

SIMULATION OF FLOWS OVER SQUARE CYLINDER USING THE IMERSPEC METHODOLOGY: FOURIER PSEUDO SPECTRAL 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 Transferência de Calor e Massa - LTCM
Av: João Naves de Ávila, 2160
Campus Santa Mônica Bl: 1M - sala: 129
CEP: 38400-902 Uberlândia-MG-Brasil

Abstract. *To solve the Navier-Stokes equations pseudo spectral methods are viable alternative, because its provide an excellent numerical accuracy and it becomes very efficient when Fast Fourier Transform algorithm is used. It presents a low computational cost when compared to anothers high-order methods. Nevertheless this method can be only applied to solve periodic flows,aiming to solve this restriction, the immersed boundary method is being used with pseudo spectral method. The present work is a evolution of work Mariano et al. (2007)with the new case test, the flow over a square cylinder at differents Reynolds' numbers, in order to continue the validation the methodologie and the computational code, demonstrating the possibility of solving non-periodic flows with the Fourier pseudo spectral method.*

Keywords*: Computational Fluids Dynamics, Fourier pseudo-spectral method, immersed boundary method, flow over square cylinder*

Phenomena involving aeroacustic, transition to turbulence and combustion are problems that modern engineering aim to understand, among other manners, using techniques of the Computational Fluids Dynamics (CFD). In the case of the aeroacustic is important to use a method that captures the sound pressure waves. In phenomena involving transition to turbulence is necessary to study the small instabilities that become the flows turbulent. In the combustion, exists processes that involve the small edges of the turbulent flow. In these problems the CFD uses methods of high order accuracy to obtain results for analyses which represent the in fact physics phenomena mentioned.

The high order methods provide an excellent accuracy, for example: the methods of high order finite differences and the compact schemes, but, on the other hand, they have as disadvantaged the computational expensive cost in comparison to the conventional methodologies. With the advent of the spectral methods joining high accuracy with low computational cost became possible. This low cost is given by the Fast Fourier Transformed (FFT), since the cost of a problem resolution with finite differences is the order of $O(N^2)$, where N is the number of the grid points, the cost of the FFT is of $O(N\log_2 N)$ (Canuto *et al*, 1988). In addition, it was also developed the projection method (Silveira-Neto, 2002; Souza, 2005 and Mariano, 2007), which disentails pressure field of the Navier-Stokes equation calculates in the spectral space. Using the projection process is not necessary to calculate the Poisson equation, like it is done by the conventional methodologies. Normally, solving this equation is the most expensive part of a CFD code. The disadvantage of the spectral methodology is the difficulty to work with complex geometries and boundary conditions.

One of the most practical methodologies to work with complex geometries is the Immersed Boundary (IB) (Peskin, 1972). It is distinguished by the imposition of a term source, which has the role of a body force imposed in the Navier-Stokes equation to represent a virtual immersed body in the flow (Goldstein *et al*, 1993). This facilitates for represent any geometry, whether it is complex or in movement.

The methodology, presented in this paper, is an evolution of the work of Mariano et al. (2007), using the Fourier pseudo-spectral method connected in the immersed boundary method. It is proposed to simulate flows with non-periodic boundary conditions making use of the term source of the immersed boundary.

First, it will be demonstrated the transformation of the Navier-Stokes equations for the Fourier spectral space, as well as the imposition of the source term. In the second part, the details of numerical implementation of the computational code developed will be demonstrated. Finally, the results of flow over a square cylinder will be shown which is a non-periodic problem solved by the Fourier spectral method, where the boundary conditions had been imposed through the force field of the immersed boundary. Beside that, concepts of buffer zone and filtering process will be explored.

2. MATHEMATICAL MODELING

In this session will be presented the classic mathematical model of the immersed boundary proposed by Lima e Silva (2003), which calculates the term source through the Virtual Physical Model, after that, the equations that govern the problem will be transformed for the Fourier spectral space using the properties of the discrete Fourier transformed and, finally, the methodology proposed by this paper will be presented, which connect the two methodologies.

