APPLICATION OF A FOURIER SPECTRAL METHOD TO A NON-PERIODIC FREE SHEAR FLOW

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 Ávilla, 2121, Campus Santa Mônica, Bloco: 5P, Uberlândia - MG
Felipe Pamplona Mariano, fpmariano@mecanica.ufu.br
Universidade Federal de Goiâs - UFG, Escola de Engenharia Elétrica e de Computação - EEEC, Av. Universitária, 1488, Bloco: A, Piso 3, Setor Leste Universitário, Goiânia - GO

Abstract. *The turbulence in fluids is one of the most challenging problems nowadays, especially in industrial applications. For example in mixtures processes, heat transfer, lubrication, combustion, and propulsion systems. Because of such interest, in the present work, the main objective is to show a new methodology to solve and to analyze the transition to turbulence of spatial jets at moderate Reynolds numbers using Large Eddy Simulation (LES) methodology. A computational code, based on Fourier Pseudo-Spectral Method (FPSM), was developed in the Laboratory of Fluids Mechanics (MFLab) of Federal University of Uberlândia, provided an excellent numerical accuracy and low computational cost in comparison with another high order methods, because it uses the Fast Fourier Transform (FFT) and projection method of pressure term, at Fourier space. The main characteristic of the developed methodology is that do not required the Poisson solver. The drawbacks of FPSM are boundary conditions, which should be periodic. In order to simulate non-periodical problems, Immersed Boundary Method (IBM) was used to impose the boundary conditions. The coupling of FPSM with IBM methods results in the IMERSPEC methodology. The present work shows the use of this new methodology in the simulations of the development of spatial jets in transition and/or turbulent state. Consistent results were obtained in comparison of literature. The simulations performed allowed to verify the transition to turbulence as well as the coherent structures that characterises it.*

Keywords: *Computational Fluids Dynamic, Fourier Pseudo-Spectral Method, Large Eddy Simulation, Jet flows.*

1. INTRODUCTION

The jets are free shear flows and their origin is associated to the passage of flow through a nozzle of a diameter D and it is discharged in an environment of same fluid. The interest on jets is due the large number of industrial applications, as mixture of components, heat exchangers, propulsion systems and even though in the fuel injection in the combustion chamber. The agreement of the dynamics of eddy structures that compose this flow and its control is a important tool for optimisation projects involving jets.

The dynamics of a jet is strongly influenced by the presence of vortices, commonly referred to as coherent structures, mainly in the region transition near the injection nozzle. The jet control can be achieved by the manipulation of these coherent structures Hilgers (1999); Angele *et al.* (2005); Tamburello and Amitay (2006). As consequence, it is a capital importance for understanding of dynamics and the topology of these eddy structures. A particular application of jet control is the manipulation of coherent structures to noise reduction, since, pairs of vortices, generated downstream of the nozzle, are an important source of noise.

The first results expressive in jets control were obtained by Crow and Champagne (2006), using loudspeakers to impose various frequencies at the entrance of jet. The jet had maximum amplification of disturbances, corresponding to the Strouhal number of $St = fD/w_1 = 0.3$. The end of the potential core has happened at position $z/d = 4$, where D is the diameter of the jet in the nozzle, w_1 is the maximum axial velocity. Therefore in this frequency correspond to a preferred mode.

Furthermore, several studies have been developed focused on different analysis of jets, *e.g.*, eddy structures, Hori and Sakakibara (2004); Matsuda and Sakakibara (2005), the inlet boundary conditions, Zaman (1985); Xu and Antonia (2002); Stanley and Sarkar (1999), the influence of Reynolds number and the forms of jet control were studied by Hilgers (1999); da Silva and Métais (2002a); Angele *et al.* (2005); Tamburello and Amitay (2006).

Although many studies developed focused on fundamental analysis of turbulent jets, by the experimental analysis of mean flow, numerical simulations still there are points that need further research. In order to study the numerical simulations of spatial developing jets, the present paper shows the centerline profile of the jet, the spreading rate, the virtual origin and the constant decaying of the jet in comparison with experimental results using a full spectral methodology, IMERSPEC, Mariano *et al.* (2010a,b). Furthermore, the transition and turbulent jets is modeling by Smagorinky Dynamic Model.

