ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil


# TURBULENT INLET CONDITIONS MODELING IN LARGE-EDDY SIMULATIONS OF FLOWS OCCURRING ON A BACKWARD-FACING STEP

**Marcelo Maia Ribeiro Damasceno**
**João Marcelo Vedovoto**
**Aristeu da Silveira Neto**
Universidade Federal de Uberlândia
Avenida João Naves de Ávila, 2121
Bloco 5P - Laboratório de Mecânica dos Fluidos
Uberlândia, MG
marcelomrd@gmail.com
jmvedovoto@mecanica.ufu.br
aristeus@mecanica.ufu.br

***Abstract.*** *Turbulence is a phenomenon which presents peculiarities when it is experimented or simulated. This occurs due to its complexity and high sensibility to the inlet conditions of the turbulent flow fields and the large range of time and length scales, characteristics that demand a large quantity of computational resources. A simplification for this situation is obtained with the use of approximations and turbulence models, which reduce the flow description level. The Large-Eddy Simulations methodology were applied aiming the modeling of the previously mentioned complexity. This method consists in the application of a filter in the transport equations to resolve the scales larger than this filter. In the present work, the remaining scales were determined by classical and dynamic Smagorinsky models and three different approximations for the characterization of the inlet conditions were applied: the superimposition of white noise on a mean velocity profile, Random Flow Generation (RFG) and Synthetic Eddy Method (SEM). It was possible to realize that the use of the dynamic Smagorinky model and the RFG or SEM methodologies resulted in a better characterization of the studied flow. The performance differences between these two inlet conditions generators were also assessed in the present work.*

***Keywords:*** *large-eddy simulations, turbulent inlet conditions*

## 1. INTRODUCTION

The turbulence phenomenon is a flow regime characterized by presenting peculiarities on its experimentation or simulation. This is due to its complexity and, also, sensibility to the turbulent flows inlet conditions. It is defined by a wide range of time and length scales, which complicates a detailed description of such flows.

One alternative for this complexity is based on the decrease of the flow description. This proposition can be obtained using approaches and turbulence models, which vary in complexity, accuracy, computational cost, etc. The use of more realistic turbulent inlet conditions is also an important factor to be evaluated due to the turbulence sensibility to this conditions.

In this context, the Computational Fluid Dynamics appears as an important tool for studying practical engineering situations. This methodology may generate faster results, in addition to lower costs related to development or evaluation of projects, when compared to experimental procedures.

The application of the Navier-Stokes equations for modeling laminar or turbulent flows allows the characterization of these phenomena in a detailed and accurate way. This characteristic generates difficulties when situations in which turbulence is observed, because these equations describe all the velocity and pressure fields for all time and length scales. It is a wide amount of information contained in these fields and, as a consequence, the direct resolution of this system of equations for practical situations becomes impossible. In this context, three main resolution methodologies can be applied: Direct Numerical Simulations (DNS), Large Eddy Simulations (LES) and Reynolds Averaged Navier-Stokes (RANS).

In the DNS methodology, all the turbulence scales are calculated using the Navier-Stokes equations without the imposition of any turbulence model. As a consequence, a mesh refinement capable of picking up all the frequency spectrum, from the largest until the smallest or the Kolmogorov scales, is required. Due to the large quantity of existing scales in engineering situations, this methodology is hard to be applied, but it is very important for describing low Reynolds flows in fluid mechanics. The other methodologies appeared from the difficulty of using this methodology for high Reynolds flows. In this context, the turbulence scales decomposition

ISSN 2176-5480



was proposed, using temporal averages or spatial filtrations.

The application of temporal averages results in a decomposition of the velocity into mean and floating parts. The application of this methodology, known as RANS, requires a complete modeling of the energy spectrum and, for this reason, models become necessary in order to calculate the additional tensor, which was generated from the advective term of the Navier-Stokes equations, after the turbulence scales decomposition.

The use of spatial and temporal filters, in the other hand, produces the filtered Navier-Stokes equations, which are related to the Large-Eddy Simulation (LES) methodology. The applied filter, that is associated to the discretization mesh, has the role of separating the flow scales. This artifice allows the modeling of structures smaller than the mesh used and the calculation of the remaining ones.

The RANS methodology requires a lower refinement when compared to the approximations aforementioned. For this reason, it is applicable in high Reynolds flows. However, a significant amount of informations is not captured because all the energy spectrum is modeled. In other words, the choice between these methodologies must be performed by the researcher, relying on what kind of analysis is intended to be done.

## 2. MATHEMATICAL MODELING

In the present work, the methodology based on the filtered Navier-Stokes equations was retained. This methodology separates the turbulent kinetic energy spectrum in two regions: the first, located above the applied filter, which will be calculated and the second, positioned below it, corresponding to the sub-grid scales. These scales were evaluated using turbulence models, which will exercise the role of transferring energy between the resolved and the unresolved scales that compose the flow spectrum. The large scales, responsible for the flow global characterization and the transport of most of the energy, are directly calculated, whilst the smallest structures are modeled.

A spatial filtering in the Navier-Stokes equations, proposed by Pope (2000), generates the following equation:

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial}{\partial x_j}(\overline{u_i}\;\overline{u_j}) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \tau_{ij}\right], \tag{1}$$

where the global tensor $\tau_{ij}$, defined by Germano *et al.* (1991), is given by:

$$\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j. \tag{2}$$

This additional tensor can be modeled using the Boussinesq's hypothesis, which proposes the calculation of the sub-grid Reynolds' tensor $\tau_{ij}$ as being proportional to the strain rate generated by the filtered velocity field and the turbulent kinetic energy. In other words:

$$\tau_{ij} = -\nu_t\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_i}{\partial x_i}\right) + \frac{2}{3}k\delta_{ij}, \tag{3}$$

where the turbulent viscosity $\nu_t$ is determined from turbulence models and the turbulent kinetic energy $k$ is incorporated to the pressure gradient term.

Most sub-grid models are based on turbulent viscosity concept. Among them, the classical and dynamic Smagorinsky's models are the most popular. For this reason, both models were applied in the present work.

