XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

# Investigation of SST and LES turbulence models in the analysis of the wind action on a suspension bridge via CFD simulations

Montiel, José Emanuel da Silva[1], Corrêa, Laís[2], Kurokawa, Fernando Akira[1]

[1]*Dept. of Civil Construction Engineering, University of São Paulo*
*Av. Prof. Almeida Prado, trav. 2, 83, 05.508-900, SP/São Paulo, Brazil*
*jose.montiel@usp.br, fernando.kurokawa@usp.br*
[2]*Dept. of Exact Sciences and Technology, Federal University of Grande Dourados*
*Dourados/Itahum Highway, km 12 - Unit II, 79.804-970, MS/Dourados, Brazil*
*laiscorrea@ufgd.edu.br*

**Abstract.** In recent years, the use of computational techniques for fluid dynamics for the quantitative prediction of flow characteristics has grown exponentially. With the advance of computational technology, CFD (Computational Fluid Dynamics) simulation of numerous complex problems in various engineering areas, such as fluid-structure interaction, became possible. However, some difficulties appear when working with these problems using the CFD approach. In general, flows are turbulent with high Reynolds numbers, which makes it necessary to use a turbulence model. Thus, it is very important to use an adequate turbulence model, especially when it is intended to analyze the dynamic response of the bridge deck, where it is essential to find out the effect of the vortex shedding. In this sense, the main objective of this work will be to investigate the turbulence models SST and LES via CFD simulation in fluid-structure problems one way, in order to analyze the behavior of the wind action on a cross section of suspension bridge. For this, the Strouhal number and the aerodynamic coefficients will be obtained: drag, lift and moment of the sectional model submitted to a wind flow with a determined velocity and different angles of attack.

**Keywords:** Computational Fluid Dynamics, Aerodynamic coefficients, Fluid-structure interaction, Strouhal number, Vortex shedding

## 1 Introduction

Due to advances in construction technology and a great evolution in the civil engineering field over the last years, ever-more resistant materials have been developed, affording the construction of slender, lightweight, and flexible structures. In addition, with human needs of the modern world, civil structures, such as long-span suspension bridges and high-rise buildings, have become increasingly larger. Thus, due to problems caused by wind-induced loads and their respective dynamic action, these structures deserve special attention (Zhang et al. [1]).

In case of bodies positioned horizontally in the flow, such as, a long-span suspension bridge, the wind loads generate the aerodynamic forces of drag, lift and moment. There are also the forces from the fluid-structure interaction (FSI) that can induce to oscillations in the structure (Hou et al. 2012). These oscillations cause modifications in structural damping, in which can lead to the arise of instability phenomenon. Instability problems are frequent in slender and flexible bridges, and this can lead a structure to fail (Limas [2]).

If there is a phenomenon resulting from the action of a fluid flow around a structure in which a wake formation arises after the body, forming alternately detached eddies and causing vibrations, it occurs an instability called flow-induced vibration (Blessmann [3]). Thus, it is justified the importance of CFD analysis to understand better structure aerodynamics and also wind and structure interaction.

In order to analyze the aerodynamic and aeroelastic coefficients and the Strouhal number inherent in this fluid-structure interaction, this paper aims to analyze the interaction between the cross section of a suspension bridge and the wind using CFD analysis. However, some shortcomings appear when working with these problems. In general, flows are turbulent with high Reynolds numbers, which makes it necessary to use a turbulence model.

Thus, it is important to use an appropriate turbulence model, especially when it is intended to analyze the dynamic response of the bridge deck where it is essential to ascertain the effect of the vortex shedding. In this sense, we intend to investigate the models: LES and k-w SST, with the view to evaluate these models, by comparing them with some reference's values.

## 2 Mathematical Description

### 2.1 Governing Equations

The governing equations that model Newtonian incompressible flows are given by eq. (1) and eq. (2).

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_i u_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial u_i}{\partial x_j}\right) + f_i \tag{2}$$

where $u_i$ are the velocity components, $t$ is time, $\rho$ is the specific mass of the fluid, $p$ is the pressure, $\nu$ is the kinematic viscosity and $f_i$ are the external forces.

### 2.2 Turbulence Models

The turbulence models investigated in this work are: SST (Shear Stress Transport) by Menter [4] and LES (Large Eddy Simulation) by Smagorinsky [5].

***LES (Large Eddy Simulation).*** This turbulence model was developed by Smagorinsky [5]. The basic idea of LES is to solve the large scales turbulent spectrum, while modeling small structures. This strategy is based on the separation of turbulent scales, removing the small sub-grid scales (SGS). Turbulent stress defined by SGS technique are:

$$(\overline{\rho u_j u_i} - \bar{\rho}\tilde{u}_j\tilde{u}_i) = \tau_{SGS} - \frac{1}{3}\tau_{ii}\delta_{ii} = -2\nu_{SGS}\bar{S}_{ij} = \nu_{SGS}\left[\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right] = \frac{1}{3}\tau_{ij}\delta_{ij} \tag{3}$$

$\bar{S}_{ij}$ is the strain tensor and it is given by:

$$\bar{S}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right). \tag{4}$$

The kinematic viscosity is related to dynamic viscosity through $\nu_{SGS} = \frac{\mu_{sgs}}{\rho}$, in which:

$$\mu_{SGS} = \rho\left(C_{SGS}\Delta\right)^2 |\bar{S}| = \rho\left(C_{SGS}\Delta\right)^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}} \tag{5}$$

The constant $C_{SGS}$, which is related to the effects of average flow, shear and deformation, varies between 0.1 and 0.24.