2. MATHEMATICAL MODELING

In this session the mathematical model of Immersed Boundary Method, based in Multi-Direct Forcing is presented Wang *et al.* (2008). After that, the equations which govern the problem will be transformed to Fourier spectral space and finally, the IMERSPEC methodology is presented.

2.1 Mathematic model for the fluid

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

$$\frac{\partial u_i}{\partial t} = -\frac{\partial}{\partial\, x_j}\left(u_i u_j\right) - \frac{\partial\, p}{\partial\, x_i} + \frac{\partial}{\partial\, x_j}\left[\upsilon_{ef}\left(\frac{\partial\, u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + f_i \tag{1}$$

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

where $\frac{\partial p}{\partial x_i} = \frac{1}{\rho}\frac{\partial p*}{\partial x_l}$; $p*$ is the static pressure in $[N/m^2]$; u_i is the velocity in i direction in $[m/s]$; $f_i = \frac{f_i^*}{\rho}$; f_i* is the term source in $[N/m^3]$; ρ is the density in $[kg/m^3]$; υ_{ef} is the effective viscosity in $[m^2/s]$, which the dynamic model is used by evaluated it; x_i 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_{PeD} (Fig. 1), but has values different from zeros only in the points in which there is coincide with the immersed geometry, *i.e.*, Γ_{PeD} and Γ_i (Fig. 1), $i = 1, 2, ..., N$, where N is the number of bluff bodies or boundary used in Γ_{PhD}. It enable the eulerian field perceives the presence of a solid interface (Enriquez-Remigio and Silveira-Neto, 2007).

$$f_i\left(\vec{x}, t\right) = \begin{cases} F_i\left(\vec{X}, t\right) & \text{if} \quad \vec{x} = \vec{X} \\ 0 & \text{if} \quad \vec{x} \neq \vec{X} \end{cases}, \tag{3}$$

where $\vec{x}$ is the position of a fluid particle and $\vec{X}$ is the position of a fluid particle that is placed besides of the solid interface.

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

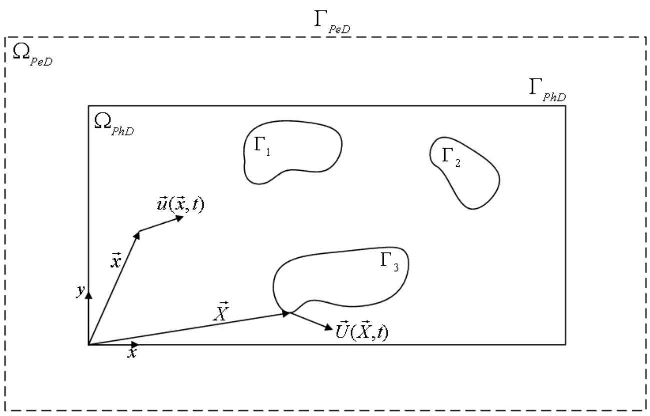

Figure 1. Schematically representation of Eulerian and Lagrangian domain.

Eq. 3 shows that the field $f_i(\vec{x}, t)$ is discontinuous, which can be numerically solved only when there is coincidence between the points that compose the interface domain with some collocation points that compose the fluid domain. In the cases that there is no coincidence between these points, very frequently in the complex geometries, it is necessary to distribute the function on its neighbourhoods. Just by calculating the Lagrangian force field, $F_i(\vec{X}, t)$, 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 we don't need to use ad-hoc constants. It allows the modeling of no-slip condition on immersed interface. The Lagrangian force $F_i\left(\vec{X},t\right)$ is available by momentum equation over a fluid particle that is joined in the fluid-solid interface:

$$F_i\left(\vec{X},t\right)=\frac{\partial U_i}{\partial t}\left(\vec{X},t\right)+\frac{\partial}{\partial X_j}(U_iU_j)\left(\vec{X},t\right)+\frac{\partial P}{\partial X_i}\left(\vec{X},t\right)-\frac{\partial}{\partial X_j}\left[\upsilon_{ef}\left(\frac{\partial U_i}{\partial X_j}+\frac{\partial U_j}{\partial X_i}\right)\right]. \tag{4}$$

The capital symbols are the same as Eq. 1, but they are calculated only over the Lagrangian interface Γ shown in Fig. 1.