The classical sub-grid scale model, proposed by Smagorinsky (1963), for the determination of the turbulent viscosity $\nu_t$ is presented by Eq. (4), in which $C_s$ represents the Smagorinsky's constant and $\Delta$ is the length-scale related to the filter (mesh spacing):

$$\nu_t = (C_S\Delta)^2\sqrt{2\overline{S}_{ij}\overline{S}_{ij}}. \tag{4}$$

The constant $C_S$ must be adjusted, for each kind of flow, with numerical values normally between 0.05 and 0.30. For situations in which homogeneous and isotropic turbulence are observed, Lilly (1967) proposed the use of $C_S = 0.18$ as an analytical value.

An important issue related to the application of this turbulence model is its lack of capacity of performing accurate calculations of the turbulent viscosity in parietal regions. For this reason, the appliance of a damping function becomes necessary. In the present work, a damping function proposed by van Driest (1956), was applied:

$$C_{SA} = C_S(1 - e^{-d^*/A^+})^2, \tag{5}$$

ISSN 2176-5480



where $d^* = du_\tau/\nu$ denotes the distance to the wall, $u_\tau = \sqrt{\tau_w/\rho}$ is related to the shear velocity, $\tau_w$ corresponds to the shear stress close to the wall, $A^+ = 25$ is a constant determined by Ferziger and Perić (2002) and $C_S$ is the Smagorinsky's constant, previously mentioned.

The dynamic Smagorinsky model, proposed by Germano *et al.* (1991), is based on a function capable of adjusting itself to the flow in time and space and in the application of two filters with different characteristic lengths. Thereby, this methodology is oriented by the informations of the energy levels contained in the smallest resolved scales, located between both filters, for modeling the energy transfer between the resolved and modeled scales. Both filtered Navier-Stokes equations are shown in Eq. (6) and Eq. (7):

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\overline{u}_i \overline{u}_j) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) - \tau_{ij}\right], \tag{6}$$

$$\frac{\partial \hat{\overline{u}}_i}{\partial t} + \frac{\partial}{\partial x_j}(\hat{\overline{u}}_i \hat{\overline{u}}_j) = -\frac{1}{\rho}\frac{\partial \hat{\overline{p}}}{\partial x} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \hat{\overline{u}}_i}{\partial x_j} + \frac{\partial \hat{\overline{u}}_j}{\partial x_i}\right) - T_{ij}\right]. \tag{7}$$

where $\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j$ denotes the Germano's global tensor and $T_{ij} = \hat{\overline{u_i u_j}} - \hat{\overline{u}}_i \hat{\overline{u}}_j$ corresponds to the subtest tensor.

The determination of the function responsible for the generation of the dynamic coefficient of this proposition is obtained from the use of the Germano's identity, $L_{ij} = \widehat{\overline{u}_i \overline{u}_j} - \hat{\overline{u}}_i \hat{\overline{u}}_j = T_{ij} - \hat{\tau}_{ij}$, and $M_{ij} = \hat{\overline{\Delta}}^2 |\hat{\overline{S}}_{ij}| \hat{\overline{S}}_{ij} - \hat{a}$, where $\hat{a} = \overline{\Delta}^2 |\overline{S}_{ij}| \overline{S}_{ij}$. The function previously mentioned is presented as follows:

$$c(\vec{x}, t) = \frac{1}{2}\frac{L_{ij} M_{ij}}{M_{ij} M_{ij}}. \tag{8}$$

The capability of this function to adjust itself to the flow in time and space is an important improvement. The application of more realistic inlet boundary conditions is also important in order to achieve better results. In this context, two methodologies were studied in the present work: the Random Flow Generation (RFG), proposed by Smirnov *et al.* (2001) and the Synthetic Eddy Method (SEM), presented by Jarrin *et al.* (2006).

### *Random Flow Generation (Smirnov* et al.*, 2001)*

The RFG is a modified version of the technique presented by Kraichnan (1970). It can be defined as orthogonal and scaling transformations applied to a continuous flow field, generated by a superposition of harmonic functions.

This methodology requires an anisotropic velocity correlation tensor

$$r_{ij} = \overline{\tilde{u}_i \tilde{u}_j} \tag{9}$$

and the determination of an orthogonal transformation tensor, $a_{ij}$, to diagonalize $r_{ij}$:

$$a_{mi} a_{nj} r_{ij} = \delta_{mn} c_{(n)}^2 \tag{10}$$

$$a_{ik} a_{kj} = \delta_{ij} \tag{11}$$

Thereby, $a_{ij}$ and $c_n$ becomes known functions in space. The variable $c_n$ represents the velocity fluctuations in the new coordinate system, produced by the transformation tensor $a_{ij}$. Hereafter, a transient flow field is generated in a three dimensional domain using the Kraichnan's modified method:

$$v_i(\vec{x}, t) = \sqrt{\frac{2}{N}} \sum_{n=1}^{N} [p_i^n cos(\tilde{k}_j^n \tilde{x}_j + \omega_n \tilde{t}) + q_i^n sen(\tilde{k}_j^n \tilde{x}_j + \omega_n \tilde{t})]. \tag{12}$$

Finally, orthogonal and scaling transformations are applied to the previously generated field $v_i$, in order to obtain a new field $u_i$:

$$\begin{aligned} w_i &= c_{(i)} v_{(i)}, \\ u_i &= a_{ik} w_k. \end{aligned} \tag{13}$$

This procedure results in a transient field $u_i(x_j, t)$ with correlation functions $\overline{u_i u_j}$ equivalent to $r_{ij}$. It is a divergence-free field for any situation involving homogeneous turbulence and, for non-homogeneous turbulence cases, it presents high convergence orders.

ISSN 2176-5480



### *Synthetic Eddy Method (Jarrin* et al.*, 2006)*

The SEM is based on the characterization of turbulence as a superposition of coherent structures. Thereby, these eddies should be generated at the domain inlet plane of the studied situation and defined by a function responsible for carrying the spatial and temporal characteristics of this phenomenon.