2.1. Mathematic model for the fluid

The Immersed Boundary methodology (Peskin, 1972) consists in working with two independents meshes: the eulerian mesh, where the fluid equations are solved and the lagrangian mesh, which represents the solid interface immersed in fluid. The eulerian mesh is cartesian and fixed, and is simulate as if it is completely full of fluid. The flow is governed by conservation momentum equation and the continuity equation. The information of the fluid/solid interface is passed to the eulerian mesh for the addition of the term source to the Navier-Stokes equations. This term play a role of a body force that represents the boundary conditions of the immersed geometry (Goldstein, 1993). The equations that govern the problem are presented in its 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 $\frac{\partial p}{\partial x_l} = \frac{1}{\rho}\frac{\partial p^*}{\partial x_l}$; p^* is the static pressure in [N/m^2]; u_l is the velocity in the l direction in [m/s]; $f_l = \frac{f_l^*}{\rho}$; f_l^* is the term source in [N/m^3]; ρ 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 boundary conditions are imposed in a classical way and the initial condition is any velocity field that satisfies the continuity equation.

The source term is defined in all domain, but presents values different from zeros only in the points that coincides with the immersed geometry, enabling that the eulerian field perceives the presence of the solid interface (Enriquez-Remigio, 2000).

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

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

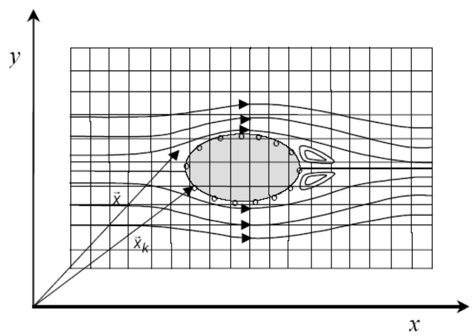

Figure 1: Schematically representation of eulerian and lagrangian domain

Using this definition 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 points that compose the fluid domain. In cases that there are not coincidence between this points, very frequently in the complex geometries, it is necessary to distribute the function $f_l(\vec{x},t)$ on its neighborhoods (Unverddi and Triggvason, 1992). Just by calculating the lagrangian force field $F_l\left(\vec{x}_k,t\right)$, it can be distribute and thus, to transmit the information geometry presence for the eulerian meshes.

2.2. Mathematic Model for the immersed interface

In this paper, the lagrangian force field is calculate by the Virtual Physical Model (VPM), which was proposed by Lima e Silva et al. (2003). One of the characteristics of this model is that is not necessary the use of ad-hoc constants. It is based in the Newton second law and allows the modeling non-slip condition on the immersed interface. The lagrangian force $F_l\left(\vec{x}_k,t\right)$ is available by momentum conservation equation over a fluid particle that is joined in the fluid-solid interface, equation (4):

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

The values of $u_l\left(\vec{x}_k,t\right)$ and $p\left(\vec{x}_k,t\right)$ are done by interpolation of the velocities and pressure, respectively, of the eulerian points near the immersed interface.

2.3 Fourier Transforms

By defining the equations that governs the flow through immersed boundary method, the next step is to transform them to the Fourier spectral space. First applies the Fourier transform in the continuity equation (2):

$$ik_j\hat{u}_j=0 \quad (5)$$

From the analytic geometry is known that the scalar product between two vectors is null, just if both are orthogonal. Therefore, from the equation (5), we have that the wave number vector k_j is orthogonal to transform velocity $\hat{u}_j$. So, is defining the plane π (Silveira-Neto, 2002), perpendicular to wave number vector $\vec{k}$ and thus, the transformed velocity vector $\hat{\vec{u}}\left(\vec{k},t\right)$, belongs to the plane π.

Now applies the Fourier transform in the momentum equation (2):

$$\frac{\partial\hat{u}_l}{\partial t}+ik_j\widehat{u_l u_j}=-ik_l\hat{p}-\nu k^2\hat{u}_l+\hat{f}_l \quad (6)$$

where k^2 is the square norm of the wave number vector, i.e. $k^2=k_jk_j$.

In agreement of the plane π definition, each one of the terms of the equation (6) assume 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 the plane π, and the non-linear, $ik_j\widehat{u_l u_j}$, and the force filed, $\hat{f}_l$, a priori, it is not known in which position it can be found in relation to plane π. By jointing the terms of the equation (6) and observing the definition of plane π, we have that:

$$\underbrace{\left[\frac{\partial\hat{u}_l}{\partial t}+\nu k^2\hat{u}_l\right]}_{\in\pi}+\underbrace{\left[ik_j\widehat{u_l u_j}-\hat{f}_l+ik_l\hat{p}\right]}_{\in\pi}=0 \quad \Rightarrow \quad (7)$$

To satisfy the equation (7), it is needed that the non-linear and the force field terms are over the plane π. For that, it is utilized the projection tensor definition (Canuto *et al*, 2002), which projects any vector over it. Therefore, by applying this definition on the right hand side of the sum done in the equation (7):

$$\left[ik_j \widehat{u_l u_j} + ik_l \hat{p} - \hat{f}_l \right] = \wp_{lm} \left[ik_j \widehat{u_m u_j} - \hat{f}_m \right] \tag{8}$$

It must be noticed that the parcel of the pressure field is orthogonal to the plane π, so, it is null after to be projected, disentailing from the calculates of Navier-Stokes equations in the spectral space. The pressure field can be recovered at the pos-processing manipulating the equation (7) (Mariano *et al*, 2007).