Using the temporal parameter, U^*, proposed by Wang *et al.* (2008) and discretizing the time operator, we have:

$$F_i\left(\vec{X},t\right)=\frac{U_i\left(\vec{X},t+\Delta t\right)-U_i^*\left(\vec{X},t\right)+U_i^*\left(\vec{X},t\right)-U_i\left(\vec{X},t\right)}{\Delta t}+RHS_i\left(\vec{X},t\right), \tag{5}$$

where Δt is the time step and

$$RHS_i\left(\vec{X},t\right)=\frac{\partial}{\partial X_j}(U_iU_j)\left(\vec{X},t\right)+\frac{\partial P}{\partial X_i}\left(\vec{X},t\right)-\frac{\partial}{\partial X_j}\left[\upsilon_{ef}\left(\frac{\partial U_i}{\partial X_j}+\frac{\partial \bar{U}_j}{\partial X_i}\right)\right]. \tag{6}$$

The Eq. 5 is solved by decomposition, giving Eqs. 7 and 8:

$$\frac{U_i^*\left(\vec{X},t\right)-U_i\left(\vec{X},t\right)}{\Delta t}+RHS_i\left(\vec{X},t\right)=0, \tag{7}$$

$$F_i\left(\vec{X},t\right)=\frac{U\left(\vec{X},t+\Delta t\right)-U_i^*\left(\vec{X},t\right)}{\Delta t}, \tag{8}$$

where $U\left(\vec{X},t+\Delta t\right)=U_{FI}$ is the immersed boundary velocity and $U_i^*\left(\vec{X},t\right)$ is given by:

$$U_i\left(\vec{X},t\right)=\begin{cases} u_i^*\left(\vec{X},t\right) & \text{if} \quad \vec{x}=\vec{X} \\ 0 & \text{if} \quad \vec{x}\neq\vec{X}\end{cases}. \tag{9}$$

Equation 7 is solved at the eulerian domain at Fourier spectral space, *i.e.*, it is replaced by solution of the transformed Eq. 1 with $f_i=0$. $u_i^*\left(\vec{x},t\right)$ is interpolated for Lagrangian domain, giving $U_i^*\left(\vec{X},t\right)$ and it is computed on Eq. 8. Then it is smeared to Eulerian collocation points. Finally, the Eulerian velocities are updated by Eq. 10:

$$u_i\left(\vec{x},t+\Delta t\right)=u_i^*\left(\vec{x},t\right)+\Delta t f_i. \tag{10}$$

2.3 Mathematic model for the fluid in Fourier spectral space

Given the equations that model the flow through immersed boundary method, the next step is to transform them to the Fourier spectral space. For instance, Fourier transform of continuity Eq. 2, gives:

$$\imath k_j\widehat{u}_j=0, \tag{11}$$

where "$\widehat{\ }$" means that variable is in Fourier spectral space. The Fourier transformation is performed using the FFT algorithm implemented by Takahashi (2006).

Eq. 11 show that the wave number vector k_i is orthogonal to transformed velocity, $\widehat{u}_i(\vec{k},t)$. We can define the plane of divergence free, named plane π. It is perpendicular to wave number vector and thus, transformed velocity belongs to the plane π. By applying the Fourier transform in the momentum equation Eq. 1:

$$\widehat{\frac{\partial u_i^*}{\partial t}}=-\frac{\partial}{\partial x_j}\widehat{\left(u_i^*u_j^*\right)}-\widehat{\frac{\partial p}{\partial x_i}}+\frac{\partial}{\partial x_j}\left[\upsilon_{ef}\left(\widehat{\frac{\partial u_i^*}{\partial x_j}+\frac{\partial u_j^*}{\partial x_i}}\right)\right], \tag{12}$$

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

By definition of the plane π, each of the terms of Eq. 12 assume a position related to it: the transient term $\frac{\partial\widehat{u}_i^*}{\partial t}$ and the viscous term $\nu k^2\widehat{u}_i^*$ belong to the plane π. The gradient pressure term is perpendicular to the plane π. The direction of non-linear term $\iota k_j \widehat{u_i^* u_j^*}$, a priori, is not known, when compared with the plane π. By joining the terms of Eq. 12, we found that:

$$\underbrace{\left[\frac{\partial \widehat{u}_i}{\partial t}\right]}_{\in \pi} + \underbrace{\left[\frac{\partial \widehat{(u_i u_j)}}{\partial x_j} + \iota k_i \widehat{p} - \frac{\partial}{\partial\, x_j}\left[\upsilon_{ef}\left(\widehat{\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}}\right)\right]\right]}_{\in \pi} = 0. \quad \Rightarrow \tag{13}$$