This methodology can be better explained from an unidimensional case, in which one unique velocity component will be generated on the interval $[a, b]$. The variable $f_\sigma(x)$ denotes a shape function of a turbulent spot, which presents a compact support on $[-\sigma, \sigma]$ and satisfies the normalization condition

$$\frac{1}{\Delta} = \int_{-\Delta/2}^{\Delta/2} f_\sigma^2(x)dx = 1, \tag{14}$$

where $\Delta = b - a + 2\sigma$. Each turbulent spot has a position $x_i$, a length scale $\sigma$ and receives a signal $\epsilon_i$. Thus, the contribution $u^{(i)}(x)$ of a turbulent spot to the velocity field, is defined as:

$$u^{(i)}(x) = \epsilon_i f_\sigma(x - x_i), \tag{15}$$

where $\epsilon_i$ represents a binary random variable, of value $-1$ or $+1$ and $x_i$ is drawn randomly on the interval $[a - \sigma, b + \sigma]$. The synthetic eddies are generated on an interval larger than $[a, b]$, in order to guarantee that the boundary points can be surrounded by eddies. The velocity signal at any point is the sum of the contributions of all synthetic eddies on the domain:

$$u(x) = \frac{1}{\sqrt{N}} \sum_{i=1}^{N} \epsilon_i f_\sigma(x - x_i), \tag{16}$$

in which $N$ denotes the quantity of synthetic eddies.

For 2d situations, the eddies are now 3d structures with compact three-dimensional supports on $[-\sigma_x, \sigma_x; -\sigma_y, \sigma_y; -\sigma_z, \sigma_z]$, satisfying a three-dimensional normalization condition of the same type as presented in Eq. (14). The inlet plane is located at $x = 0$ and it has dimensions $[0, L_z] \times [0, L_y]$. The position $(x_i, y_i, z_i)$ of synthetic eddy $i$ is drawn randomly on $[-\sigma_x, \sigma_x] \times [-\sigma_y, L_y + \sigma_y] \times [-\sigma_z, L_z + \sigma_z]$. The eddies are advected through the inlet plane with a reference velocity scale $U_0$, using Taylor's frozen turbulence hypothesis:

$$x_i(t + dt) = x_i(t) + U_0 dt \tag{17}$$

In case $x_i(t) > \sigma_x$, the synthetic eddie will be reallocated at $x = -\sigma_x$, in order to be advected again. Thus, the synthetic velocity signal is defined as:

$$u'_j(x, t) = \frac{1}{\sqrt{N}} \sum_{i=1}^{N} \epsilon_{ij} f_j(x - x_i(t)), \tag{18}$$

in which $\epsilon_{i,j}$ denotes the sign of vortex $i$ on component $j$ and are independent random steps of values $-1$ or $+1$.

The independence of $\epsilon_{ij}$ ensures that the generated inflow signal satisfies the condition $\overline{u_i u_j} = \delta_{ij}$. Thus, the availability of the Reynolds' stress tensor, $R_{ij}$ and of the mean velocity profile, $\bar{u}_i$, obtained from previous experiments, allows the transformation of the generated signal, in order to make it resembles these characteristics (Lund *et al.*, 1998). The final velocity field $u_i$ is, therefore, reconstructed from a synthetic field $u'_i$, a mean velocity profile and a Cholesky's decomposition, obtained from the provided Reynolds' stress tensor:

$$u_i = \bar{u}_i + a_{ij} u'_j, \tag{19}$$

in which $a_{ij}$ is related to the Cholesky's decomposition (Lund *et al.*, 1998).

In the next section, the validation of the computational code used and the results of these methodologies application are presented and discussed.

## *3. RESULTS AND DISCUSSIONS*

The computational code used in the present work, named Fluids 3D, was developed by Vedovoto *et al.* (2011), it is discretized by the finite volumes technique, composed by staggered variables three dimensional fields and, also, conservative. A centered differences scheme is applied for denoting the diffusive and advective contributions of the transport equations and a fully implicit approximation is adopted. The resultant linear

ISSN 2176-5480



systems are resolved by using the MSIP - Modified Strongly Implicit Procedure (Schneider and Zedan, 1981) for the velocity components. The mesh is cartesian, structured and uniform.

This computational code adopt an approximation based on the pressure. For this reason, an algorithm for the pressure-velocity coupling becomes necessary. Thereby, a projection method based on the fractional steps technique is applied, resulting in a Poisson's equation composed by variable coefficients, which is resolved with the solver BICGSTAB - Bi-Conjugate Gradient Stabilized (van der Vorst, 1981; Norris, 2001). This computational code was used to study the influence of different turbulence models and inlet boundary conditions in a flow characterization.

Moreau *et al.* (1996) performed experiments with and without reactions in a combustion chamber, called A3C. The data related to the longitudinal mean velocity fields $\bar{u}$ and its rms fluctuations $u'_{rms}$ were obtained, by the mentioned authors, using laser velocimetry.

The main characteristics of this experiment were the following: length $L = 0.9\ m$, height $H_d = 0.1\ m$, width $W = 0.1\ m$ and a backward-facing step with height $H = 0.035\ m$ and length $z_H = 0.1\ m$. The flow experimented $Re = 48750$ and the inlet mean velocity profile, fitted from the experimental data, was defined by:

$$u(z) = \begin{cases} \overline{U}\left\{1 - \left[\dfrac{z - \left(\dfrac{H_u}{2} + H\right)}{\dfrac{H_u}{2}}\right]^{\psi}\right\} & \text{if } H < z \leq H_d \\ 0 & \text{if } 0 \leq z \leq H, \end{cases} \tag{20}$$

where $\overline{U} = 55\ m/s$, $H_u = 0.065\ m$, $\psi = 10$ and $H = 0.035\ m$.

Since the studied situation was a turbulent flow, its calculations required larger amounts of computational resources. For this reason, the simulations were realized in a SGI Altix XE 1300 system, located at the MFLab, Federal University of Uberlândia. With this equipment, it was possible to perform numerical simulations of the mentioned experiment, using two different turbulence models and three distinct turbulent inlet generation methods. A brief description of the experiments developed in the present work is presented in Tab. 1:

Table 1. Numerical simulations performed in the present work.

| Mesh refinement | Number of processors | Turbulence model | Inlet boundary conditions generation method |
|---|---|---|---|
| 450x50x50 | 80 | Smagorinsky | White noise |
| 450x50x50 | 80 | Dynamic | White noise |
| 450x50x50 | 100 | Dynamic | *Random Flow Generation* (1000 Fourier modes) |
| 450x50x50 | 36 | Dynamic | *Synthetic Eddy Method* (10000 eddies) |
| 450x50x50 | 100 | Dynamic | *Synthetic Eddy Method* (100000 eddies) |