Other important point is about the non-linear term, in which appears the product of transformed functions, in agreement with the Fourier transformed properties, this operation is a convolution product and its solution is given by convolution integral. Therefore the momentum equation in the Fourier space, using the method of the projection, assumes the following form:

$$\frac{\partial \hat{u}_l\left(\vec{k}\right)}{\partial t} + \nu k^2 \hat{u}_l\left(\vec{k}\right) = \wp_{lm} \hat{f}_m - ik_j \wp_{lm} \int_{\vec{k}=\vec{r}+\vec{s}} \hat{u}_m\left(\vec{r}\right) \hat{u}_j\left(\vec{k}-\vec{r}\right) d\vec{r} \tag{9}$$

2.4 Coupling of the methods in the Fourier spectral space

The necessary derivatives for the solver of the lagrangian force, equation (4), are make generating a new field of velocity

$$u_l^F = \begin{cases} u_l \text{ se } \vec{x} \neq \vec{x}_k, \\ u_k \text{ se } \vec{x} = \vec{x}_k, \end{cases} \tag{10}$$

where u_k is the fluid velocity in the points of the immersed boundary and, u_l^F is the eulerian velocity field modified by imposed boundary conditions. After, this field is transformed to Fourier space and it is calculate the derivatives of the langrangian force field using the Fourier transformed propeties:

$$\hat{F}_l\left(\vec{k},t\right) = \frac{\partial \widehat{u_l^F}}{\partial t} + \frac{\widehat{\partial (u_l^F u_j^F)}}{\partial x_j} + ik_l \widehat{p^F} - \nu k^2 \widehat{u_l^F} \tag{11}$$

After calculate $\hat{F}_l\left(\vec{k},t\right)$ make the inverse Fourier transformed in this field using the definition done in the equation (3) it get eulerian force field, $f_l\left(\vec{x},t\right)$, in the physic space. Last, transformed it for the spectral space, $\hat{f}_l\left(\vec{k},t\right)$ and added it in the equation (9).

3. NUMERICAL METHOD

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) how:

$$\hat{f}_k = \sum_{n=-N/2+1}^{N/2} f_n e^{\frac{-i2\pi kn}{N}} \tag{12}$$

when k is the wave number, N is the number of meshes points, n get the position x_n of the collocation points ($x_n = n\Delta x$) and $i = \sqrt{-1}$.

The DFT has the restriction of the using periodics boundary conditions, by limiting the use of Fourier spectral transformed for the problems that satisfy this boundary conditions.

Same with this restriction, the Fourier spectral method is very used for example in the simulations of temporal jets and turbulence isotropic, because its low computational cost gives by Fast Fourier Transform (FFT) (Cooley and Tukey, 1965). This algorithm solver the DFT with of the way very efficiently $O(N\log_2 N)$, whereas the calculation of (12) is of $O(N^2)$, where N is the collocation points number. For the systems with many collocation points, for example: tridimensionals problems, the spectral method is very cheap when compared with another conventional high order methodologies.

3.1. Treatment of the non-linear term

The non-linear term can be handing by different forms: advective, divergent, skew-symmetric, or rotational (Canuto *et al*.,1988 and Souza, 2005), in spite of being the same mathematically, they present different properties when discrteized. The skew-symmetric form is more stable and present the best results, but is twice more onerous that the rotational form. However this inconvenience can be solved using the alternate skew-symmetric form, this is consists in aternate between the advective and divergent forms in each time step (Zang, 1987), this is proceeding adopted for this paper.

For all forms of the handing the non-linear term is necessary solve the convolution integral, but the numerical solution of this integral is computational expensive. So, using by to solver this problem, the pseudo-spectral method, which calculates the velocity product in the physical space and transformed this product for the spectral space Souza (2005).