The Eq. 13 imply that:

$$\left[\frac{\partial \widehat{(u_i u_j)}}{\partial x_j} + \iota k_i \widehat{p} - \frac{\partial}{\partial\, x_j}\left[\upsilon_{ef}\left(\widehat{\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}}\right)\right]\right] = \wp_{im}\left[\frac{\partial \widehat{(u_m u_j)}}{\partial x_j} - \frac{\partial}{\partial\, x_j}\left[\upsilon_{ef}\left(\widehat{\frac{\partial u_m}{\partial x_j} + \frac{\partial u_j}{\partial x_m}}\right)\right]\right]. \tag{14}$$

where $\wp_{im}$ is the projection tensor (Canuto *et al.*, 2006).

The gradient pressure field is orthogonal to the plane π. So, the pressure and velocities fields at Fourier space are not coupled anymore. Nevertheless the pressure field can be recovered as a pos-processing procedure, as shown by (Mariano *et al.*, 2010a).

Other important point is the non-linear term, from which appears a 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 integral is very expensive to be solved. So, this is solved by pseudo-spectral Fourier method, Canuto *et al.* (2007). Therefore the momentum equation in the Fourier space, using the projection method, assumes the following form:

$$\frac{\partial \widehat{u}_i\left(\vec{k},t\right)}{\partial t} = -\iota k_j \wp_{im} \int\limits_{\vec{k}=\vec{r}+\vec{s}} \widehat{u_m}\left(\vec{r}\right)\widehat{u}_j\left(\vec{k}-\vec{r}\right) d\vec{r} + \iota k_j \wp_{im} \int\limits_{\vec{k}=\vec{r}+\vec{s}} \widehat{\upsilon_{ef}}\left(\vec{r}\right)\widehat{\left(\frac{\partial u_m}{\partial x_j} + \frac{\partial u_j}{\partial x_m}\right)}\left(\vec{k}-\vec{r}\right) d\vec{r}. \tag{15}$$

The non-linear term can be handed by different forms: advective, divergent, skew-symmetric or rotational (Canuto *et al.*, 2007), in spite of being the same mathematically, they present different properties when discretized. The skew-symmetric form is more stable and present best results. Therefore, this procedure is used in the present work. The non-linear term is solved using the pseudo-spectral method (Canuto *et al.*, 2007). The velocity product is calculated at physical space and transformed to the spectral space.

2.4 Proposed Methodology: IMERSPEC

The proposed algorithm is described as:

1) Solve Eq. 14 in the Fourier spectral space and obtain the temporal parameter $\widehat{u^*}_i(\vec{k},t)$, using the low dispersion and low storage Runge-Kutta method, proposed by Allampalli *et al.* (2009);

2) Use the Inverse Fast Fourier Transformer in $\widehat{u^*}_i(\vec{k},t)$ and obtain $u_i^*(\vec{x},t)$ at physic space;

3) Interpolate $u_i^*(\vec{x},t)$ to the Lagrangian domain using Eq. 9;

4) Calculate the Lagrangian force, $F_i^*(\vec{X},t)$, by Eq. 8;

5) Calculate the Eulerian force, $f_i^*(\vec{x},t)$ using Eq. 9;

6) Update the Eulerian velocity, $u_i\left(\vec{x},t+\Delta t\right)$ by Eq. 10 and transform it, using FFT, to the spectral space, obtaining $\widehat{u}_i\left(\vec{x},t+\Delta t\right)$, and return to step 1.

3. Results

All simulations were performed in a domain $Lx = 8d$, $Ly = 8d$ and $Lz = 32d$, divided with 128x128x512 collocation points in the spanwise direction (x e y) and streamwise direction (z) respectively. A buffer zone and a forcing zone were used. The domain of interest is the total domain less the buffer zone and forcing zone. So the domain of interest has $Lx = 8d$, $Ly = 8d$ e $Lz = 24d$.