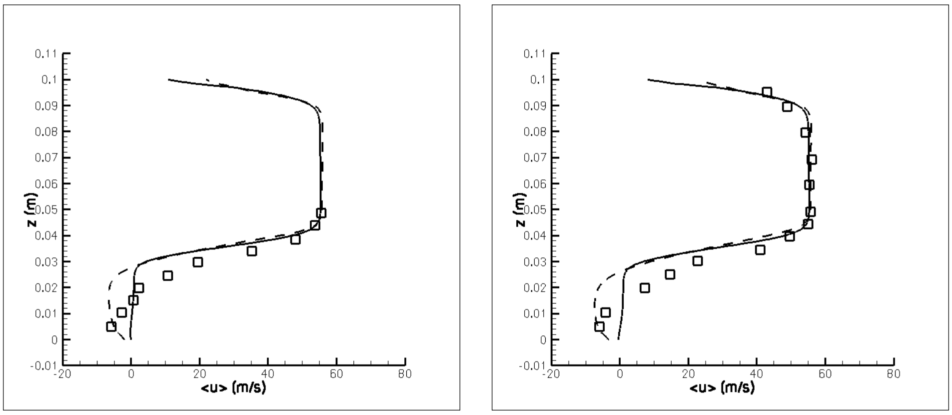
At first, it was realized a comparison between two different 3d simulations of the mentioned situation: the use of the classical turbulence model, proposed by Smagorinsky (1963), with $C_S = 0,18$, and the application of the dynamic Smagorinsky model, proposed by Germano *et al.* (1991). The obtained results were compared to the experimental data of Moreau *et al.* (1996).

Both simulations were performed using 80 processors, with a mesh refinement of 450x50x50 volumes. When the classical Smagorinsky model was applied, the calculations were developed until 3.55 physical seconds, which required 322,800 iterations and 233.54 hours of computational time. The application of the dynamic Smagorinsky model required different conditions. The calculations were performed until the simulation achieved 4.82 physical seconds, situation which required 447,740 iterations. In this case, it was required 318.48 hours to complete the numerical simulation.

The results presented in Fig. 1 show the mean velocity profiles determined in the present work and the experimental data obtained by Moreau *et al.* (1996). The mean velocity profiles were better represented when the dynamic Smagorinsky model was applied. It can be visualized by the analysis of the flow occurring close to the inferior wall. After that position, it is noticeable that this approximation was capable of following the tendency of the data obtained by Moreau *et al.* (1996). However, the calculated values were far from the absolute values achieved by the adopted reference.

ISSN 2176-5480

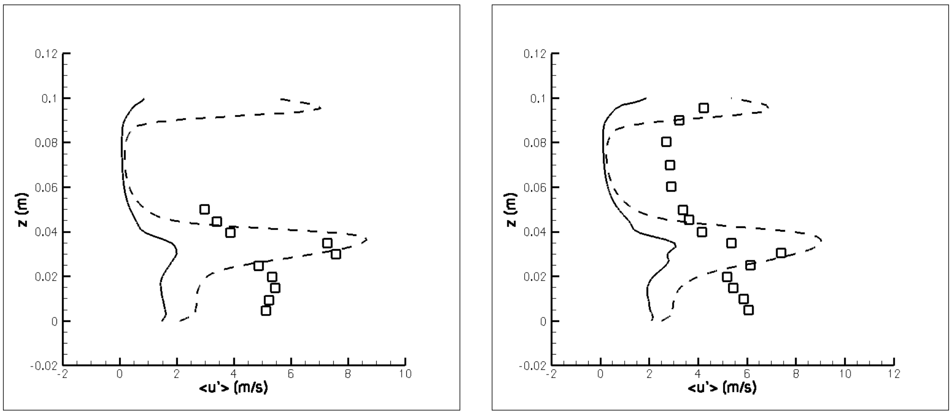





(a) (b)

Figure 1. Mean velocity profiles obtained from the application of the turbulence models proposed by Smagorinsky (1963) and Germano *et al.* (1991): (a) $x_m = 0.08\ m$ and (b) $x_m = 0.10\ m$. □ Moreau *et al.* (1996), — Classical Smagorinsky's model and - - - Dynamic Smagorinsky's model.

The results related to the mean velocity fluctuations, presented in Fig. 2, suggest that a better fit to the experimental data was also achieved when the approximation proposed by Germano *et al.* (1991) was used. The evaluation of a dynamic function, which fit to the flow in time and space instead of a constant, as performed when the classical Smagorinsky model is used, is the main reason to obtain better results. However, there is still a noticeable deviance between the numerical results and the experimental data.




(a) (b)

Figure 2. Mean velocity fluctuations, obtained from the application of the turbulence models proposed by Smagorinsky (1963) and Germano *et al.* (1991): (a) $x_m = 0.08\ m$ and (b) $x_m = 0.10\ m$. □ Moreau *et al.* (1996), — Classical Smagorinsky's model and - - - Dynamic Smagorinsky's model.

The significant difference between the calculated results and the experimental data boosted us to study and apply more realistic inlet boundary conditions to be used in the numerical simulations. For this purpose, an investigation of the influence of the turbulent inlet boundary conditions should be done.

Therefore, two distinct generation methods of turbulent inlet boundary conditions were applied to the following numerical simulations. At first, the methodology proposed by Smirnov *et al.* (2001), named Random Flow Generation (RFG), was applied. It consists of Fourier decompositions with coefficients calculated from spectral data obtained in different positions along the domain. Then, the approximation named Synthetic Eddy Method (SEM), proposed by Jarrin *et al.* (2006) was used. It is based on the creation of a box of eddies at the domain inlet.

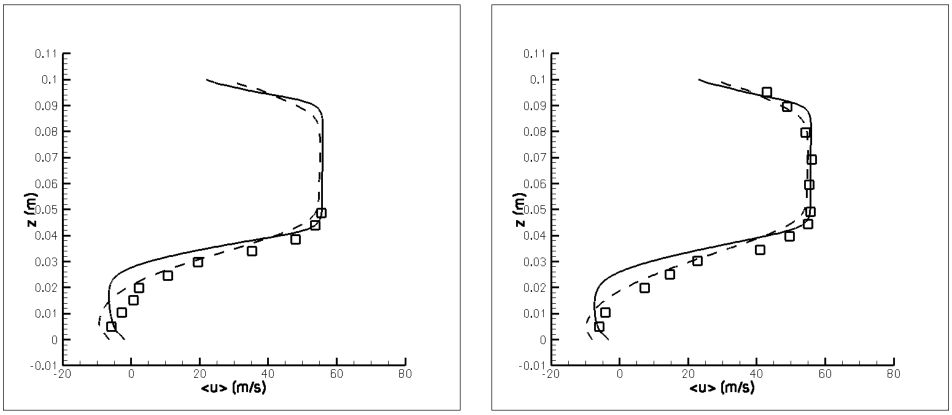
The application of RFG methodology on the studied problem was performed with 100 processors with a mesh of 450x50x50 volumes. The calculations were performed until 7.45 physical seconds, which required 502180 iterations. It was necessary 166.12 hours of computational time to conclude this simulation.

