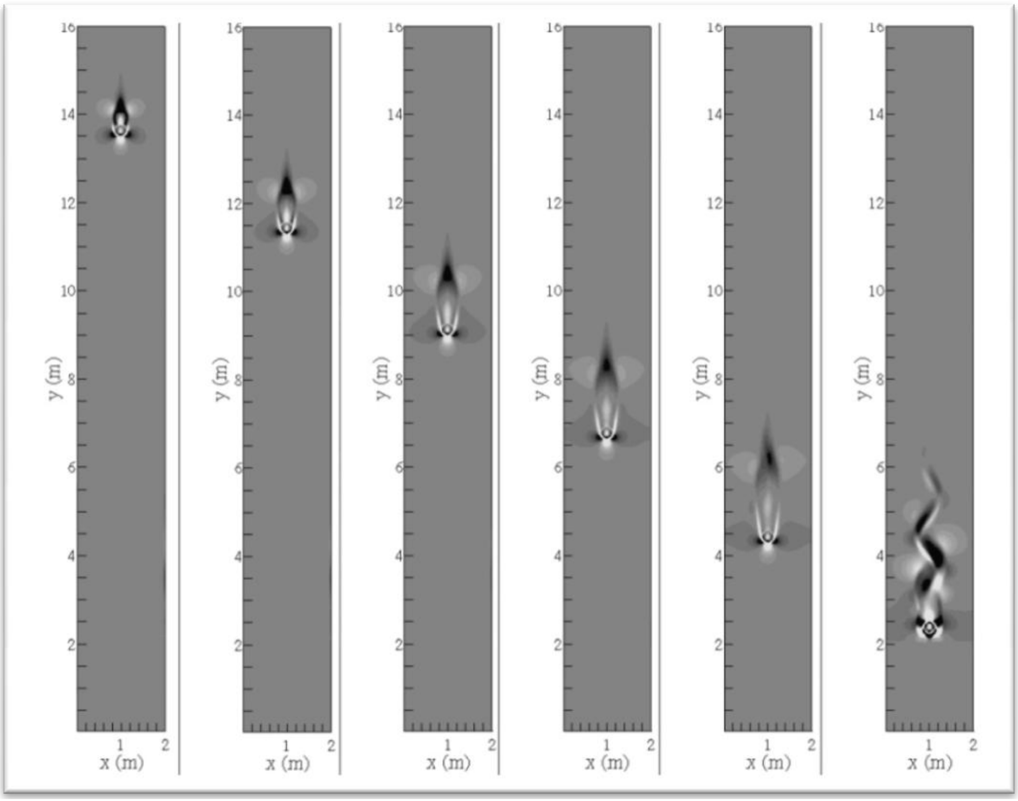


Figure 9. Temporal evolution of vorticity field of free fall particle with $\nu=0.01$ and $\rho_p=1.25$.

In Fig. 9 is possible to see that in fall begins there is formation of two symmetric recirculation bubbles. They grow up until to become unstable and formed the vortex shedding. In Tab. 2 is done a comparison of terminal Reynolds among different authors. There is a good agreement among the results, by showing the capability of method.

Table 2. Comparison of terminal Reynolds number of different simulations.

Authors	ν=0,1 ρ_p=1,25	ν=0,1 ρ_p=1,50	ν=0,01 ρ_p=1,25	ν=0,01 ρ_p=1,50
Wan e Turek (2007)	17,15	32,76	270,77	465,52
Wang et al. (2007)	17,307			503,38
Ulhmann (2005)				495,00
Present work	17,73	33,62	293,96	527,51

4. CONCLUSIONS

The motivations of this paper are improved the pseudo-spectral methodology, that is high order method and low computational cost, but restrained to periodic boundary conditions. Looking forward this aim a fusion of immersed boundary and the classic Fourier pseudo-spectral method was made.

The Fourier pseudo-spectral method allows solves the incompressible Navier-Stokes equations with the high order accuracy. In case where the equations to be solved are periodic the methodology accuracy order is high. Other great vantage is the computational cost when compared another high order methods, because the pressure disentail and the use FFT algorithm.

In the simulations of flowing over a circular cylinder and free fall particle it is possible to observe the drag and lift coefficients, Sthrouhal number and terminal Reynolds numbers are similar to another authors, and the vortex shedding are reasonable. The disadvantages are the requirement of using the buffer zone and the accuracy of methodology Fourier pseudo-spectral is penalized, but the computational cost is still low.

5. ACKNOWLEDGEMENTS

The authors thank the College of Engineering Mechanical (FEMEC) of the University Federal of Uberlândia (UFU), Capes, FAPEMIG and CNPq for financial support.