The velocity profile input on the domain interest entry is the same for all simulations, and is given by:

$$w\left(r,\theta,z,t=0\right) = \frac{w_1+w_2}{2} - \frac{w_1-w_2}{2}\tanh\left(\frac{1}{4}\frac{R}{\theta}\left(\frac{r}{R}-\frac{R}{r}\right)\right), \tag{16}$$

where w_1 is the input velocity of the jet, w_2 is the velocity of "co-flow", r is the radius of the jet and θ is the momentum thickness. The rate R/θ defines the slope velocity profile, and has strong influence on the process transition of turbulence, and in general, in that the rate increases, the instability of the jet also increases (Michalke and Hermann, 2006). In the present work rate was used to $R/\theta = 20$, $w_1 = 1,025\,[m/s]$ and $w_2 = 0,025\,[m/s]$. The Reynolds number is given by Eq. 17, in all simulations the Reynolds number used equals 1050, where:

$$Re_d = \frac{\left(w_1-w_2\right)d}{\upsilon}. \tag{17}$$

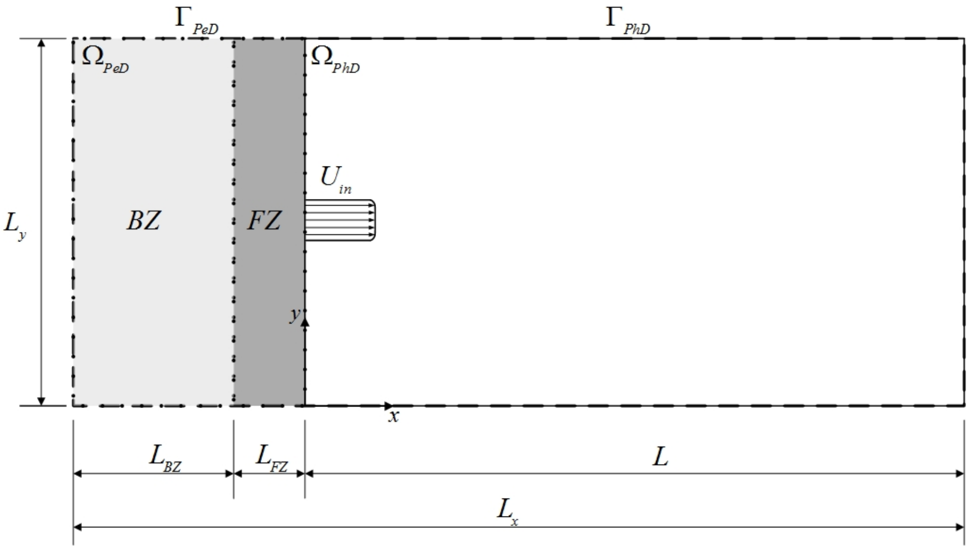

Figure 2. Domain for the simulations.

It is important to note that the entry velocity profile is imposed by Immersed Boundary Method, because the Fourier pseudo-spectral method is evaluated only using the periodical boundary conditions. Then, we extend the domain using the buffer zone and forcing zone, Mariano *et al.* (2010c), as shown in Fig. 2.

The average velocity for a jet the axis of symmetry, in the region of "self preserving", must obey following relationship:

$$\frac{w_1 - w_2}{\Delta\overline{w}_c} = \frac{1}{b_u d}\left(z - z_0\right), \tag{18}$$

where, $\Delta\overline{w}_c = \overline{w}\left(0,0,z\right) - w_2$ is the velocity over on the average velocity on the centerline of the jet $(\overline{w}\left(0,0,z\right))$ in relation co-flow velocity w_2, at a distance z nozzle exit jet, z_0 is the virtual origin of the jet and b_u is the constant rate of decay.

To define the virtual origin, it is part of the line given by Eq. 19, which is adjusted to final potential core of the jet. The virtual origin is obtained by extrapolation of the line Eq. 19 until $\frac{w_1-w_2}{\Delta\overline{w}_c}$ reach the value of 0, where a and b are the coefficients of the line,

$$\frac{w_1 - w_2}{\Delta\overline{w}_c} = az + b. \tag{19}$$

The half width of jet is defined as the radial distance between the centerline of the jet $(r = 0)$ to the position where the average velocity is equal to half the average velocity on centerline of the jet and is given by:

$$\overline{w}\left(\delta_{1/2}\left(z\right), 0, z\right) = \frac{1}{2}\left(\overline{w}\left(0,0,z\right) - w_2\right). \tag{20}$$

And the rate of spreading of the jet is given by:

$$s = \frac{\delta_{1/2}}{z - z_0}. \tag{21}$$

In all simulations is added to the velocity profile, at the inlet boundary condition, a fluctuating velocity to model the residual turbulence and induce the transition to turbulence. Were tested two types of fluctuations, based on a digital generation noise and another using white noise.