Figure 3 shows a better fitting when the calculated mean velocity profiles are compared with the experimental data obtained by the adopted reference. This affirmation can be evaluated by observing a more accurate characterization of this flow in regions closer to the bottom wall.

The analysis of the mean velocity fluctuations profiles, presented by Fig. 4, suggests a significant improvement

ISSN 2176-5480

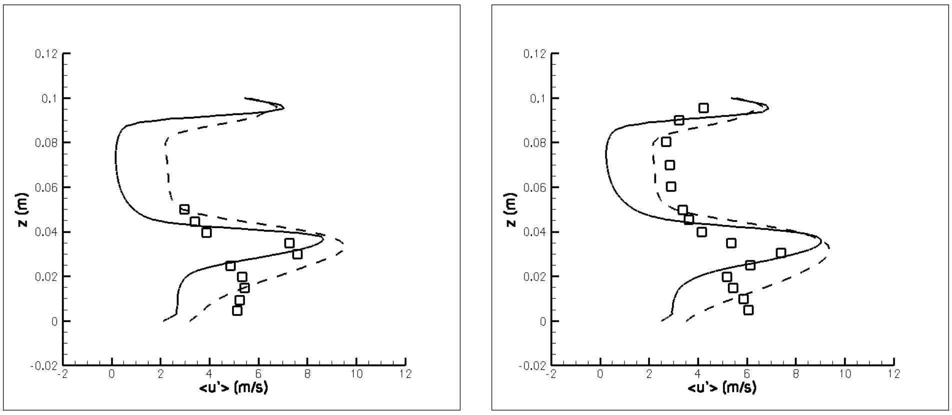





(a) (b)

Figure 3. Mean velocity profiles obtained from the application of the turbulence model proposed by Germano *et al.* (1991) and the RFG method for the inlet turbulent conditions: (a) $x_m = 0.08$ $m$ and (b) $x_m = 0.10$ $m$. □ Moreau *et al.* (1996), — Dynamic Smagorinsky's model and - - - Dynamic Smagorinsky's model with RFG.

to the obtained results, when these are compared to the adopted reference. It is noteworthy that the results obtained in the present work behave similarly to the experimental data and these are more consistent in regions further from the sudden expansion.




(a) (b)

Figure 4. Mean velocity fluctuations obtained from the application of the turbulence model proposed by Germano *et al.* (1991) and the RFG method for the inlet turbulent conditions: (a) $x_m = 0.08$ $m$ and (b) $x_m = 0.10$ $m$. □ Moreau *et al.* (1996), — Dynamic Smagorinsky's model and - - - Dynamic Smagorinsky's model with RFG.

After concluding the calculations presented before, studies concerning the methodology proposed by Jarrin *et al.* (2006) were performed. The Synthetic Eddy Method was firstly implemented in a computational code dedicated to evaluate its performance. Such a numerical code uses Reynolds' stress tensors experimental data and a mean velocity profile, along with the mentioned method, in order to create a turbulent velocity signal at the domain inlet.

This methodology depends on the local turbulence characteristic length which is determined, according to Pope (2000), using Eq. (21):

$$L = \frac{k^{3/2}}{\epsilon}, \tag{21}$$

where $k = \frac{\overline{u'^2}+\overline{v'^2}+\overline{w'^2}}{2}$, $L$ is the integral scale and $\epsilon$ is the dissipation rate of turbulent kinetic energy.

However, it is well known that the determination of the dissipation rate of a flow is hard to be achieved. It is needed for the determination of the necessary integral characteristic length. For this reason, some attempts were realized and studied using the computational code mentioned earlier.

The influence of the number of eddies applied and the performed iterations was evaluated from a set of numerical simulations, which consisted of calculations with 10,000 and 100,000 iterations with six different quantities of eddies and a time step with a constant value of $3.6 \times 10^{-5}$ $s$. These evaluations were based on a comparison between the experimental Reynolds' stress tensors experimented by the adopted reference and the results obtained in the present work.

At first, a characteristic length based on the step height, with a constant value of $L = 0.035\ m$ was used. At this time, the proposed simulations were divided in two groups, $A$ and $B$. The first was composed by 10000 iterations and six different quantities of generated eddies (100, 500, 1000, 2000, 10000 and 50000). The second differs only on the quantity of iterations, which was 100000.

Then, another set of simulations was based on a proposition performed by Pope (2000), in which the dissipation rate was equivalent to the product between the kinematic viscosity and the strain rate, which is given by Eq. (22):

$$\epsilon \equiv 2\nu S_{ij} S_{ij}. \tag{22}$$

As a consequence, it was possible to determine the dissipation rate from the kinematic viscosity, the mean velocity and the step height, all obtained from the studied flow. This formulation is presented in Eq. (23):

$$\epsilon = 2\nu \left(\frac{\partial \overline{u}_i}{\partial x_j}\right)\left(\frac{\partial \overline{u}_i}{\partial x_j}\right) \approx 2\nu \frac{\overline{U}^2}{H^2}. \tag{23}$$

With the obtained dissipation rate, it was possible to realize the proposed simulations. Similarly to the first proposition, the calculations were divided in two groups, $C$ and $D$, in which the only difference is the characteristic length determination.

The obtained $R_{11}$ Reynolds' stress tensor component was compared with the experimental data by the $L_2$ norm, obtained by Eq. (24):

$$L_2 = \sqrt{\frac{1}{N}\sum_{i=1}^{N}(R_{11_{calc}}^{(i)} - R_{11_{exp}}^{(i)})^2}, \tag{24}$$

where $N$ denotes the quantity of points in the domain. The achieved results are presented in Fig. 5.