3.2. Filtering

The Fourier spectral method is influenced by discontinuous fields, because they yield the Gibbs phenomenon. It introduces errors in the high frequencies losing the spectral accuracy. In two steps of the methodology proposed by the present work, equations (3) and (10), appear the discontinuous fields, aiming to avoid the Gibbs phenomenon utilize filters:

$$\hat{f}\left(\vec{k},t\right)_{filtered} = \sigma(\theta)\hat{f}\left(\vec{k},t\right) \qquad (13)$$

where $\sigma(\theta)$ is the filter function. In this paper use the sharpened raised cosine filter (14) proposed by Kopriva (1986):

$$\sigma(\theta) = \sigma_0^4\left(35 - 84\sigma_0 + 70\sigma_0^2 - 20\sigma_0^3\right) \qquad (14)$$

where σ_0 is done by (15).

$$\sigma_i = 1/2\left(1 + \cos\theta_i\right) \qquad (15)$$

where: $\theta_i = Lk_i / N$;

4. RESULTS

To validate the proposed methodology and developed code, a classical problem used in CFD was chosen: the flow over a square cylinder, which is a benchmark of Computational Fluid Dynamics. This case allowed to show the solution of incompressible two-dimensional Navier-Stokes equations using Fourier pseudo-spectral method with non-periodic boundary conditions imposed by immersed boundary.

4.1 Flow over a square cylinder

An inlet profile flow with velocity U_∞ in [*m/s*] was generated, the flow cross the section of a square cylinder (Figure 2) and examines the drag (*Cd*) (Equation 16) and lift (*Cl*) (Equation 17) coefficients, these variables determine the force that acts on bodies immersed in the flow, the drag coefficient determines the resistance force of the fluid on the body immersed, while the lift coefficient determines the force that exists in the perpendicular direction of the incoming flow. An interesting problem in aeronautical engineering is the optimization of airfoils, that consists in maximize the lift and minimize the drag of the airfoil profiles. Another parameter analyzed is the Strouhal number (*St*) (Equation 18) which determines the non-dimensional vortex shedding, it that is important to solve problems of fluid-structure, for example, pillars of bridges, or wings of aircraft, submitted to a flow, if the frequency of vortex shedding is close to the natural frequency is extremely damaging to this structures.

$$Cd = \frac{2\sum F_x}{\rho A_y U_\infty^2}, \qquad (16)$$

$$Cl = \frac{2\sum F_y}{\rho A_x U_\infty^2},\qquad (17)$$

$$St = \frac{freq.D}{U_\infty},\qquad (18)$$

where: *Fx* and *Fy* are the forces estimated at each lagrangian point with equation (11) in [*N*]; *Ax* and *Ay* are the projected frontal area in direction *x* and *y*, respectively . In the bidimensional case, these areas are given in [m^2] considering the perpendicular dimension of surface equal to one, *D* is the characteristic diameter in [*m*] and *freq* is the vortex shedding frequency downstream of the cylinder.

The domain of the case has been simulated is 4π *x* 2π [m^2] and it is discretized with *1024 x 512* collocation points. The cylinder has a diameter of the *D=0,20* [*m*], with *64* lagrangian collocation points, (*16* each face). The cylinder center position in domain is *x=3,142* [*m*] and *y=3,142* [*m*].

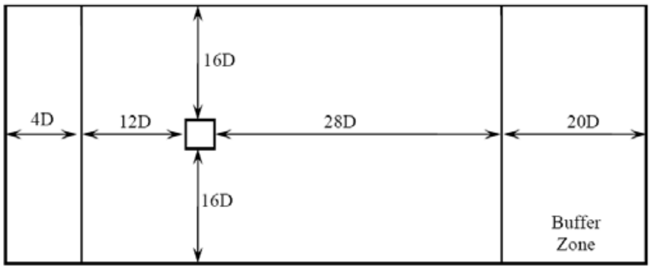

Figure 2: Calculus domain – square cylinder.

All boundary conditions are of periodicity, but an uniform profile of velocity (U_∞*=1,0 m/s*) is imposed, through the field force of the immersed boundary methodology, at *x=0,785* [*m*]. Another important parameter is the Reynolds number, that is *Re=100* based at the cylinder diameter. In outflow condition it is imposed a buffer zone:

$$ZA = \phi(Q_l - Qt_l)\qquad (19)$$

where *Q* is the problem solution, that is, *u* and *v*, *Qt* is the target solution, represents, the solution required in the final buffer zone, in this case, the target solution is the uniform profile U_∞, and Φ is the parameter of stretching vortex, calculated by (20):

$$\phi_\eta = \beta\left(\frac{x_\eta - x_{za}}{x_f - x_{za}}\right)^\alpha,\qquad (20)$$

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

At figure 3 is shown the vorticity field of the simulation at *Re=100* in $t^*= 500$, that enables to be seen the vortex shedding, beyond the inlet profile and buffer zone positions.