6. REFERENCES

Berland, J., Bogey, C. and Bailly, C., 2006, "Low-dissipation and low-dispersion fourth-order Runge–Kutta algorithm", Computers and Fluids, Vol.35, pp. 1459-1463.

Briggs W.L. and Henson, V.E., 1995, "The DFT", SIAM.

Canuto, C., Hussaini, M.Y., Quarteroni, A. and Zang, T.A., 2006, "Spectral methods: fundamentals in single domains", Ed: Springer-Verlag, New York, United States, 563p.

Canuto, C., Hussaini, M.Y., Quarteroni, A. and Zang, T.A., 2007, "Spectral methods: evolution to complex geometries and applications to fluid dynamics", Ed: Springer-Verlag, New York, United States, 596p.

Cooley, T.W. and Tukey, J.W., 1965, "An algorithm for the machine calculation of complex Fourier series", Mathematics Computation, Vol.19, pp. 297-301.

Enriquez-Remigio, S. and Silveira-Neto, A., 2007, "A new modeling of fluid-structure interaction problems through immersed boundary method/virtual physical model (IBM/VPM)", Proceedings of the 19th Brazilian Congress of Mechanical Engineering, Vol.1, pp. 1-10.

Goldstein, D., Handler, R. and Sirovich, L., 1993. "Modeling a no-slip flow with an external force field". Journal Computational Physics, Vol. 105, pp. 354-366.

Griffith, B.E. and Peskin, C.S., 2005, "On the order of accuracy of the immersed boundary method: higher order convergence rates for sufficiently smooth problems", Journal Computational Physics, Vol.208, pp. 75-105.

Lai M.C. and Peskin, C.S., 2000, "An immersed boundary method with formal second order accuracy and reduced numerical viscosity", Journal of Computational Physics, Vol. 160, pp. 705–719.

Le, D.V., Khoo, B.C. and Lim, K.M., 2008, "An implicit-forcing immersed boundary method for simulating viscous flows in irregular domains". Computer Methods in Applied Mechanics and Engineering, Vol.197, pp. 2119-2130.

Lima e Silva, A., Silveira-Neto, A. and Damasceno, J., 2003, "Numerical simulation of two dimensional flows over a circular cylinder using the immersed boundary method", Journal of Computational Physics, Vol.189, pp. 351–370.

Mariano, F.P., 2007, "Simulação de escoamentos não periódicos utilizando as metodologias pseudo-espectral de Fourier e da fronteira imersa acopladas". Dissertação de Mestrado em Engenharia Mecânica, Faculdade de Engenharia Mecânica, Universidade Federal de Uberlândia.

Mariano, F.P., Moreira, L.Q., Silveira-Neto, A., da Silva, C.B. and Pereira, J.C.F., 2010, "A new incompressible Navier-Stokes solver combining Fourier pseudo-spectral and immersed boundary methods", Computer Modeling in Engineering Science, Vol.1589, pp. 1-35.

Mariano, F.P., Moreira, L.Q. and Silveira Neto, A., 2010, "Mathematical Modeling of non-periodic flows using Fourier pseudo-spectral and immersed boundary methods", Proceedings of the 5th European Conference on Computational Fluid Dynamics, Vol.1, Lisbon, Portugal, pp. 1-17.

Peskin, C.S., 2002, "The immersed boundary method", Acta Numerica, Vol.11, pp. 479–517.

Souza, A.M., 2005. "Análise numérica da transição à turbulência em escoamentos de jatos circulares livres", Tese de Doutorado em Mecânica, Faculdade de Engenharia Mecânica, Universidade Federal de Uberlândia.

Uhlmann, M., 2005, "An immersed boundary method with direct forcing for the simulation of particulate flows", Journal of Computational Physics, Vol.209, pp. 448-476.

Uzun, A., 2003, "3-D Large-eddy simulation for jet aeroacoustics", Tese de Doutorado em Engenharia Mecânica, Purdue University.

Wan, D. and Turek, S., 2007, "An efficient multigrid-FEM method for the simulation of solid–liquid two phase flows", Journal of Computational and Applied Mathematics, Vol. 203, pp. 561-580.

Wang Z., Fan J. and Luo, K., 2008, "Combined multi-direct forcing and immersed boundary method for simulating flows with moving particles", International Journal of Multiphase Flow, Vol.34, pp. 283-302.

Xu, S. and Wang, Z.J., 2006, "An immersed interface method for simulating the interaction of a fluid with moving boundaries", Journal of Computational Physics, Vol. 216, pp. 454-493.

5. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.