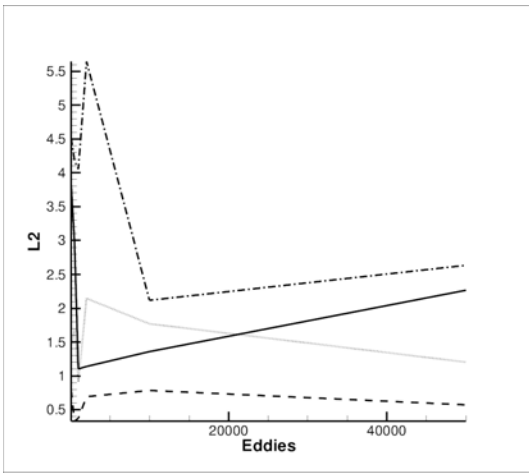



Figure 5. Influence of the quantity of generated eddies and the characteristic length determination proposition in the $L_2$ norm. — case A, - - - case B, - · - case C and ··· case D

It is worth to note that there is a significant variance between the $L_2$ norm values, according to the quantity of generated eddies. It is noteworthy that the best results were achieved when the cases B and D were simulated, in this situations, a larger quantity of iterations was used. In the aforementioned cases it is also perceptible that the increase of the number of eddies results in better values of the analyzed norm.

The influence of the quantity of generated eddies and the characteristic length determination proposition in the processing time required by the proposed methodologies is shown in Fig. 6. It is possible to note that increasing the quantity of generated eddies in this methodology results in a linear increase of computational time required to conclude the calculations. As expected, the cases which uses a larger amount of iterations (B and D) required more computational resources. At last, but not the least, it is worth to note that the application of the first proposition, which was based on the application of a constant characteristic length, was the most expensive among the cases studied, while the use of the second proposition, based on Eq. (22), required fewer computational resources.

ISSN 2176-5480



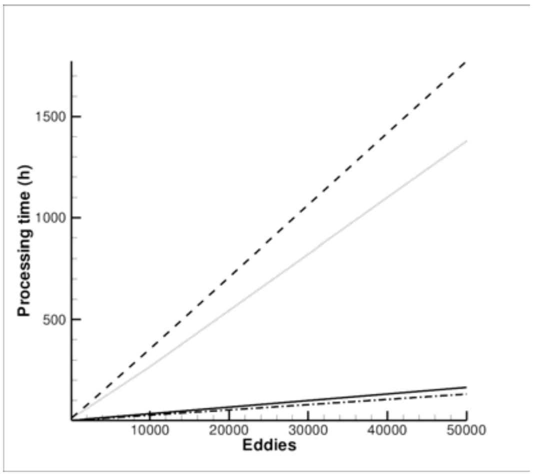



Figure 6. Influence of the quantity of generated eddies and the characteristic length determination proposition in the processing time required by the proposed methodology. — case A, - - - case B, - · - case C and ··· case D.

The results obtained with the appliance of the Synthetic Eddy Method were very promising, mainly for presenting the turbulent kinetic energy distributed in a $-5/3$ slope along the flow frequencies, when this spectrum is evaluated. For this reason, this methodology was implemented in the FLUIDS 3D code, developed by Vedovoto *et al.* (2011) and it was used to characterize the flow experimented by Moreau *et al.* (1996). For this purpose, four numerical simulations were performed.

At first, the study of the influence of the quantity of generated eddies in the determination of the mean velocity and fluctuation profiles was developed. This analysis was performed with two different numerical simulations.

In the first simulation, 10,000 eddies were generated and 36 processors were used with a mesh of 450x50x50 volumes. The calculations were performed until 1.22 physical seconds, which required 87,180 iterations and 71.88 hours of computational time. In the second one, by the other hand, 100,000 eddies were generated and 100 processors were used with the same mesh refinement. The calculations were performed until 2.20 physical seconds, which required 155,360 iterations and 94.51 hours of computational time. The achieved results are shown in Fig. 7 and Fig. 8.

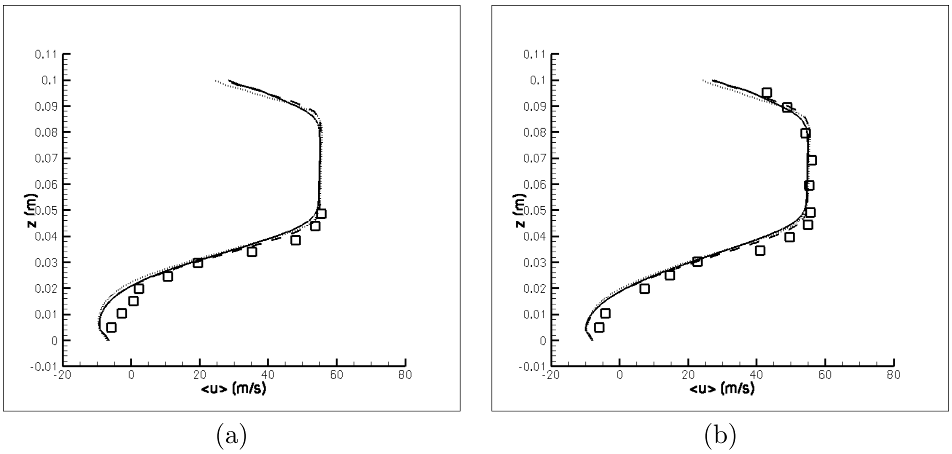


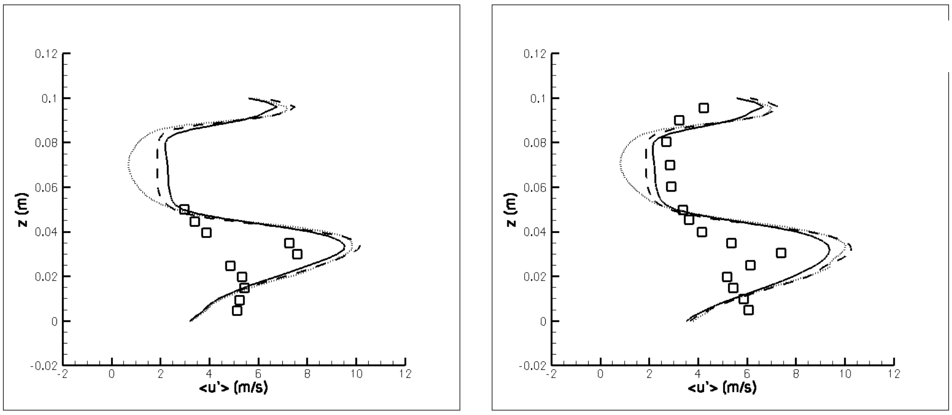

Figure 7. Mean velocity profiles obtained from the application of the turbulence model proposed by Germano *et al.* (1991) and the use of SEM and RFG methods for the turbulent inlet conditions: (a) $x_m = 0.08\ m$ and (b) $x_m = 0.10\ m$. □ Moreau *et al.* (1996), — RFG, - - - SEM - 10000 eddies, - · - SEM - 100000 eddies.