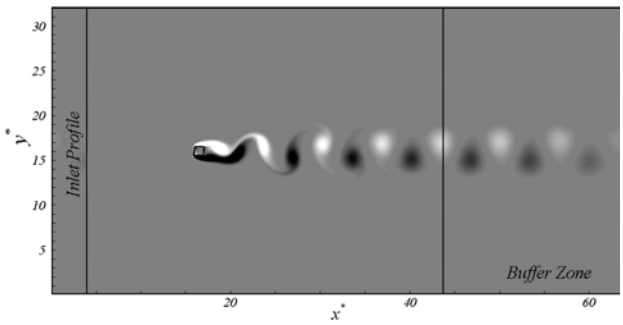

Figure 3: Flow over a square cylinder at *Re=100*, vorticity field $\left(-1 < \omega < 1\right)$.

Figure 4 shows the vorticity field evolution for different time steps. The first time showed at figure 3a, at the beginning of the simulation, arise the two recirculation bubbles, at 3b there is the formation of instability, and in the sequence appears the vortex shedding at 3c.

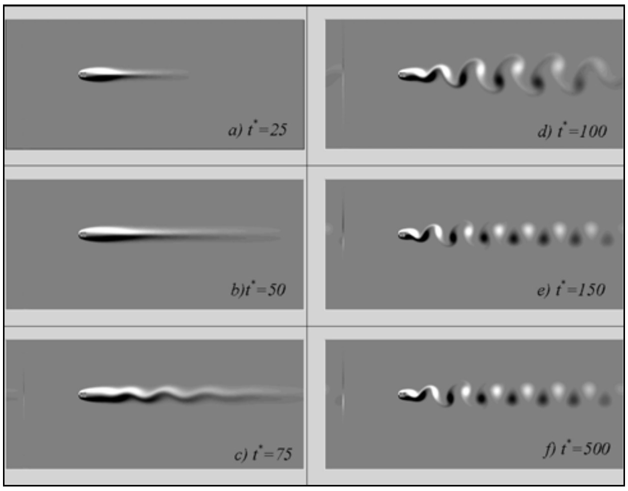

Figure 4: Evolution of vorticity field. Flow over a square cylinder at *Re=100*.

The table 1, demonstrate the comparison with several authors, Lima e Silva (2002) used the same immersed boundary method used in this work (Virtual Physical Model), but discretized by second order finite differences, Fuka and Brechler (2007) used a variant of immersed boundary with direct forcing method and Okajima (1982) is an experimental work. The obtained results of *St* are quite similar, but the *Cd* coefficient is a bit different, due to its large influence of the buffer zone.

Table 1: Comparison of drag coefficient and Strouhal number.

Authors	Lima e Silva (2003)		Fuka and Brechler (2007)		Okajima (1982)		Present Work	
Re	Cd	St	Cd	St	Cd	St	Cd	St
100	1,73	0,14	1,62	0,14	-	0,13	1,58	0,14
150	1,72	0,16	1,63	0,15	-	0,14	1,63	0,15

The other important parameter of comparison is given by L_2 *norm* (equation 21) at the immersed boundary, rigorously, should be zero, but due to filtering process it is injured as can see at figure 5.

$$L_2 = \sqrt{\frac{1}{N_x}\frac{1}{N_y}\sum_{i=1}^{N_x}\sum_{j=1}^{N_y}\left\| u_l^{num}\left(x_l, y_j, t\right) - u_l^{w}\left(x_l, y_j, t\right)\right\|^2} \qquad (21)$$

where u_l^{num} is the numeric velocity at the immersed boundary and u_l^{w} is the velocity at boundary, in this case , $u_l^{w} = 0$, the cylinder is stopped.

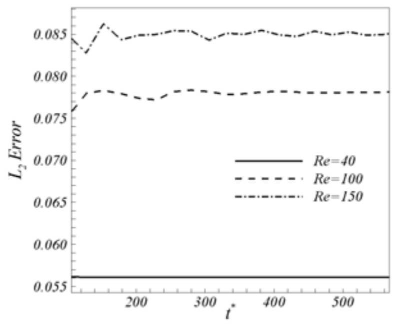

Figure 5: Time evolution of L_2 *norm*.