The generation of digital noise was used the technique proposed by Smirnov *et al.* (2001). This method is capable to generate velocity fluctuations in three dimensions with length scales and time scales specific. The length scale used is $d/5$ (Wang *et al.*, 2010) on direction z and $d/10$ in directions x e y and the time scale was used $2dw_1/5$. The Fig. 3 illustrates the mean dimensionless velocity for the profile component w on the centerline of the jet. The axial velocity component is a dimensionless by $w* = \frac{(\overline{w}(0,0,z)-w_2)}{w_1-w_2}$.

Three mean velocity profiles are showed, where the velocity fluctuations are generated by the method of Smirnov *et al.* (2001) and one for white noise. Note that when a high turbulent intensities is injected the transition to turbulence happens in advanced. On the contrary, when injected low intensity turbulent the transition to turbulence happens later. Using white noise, even with a turbulence intensity considerable, the jet remained laminar. Similar result was also found in Stanley and Sarkar (1999).

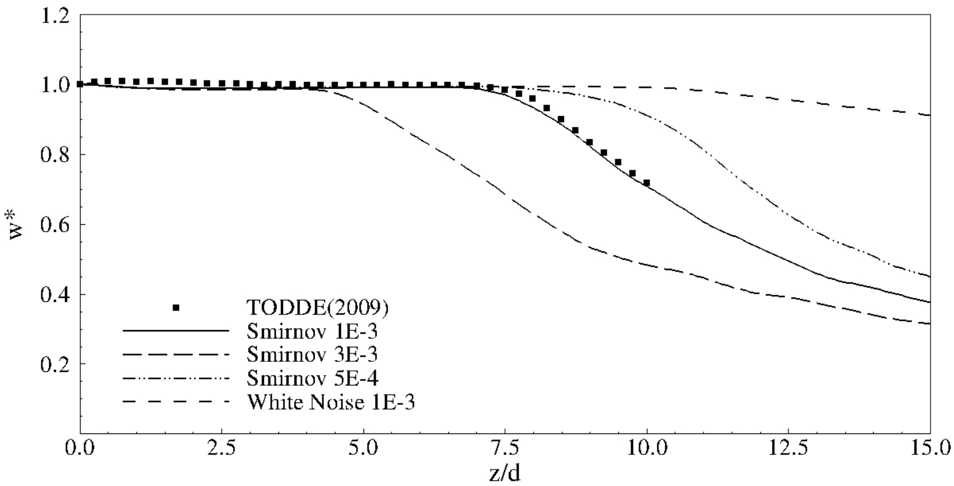

Figure 3. Average axial velocity profile jet.

The isosurfaces the of the criterion $Q = 0.3$ are shown in Fig. 4. Observe that the region of interest are noticed a pairing of vortices, also presented by da Silva and Métais (2002b), In sequence, the interaction of longitudinal vortices with Kelvin-Helmholtz and finally the structures are already in chaotic state degenerated to turbulence.

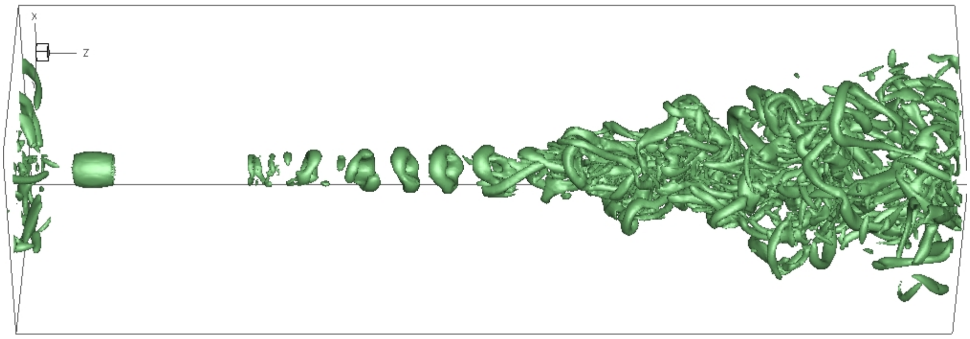

Figure 4. Criterion $Q = 0.3$ for spatial jet.