From the analysis of the mean velocity profiles, presented by Fig. 7, it is possible to observe that there is not a slightly difference between the application of 10000 or 100000 eddies. A larger amount of eddies is responsible for an improvement to what was obtained when the RFG method was used.

From the evaluation of the mean velocity fluctuation profiles, presented in Fig. 8, it is possible to reinforce what was commented over the last paragraph. A larger amount of generated eddies was also responsible for a better characterization of the studied flow. For this reason, the situation in which the larger amount of generated

ISSN 2176-5480



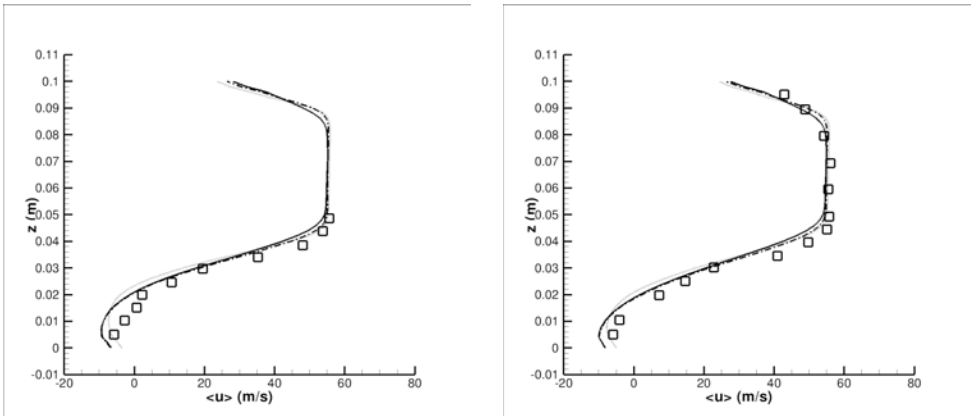



(a) (b)

Figure 8. Mean velocity fluctuations obtained from the application of the turbulence model proposed by Germano *et al.* (1991) and the use of SEM and RFG methods for the inlet turbulent conditions: (a) $x_m = 0.08\ m$ and (b) $x_m = 0.10\ m$. □ Moreau *et al.* (1996), — RFG, - - - SEM - 10000 eddies, - · - SEM - 100000 eddies.

eddies is used was applied to the last numerical simulations performed, which differ solely in the characteristic length calculation methodology.

The simulation in which the calculation of the characteristic length was performed via Eq. (22) required 114.4 hours of computational time for the numerical simulation of 2.14 physical seconds and 151,570 iterations. When a constant characteristic length was applied, 167.7 hours were needed to perform 350,880 iterations and 4.95 physical seconds. The mean velocity and mean velocity fluctuation profiles are shown in Fig. 9 and Fig. 10.




(a) (b)

Figure 9. Mean velocity profiles obtained from the application of the turbulence model proposed by Germano *et al.* (1991) and the use of SEM and RFG methods for the turbulent inlet conditions: (a) $x_m = 0.08\ m$ and (b) $x_m = 0.10\ m$. □ Moreau *et al.* (1996), — RFG, ··· SEM - case B, - · - SEM - case D.

The analysis of the presented mean velocity profiles is sufficient to observe that the appliance of a constant characteristic length was capable to generate a better description of the flow, when a comparison with the other results is realized.

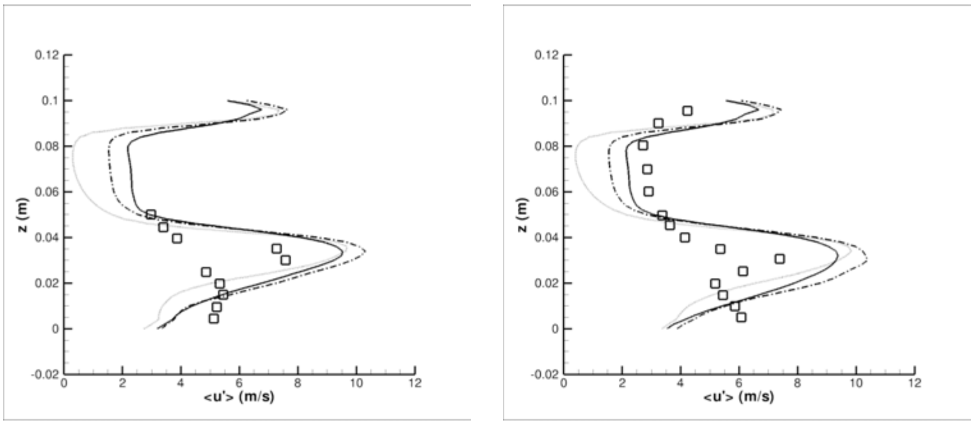
From the evaluation of the mean velocity fluctuations, it is noteworthy that, among the application propositions of the SEM, the use of Eq. (21) and Eq. (23), was the methodology which resulted in a better characterization of the studied flow. However, the best description was obtained using the methodology proposed by Smirnov *et al.* (2001).

A comparison between the computational costs required by both turbulent inlet conditions generators is shown in Tab. 2.

From the analysis of this table, it is worth to observe that, when an equal amount of physical time simulated is analyzed, the application of a constant characteristic length with the methodology proposed by Jarrin *et al.* (2006) was the simulation which required the smaller computational resources. The method proposed by Smirnov *et al.* (2001) needed the smallest quantity of iterations for developing the same task. Finally, the appliance of Eq. (21) and Eq. (22) for the determination of the characteristic length in SEM required a larger amount of

ISSN 2176-5480






(a) (b)

Figure 10. Mean velocity fluctuations obtained from the application of the turbulence model proposed by Germano *et al.* (1991) and the use of SEM and RFG methods for the inlet turbulent conditions: (a) $x_m = 0.08$ $m$ and (b) $x_m = 0.10$ $m$. □ Moreau *et al.* (1996), — RFG, ··· SEM - case B, - · - SEM - case D.