5. CONCLUSIONS

The motivations of this paper are improving the pseudo-spectral methodology, which is high order method and low computational cost, but restrained to periodic boundary conditions. Searching for this aim a fusion of immersed boundary and the classic Fourier pseudo-spectral method was made. The Fourier pseudo-spectral method allows the solution of the incompressible Navier-Stokes equations with the high order accuracy, in cases where the equations to be solved are periodic and steady the methodology accuracy order is restrained to machine accuracy (Mariano, 2007; Moreira, 2007). Other great vantage is the computational cost in comparison to another high order methods, because of the pressure disentail and the use of FFT algorithm. The connection between the Fourier pseudo-spectral and immersed boundary methodologies allows the simulation of non-periodic-flows.

In the simulations of the flow over a square cylinder it is possible observe the drag coefficient and Sthrouhal number quite similar to other authors, and the vortex shedding are reasonable. The disadvantages are the requirement of using use the buffer zone and the filtering process. In the future, it is expected the solution of problems with no coincidence between lagrangian and eulerian points, allowing to solve the complex and movel geometries.

6. ACKNOWLEDGEMENTS

The authors thank the College of Engineering Mechanical (FEMEC) of the University Federal of the Uberlândia, and Capes to support.

7. REFERENCES

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

Canuto C., Hussaini, M.Y. Quarteroni, A. and Zang, T.A., 1988. “Spectral methods in fluid dynamics”. 2nd ed. Springer-Verlag.

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.

Fuka, V. and Brechler, J., 2007 “High Resolution Flow Modelling for Meteorological Purposes”. WDS'07 Proceedings of Contributed Papers, Part III, PP.150–155. MATFYZPRESS.

Goldstein, D., Adachi, T. and Sakata, H., 1993. “Modeling a no-slip flow with an external force field”. Journal Computational Physics, New York. v. 105, pp. 354.

Kopriva, D.A., 1986. “A spectral multidomain method for the solution of hyperbolic systems”, Applied Numerical Mathematics, Amsterdan. Vol. 2, pp. 221-241.

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, New York. Vol. 189, pp. 351–370.

Mariano, F. P., Moreira, L. Q. and Silveira-Neto, A., 2007. “Fourier Pseudo Spectral-Immersed Boundary Method Aplied Non-Periodic Flows. 19th International Congress of Mechanical Engineering, Brasília, DF, Brazil.

Mariano, F.P., 2007. “Simulação de escoamentos não periódicos utilizando as metodologias pseudo-espectral de Fourier e da fronteira imersa acopladas”. 119 f. Dissertação (Mestrado em Engenharia Mecânica) – Faculdade de Engenharia Mecânica, Universidade Federal de Uberlândia.

Okajima, A., 1982. “Strouhal numbers of rectangular cylinders”. Journal of Fluid Mechanic, v.123, pp.379–398.

Peskin, C.S., 1972. “Flow patterns around heart valves: a numerical method”. Journal of Computational Physics, New York. Vol. 10, pp. 252-271.

Silveira-Neto, A., 2002. “Turbulência nos fluidos aplicada”. Uberlândia: UFU, 2002. Apostila da Disciplina Mecânica dos Fluidos do Programa de Pós-Graduação da Universidade Federal de Uberlândia.

Souza, A.M., 2005. “Análise numérica da transição à turbulência em escoamentos de jatos circulares livres”. 180 f. Tese. (Doutorado em Mecânica) - Faculdade de Engenharia Mecânica, Universidade Federal de Uberlândia.

Unverdi, S. and Tryggvason, G. 1992. “A front-tracking method for viscous, incompressible, multifluid flows”. Journal of Computational Physics, New York. Vol. 100, pp. 25-37.

Uzun, A. 2003. “3D Large-eddy simulation for jet aeroacoustics”. Purdue University. Pardue. Thesis Doctorate.

Zang, T.A., 1987. “Numerical simulation of non-linear interactions in channel and boundary-layer transition”. In: R.W. Miksad, T.R. Akylas, T. Herbert, “Nonlinear wave interactions in fluids”, New York: ASME, pp. 131-145.

Enriquez-Remigio, S.A., 2000. “Introdução de fontes e sumidouros em escoamentos incompressíveis bidimensionais por intermédio do Método da Fronteira Imersa”. Dissertação (Mestrado em Mecânica) - Faculdade de Engenheira Mecânica, Universidade de São Paulo, São Paulo.

8. RESPONSIBILITY NOTICE

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