The Tab. 1 shows the comparison for the virtual origin (z_0) Eq. 19, constant spreading (b_u) Eq. 18 and spreading rate (s) Eq. 21. A good agreement of the present work results compared with the experimental works of Todde *et al.* (2009); O'Neill *et al.* (2004), showing the great accuracy of IMERSPEC methodology.

Table 1. Comparison of decay constant, spreading rate and virtual origin of jet.

Works	z_0	b_u	s
Todde *et al.* (2009)	$4,17$	$4,13$	–
O'Neill *et al.* (2004)	–	$5,60$	$0,1$
Present work	$4,05$	$4,13$	$0,1$

4. CONCLUSIONS

The IMERSEPEC methodology (Mariano *et al.* (2010b)) was presented and applied to the jet circular in three dimensional flows. The IMERSEPEC shows high sensitivity inlet boundary condition, as possible, reproduce numerically with fidelity the experiments. The results obtained are very close to experimental data for $Re_d = 1050$. .

It is important to note that IMERSPEC methodology was developed to incompressible flows and provide good features for Navier-Stokes solutions, as well as high accuracy and high convergence rates. Furthermore, CPU time should be less than high order methodologies in physical space. This is because the pressure linear solver for Poisson equation is replaced by product of vector-matrix, providing by Fourier pseudo-spectral method.

5. ACKNOWLEDGEMENTS

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

6. REFERENCES

Allampalli, V., Hixon, R., Nallasamy, M. and Sawyer, S.D., 2009. "High-accuracy large-step explicit Runge-Kutta (HALE-RK) schemes for computational aeroacoustics". *Journal of Computational Physics*, Vol. 228, No. 10, pp. 3837–3850. ISSN 00219991. doi:10.1016/j.jcp.2009.02.015.

Angele, K., Kurimoto, N., Suzuki, Y. and Kasagi, N., 2005. "Three-dimensional vortical structures in a coaxial jet manipulated with micro-actuators". In *6th World Conference on Experimental Heat Transfer, Fluid Mechanics, and Thermodynamics, Matsushima, Miyagi, Japan*.

Canuto, C., Hussaini, M., Quarteroni, A. and Tomas A. Zang, J., 2007. *Spectral Methods : Evolution to Complex Geometries and Applications to Fluid Dynamics*. Springer BerlinHeidelbergNew York, New York, 1st edition. ISBN 1434-8322. doi:2007924823.

Canuto, C., Hussaini, M., Quarteroni, A. and Zang, T.A., 2006. *Spectral Methods Fundamentals in Single Domains*. Springer-Verlag Berlin Heidelberg, New York, 1st edition. ISBN 13 978-3-540-30725-9.

Crow, S.C. and Champagne, F.H., 2006. "Orderly structure in jet turbulence". *Journal of Fluid Mechanics*, Vol. 48, No. 03, pp. 547–591. ISSN 00221120. doi:10.1017/S0022112071001745.

da Silva, C.B. and Métais, O., 2002a. "On the influence of coherent structures upon interscale interactions in turbulent plane jets". *Journal of Fluid Mechanics*, Vol. 473, pp. 103–145. ISSN 0022-1120. doi:10.1017/S0022112002002458.

da Silva, C.B. and Métais, O., 2002b. "Vortex control of bifurcating jets : A numerical study". *Physics of Fluids*, Vol. 14, No. 11, pp. 3798–3819. doi:10.1063/1.1506922.

Enriquez-Remigio, S.A. and Silveira-Neto, A., 2007. "A New Modeling Of Fluid-Structure Interaction Problems Through Immersed Boundary Method/Virtual Psysical Model (Ibm/Vpm)." In *Proceedings of the Cobem*.

Goldstein, D., Handler, R. and Sirovich, L., 1993. "Modeling a No-Slip Flow Boundary with an External Force Field". *Journal of Computational Physics*, Vol. 105, No. 2, pp. 354–366. ISSN 00219991. doi:10.1006/jcph.1993.1081.

Griffith, B. and Peskin, C., 2005. "On the order of accuracy of the immersed boundary method: Higher order convergence rates for sufficiently smooth problems". *Journal of Computational Physics*, Vol. 208, No. 1, pp. 75–105. ISSN 00219991. doi:10.1016/j.jcp.2005.02.011.