Table 2. Computational cost required by the turbulent inlet conditions proposed by Smirnov *et al.* (2001) and Jarrin *et al.* (2006).

| Inlet conditions generation method | Simulated physical time ($s$) | Processing time ($h$) | Quantity of Iterations |
|---|---|---|---|
| Random Flow Generation | | 67,05 | 141820 |
| Synthetic Eddy Method - case 1 | 2.10 | 64.95 | 149060 |
| Synthetic Eddy Method - case 2 | | 112.48 | 148550 |

computational time and iterations to perform the proposed calculations.

# 4. CONCLUSIONS

The realization of this work was intended to study the influence of turbulent inlet boundary conditions in numerical simulations of flows. The study of application of more realistic inlet conditions in large-eddy simulations was also an essential part of the present work.

The aforementioned studies were performed with a computational code developed by Vedovoto *et al.* (2011), in which numerical simulations involving combinations of turbulence modeling and turbulent inlet conditions generation methods were realized. The obtained results were compared with experiments performed by Moreau *et al.* (1996). These experiments denoted a turbulent flow occurring inside a combustion chamber.

At first, the classical and dynamic Smagorinky's turbulence models were compared. From the analysis of these simulations, it was possible to realize that the dynamic Smagorinky's turbulence model gives a better flow characterization, when compared with the classical model. The presence of a function capable of adjusting itself to the flow in time and space, over the use of a constant value, was the main motive for this improvement.

The application of realistic turbulent inlet boundary conditions methods resulted in even better results, mainly when the mean velocity fluctuation profiles were analyzed. It is due to a better distribution of the turbulent kinetic energy, presenting the larger amount of energy transported by the larger turbulent structures.

Interesting results were obtained with the application of the Synthetic Eddy Method. This methodology achieved the best flow description when the mean velocities were analyzed. However, the mean velocity fluctuation profiles were better characterized by Random Flow Generation method.

From an exclusive analysis of the results obtained with the application of the SEM, it is noticeable that the use of a larger amount of eddies resulted in a better characterization of the mean velocity fluctuation profiles. It was observed, also, that the hypothesis in which the dissipation rate was equivalent to the product between the kinematic viscosity and the strain rate achieved the best description of the flow experimented by Moreau *et al.* (1996).

Another important conclusion is related to the computational cost of each methodology. When an equal period of time was simulated, it was noticeable that the application of a constant characteristic length in the SEM was the methodology which required the smallest computational cost. On the other hand, the smallest quantity of iterations was needed when the RFG method was applied.

## 5. ACKNOWLEDGEMENTS

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

## 6. REFERENCES

Ferziger, J.H. and Perić, 2002. *Computational Methods for Fluid Dynamics*. Springer-Verlag.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H., 1991. "A dynamic subgrid-scale eddy viscosity model". *Physics of Fluids A: Fluid Dynamics*, Vol. 3, pp. 1760 – 1765.

Jarrin, N., Benhamadouche, S., Laurence, D. and Prosser, R., 2006. "A synthetic-eddy-method for generating inflow conditions for large-eddy simulations". *International Journal of Heat and Fluid Flow*, Vol. 27, pp. 585 – 593.

Kraichnan, R.H., 1970. "Diffusion by a random velocity field". *Physics of Fluids*, Vol. 13, pp. 22 – 31.

Lee, T. and Mateescu, D., 1998. "Experimental and numerical investigation of 2-d backward-facing step flow". *Journal of Fluids and Structures*, Vol. 12, pp. 703 – 716.

Lilly, D.K., 1967. "The Representation of Small-Scale Turbulence in Numerical Simulation Experiments". In *IBM Scientific Computing Symposium on Environmental Sciences*. Yorktown Heights, New York, USA, pp. 195–210.

Lund, T., Wu, X. and Squires, D., 1998. "Generation of turbulent inflow data for spatially-developing boundary layer simulations". *Journal of Computational Physics*, Vol. 140, pp. 233 – 258.

Mariano, F.P., 2011. *Solução Numérica das Equações de Navier-Stokes usando uma Hibridação das Metodologias Fronteira Imersa e Pseudo-Espectral de Fourier*. Ph.D. thesis, Universidade Federal de Uberlândia, Uberlândia, Brasil.

Moreau, P., Tanguy, B., Gicquel, P., Poirot, M. and Sauzin, J.L., 1996. "Validation expérimentale du modèle peul et du code diamant dans le cadre de l'opération a3c". *Technical Report*, Vol. ONERA.

Norris, S.E., 2001. *A Parallel Navier Stokes Solver for Natural Convection and Free Surface Flow*. Ph.D. thesis, Faculty of Mechanical Engineering, University of Sydney, Sydney, Australia.

Pope, S.B., 2000. *Turbulent Flows*. Cambridge University Press.

Schneider, G.E. and Zedan, M., 1981. "A modified strongly implicit procedure for the numerical solution of field problems". *Numerical Heat Transfer*, Vol. 4, pp. 1 – 19.

Shaanan, S., Ferziger, J.H. and Reynolds, W.C., 1975. "Numerical simulation of turbulence in presence of shear". *Rep. TF-6, Dept. Mechanical Engineering, Stanford University*.

Silveira-Neto, A.d., Grand, D., Metais, O. and Lesieur, M., 1993. "A numerical investigation of the coherent vortices in turbulence behind a backward-facing step". *Journal of Fluid Mechanics*, Vol. 256, pp. 1 – 25.

Smagorinsky, J., 1963. "General circulation experiments with the primitive equations". *American Metereological Society*, Vol. 91, pp. 99 – 164.

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, pp. 359 – 371.

van der Vorst, H.A., 1981. "Iterative solution methods for certain sparse linear systems with a non-symmetric matrix arising from pde-problems". *Journal of Computational Physics*, Vol. 44, pp. 1 – 19.

van Driest, E.R., 1956. "On turbulent flow near a wall". *Journal of the Aeronautical Sciences*, Vol. 23, pp. 1007 – 1011.

Vedovoto, J.M., Silveira-Neto, A.d., Mura, A. and Silva, L.F.F.d., 2011. "Application of the method of manufactured solutions to the verification of a pressure-based finite-volume numerical scheme". *Computers & Fluids*, Vol. 51, pp. 85 – 99.

## 7. RESPONSIBILITY NOTICE

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