Hilgers, A., 1999. "Parameter optimization in jet flow control". *Optimization*.

Hori, T. and Sakakibara, J., 2004. "High-speed scanning stereoscopic PIV for 3D vorticity measurement in liquids". *Measurement Science and Technology*, Vol. 15, pp. 1067–1078. doi:10.1088/0957-0233/15/6/005.

Mariano, F.P., Leonardo de Queiroz Moreira, Aristeu da Silveira-Neto, da Silva, C.B. and Pereira, C.F., 2010a. "A new incompressible Navier-Stokes solver combining Fourier pseudo-spectral and immersed boundary methods". *Computer Modeling in Engineering & Sciences*, Vol. 59, No. 2, pp. 181–216.

Mariano, F.P., Moreira, L.d.Q. and Silveira-Neto, A., 2010b. "Mathematical Modeling of Non-Periodic Flows Using Fourier Pseudo-Spectral and Immersed Boundary Methods". In J.C.F. Pereira and A. Sequeira, eds., *V European Conference on Computational Fluid Dynamics*. Lisbon, June, pp. 14–17.

Mariano, F.P., Moreira, L.D.Q. and Silveira-Neto, A., 2010c. "Simulação de Escoamentos Sobre Degrau Usando os Método Pseudo-Espectral de Fourier Acoplado com o Método da Fronteira Imersa". In *EPTT 2010 - VII Escola de Primavera de Transição e Turbulência*. Ilha Solteira-SP, 1.

Matsuda, T. and Sakakibara, J., 2005. "On the vortical structure in a round jet". *Physics of Fluids*, Vol. 17, pp. 1–11. doi:10.1063/1.1840869.

Michalke, A. and Hermann, G., 2006. "On the inviscid instability of a circular jet with external flow".

O'Neill, P., Soria, J. and Honnery, D., 2004. "The stability of low Reynolds number round jets". *Experiments in Fluids*, Vol. 36, No. 3, pp. 473–483. ISSN 0723-4864. doi:10.1007/s00348-003-0751-5.

Smirnov, A., Shi, S. and Celik, I., 2001. "Random flow generation technique for large eddy simulations and particle-dynamics modeling". *Journal of Fluids Engineering*, Vol. 123, p. 359. doi:10.1115/1.1369598.

Stanley, S.A. and Sarkar, S., 1999. "Influence of Nozzle Conditions and Discrete Forcing on the Developing Region of Turbulent Planar Jets". *AIAA Journal*, Vol. 38, No. LBNL-43150, pp. 1–40.

Takahashi, D., 2006. "A Hybrid MPI / OpenMP Implementation of a Parallel 3-D FFT on SMP Clusters". pp. 970–977.

Tamburello, D. and Amitay, M., 2006. "Manipulation of an axisymmetric jet using continuous control jets". *Journal of Turbulence*, Vol. 7, No. 59, pp. 1–24. doi:10.1080/14685240600854892.

Todde, V., Spazzini, P.G. and Sandberg, M., 2009. "Experimental analysis of low-Reynolds number free jets Evolution along the jet centerline and Reynolds number effects". *Experiments in Fluids*, Vol. 3, No. 47, pp. 279–294. doi: 10.1007/s00348-009-0655-0.

Uhlmann, M., 2005. "An immersed boundary method with direct forcing for the simulation of particulate flows". *Journal* *of Computational Physics*, Vol. 209, No. 2, pp. 448–476. ISSN 00219991. doi:10.1016/j.jcp.2005.03.017.

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. doi:10.1016/j.ijmultiphaseflow.2007.10.004.

Wang, Z., He, P., Lv, Y., Zhou, J., Fan, J. and Cen, K., 2010. "Direct Numerical Simulation of Subsonic Round Turbulent Jet". *Flow, Turbulence and Combustion*, Vol. 84, No. 4, pp. 669–686. ISSN 1386-6184. doi:10.1007/s10494-010-9248-5.

Xu, G. and Antonia, R.A., 2002. "Effect of different initial conditions on a turbulent round free jet". doi:10.1007/s00348-002-0523-7.

Zaman, K.B.M.Q., 1985. "Effect of initial condition on subsonic jet noise".

7. Responsibility notice

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