***SST (Shear Stress Transport).*** The SST model is also known as an adaptation of the BSL (Baseline $\kappa$ - $\omega$) model. It was proposed by Menter [4] in which two other models are combined, $\kappa - \varepsilon$ and $\kappa - \omega$, (Launder and Spalding [6]; Yakhot *et al* [7]). In the external region of the flow, the $\kappa - \varepsilon$ model formulation is applied. In the region close to the wall, the transport equations of the $\kappa - \omega$ model are used.

The SST turbulence model is composed by two transport equations: one is the turbulent kinetic energy equation ($\kappa$), eq. (6), and the other is the rate of turbulent kinetic energy specific dissipation equation ($\omega$), given by eq. (7).

$$\frac{\partial \kappa}{\partial t} + \frac{\partial\left(\kappa \bar{u}_j\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\kappa}\right)\frac{\partial \kappa}{\partial x_j}\right] + P_\kappa - C_\mu \kappa\omega \tag{6}$$

where $\bar{u}_j$ are the velocity components, $\nu_t$ is the turbulent viscosity. The term $P_\kappa$ is given by:

$$P_\kappa = 2\nu_t \bar{S}_{ij}\frac{\partial \bar{u}_j}{\partial x_j} \tag{7}$$

The equation of turbulent kinetic energy dissipation, $\omega$, for the SST model is given by:

$$\frac{\partial \omega}{\partial t}+\frac{\partial\left(\omega \bar{u}_j\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right]+\alpha\frac{\omega}{\kappa}P_k-\beta\omega^2+2\left(1-F_1\right)\frac{\sigma_\omega}{\omega}\frac{\partial \kappa}{\partial x_i}\frac{\partial \omega}{\partial x_i} \tag{8}$$

The terms $\sigma_\kappa$, $\sigma_\omega$, $\alpha$, $\beta$ e $C_\mu$ are empirical constants of the model. The term $F_1$ is given by:

$$F_1=\tanh\left\{\min\left[\max\left(\frac{\sqrt{\kappa}}{C_\mu \omega d};\frac{500\nu}{d^2\omega}\right);\frac{4\rho\sigma_{\omega 2}\kappa}{CD_{\kappa\omega}d^2}\right]\right\} \tag{9}$$

in which $y$ is the distance from the no-slip surface, $\sigma_{\omega 2}$ is a constant and $CD_{\kappa\omega}$ is defined by:

$$CD_{\kappa\omega}=\max\left(2\rho\sigma_{\omega_2}\frac{1}{\omega}\frac{\partial \kappa}{\partial x_i}\frac{\partial \omega}{\partial x_i},10^{-20}\right) \tag{10}$$

the turbulent viscosity is calculated as follows:

$$\nu_t=\frac{a_1\kappa}{\max\left(a_1\omega;|\bar{S}|F_2\right)} \tag{11}$$

being $a_1$ an empirical constant equal to 0.3 and $|\bar{S}|$ the modulus of the mean flow strain rate tensor. $F_2$ is determined by:

$$F_2=\tanh\left\{\left[\max\left(\frac{2\sqrt{\kappa}}{C_\mu d};\frac{500\nu}{d^2\omega}\right)\right]^2\right\} \tag{12}$$

The constants $\sigma_\kappa$, $\sigma_\omega$, $\alpha$ and $\beta$ are determined by $\phi=F_1\phi_{\kappa-\omega}+(1-F_1)\phi_{\kappa-\epsilon}$, where $\phi_{\kappa-\omega}$ are the coefficients of the $\kappa-\omega$ model and $\phi_{\kappa-\varepsilon}$ are the coefficients of the $\kappa-\varepsilon$ model. The constants of the $\kappa-\omega$ SST model are presented in table 1.

Table 1. Constant values in the SST turbulence model

| $C_\mu$ | $\alpha_{\kappa-\omega}$ | $\beta_{\kappa-\omega}$ | $\sigma_{\kappa,\kappa-\omega}$ | $\sigma_{\omega,\kappa-\omega}$ | $\alpha_{\kappa-\varepsilon}$ | $\beta_{\kappa-\varepsilon}$ | $\sigma_{\kappa,\kappa-\varepsilon}$ | $\sigma_{\omega,\kappa-\varepsilon}$ |
|---|---|---|---|---|---|---|---|---|
| 0.09 | 5/9 | 3/40 | 0.85 | 0.5 | 0.44 | 0.0828 | 1 | 0.856 |

### 2.3 Fluid-structure interaction

Fluid structure interaction is a complex phenomenon of interaction between two continuous media, where the action of the fluid surrounding the solid produces static and dynamics loads that tend to elastically deform the structure, inducing vibration (Mannini [8]). If this vibration presents considerable displacements, then it is capable of interfering in the fluid flow around the solid, changing its aerodynamic behavior.

The aerodynamic drag $(C_D)$, lift $(C_L)$ and torsional moment $(C_M)$ coefficients are important dimensionless parameters of the flow interacting with the immersed body, which depend on the geometric characteristics of the cross section of the object, the angle of incidence of the fluid on the structure and also the Reynolds number of the flow. These coefficients are given respectively, by:

$$C_D=\frac{\overline{F}_D}{\frac{1}{2}\rho U^2 A} \qquad C_L=\frac{\overline{F}_L}{\frac{1}{2}\rho U^2 A} \qquad C_M=\frac{\overline{M}_T}{\frac{1}{2}\rho U^2 A^2} \tag{13}$$

where $F_D$ and $F_L$ are the mean drag and lift forces, respectively, $M_T$ is the mean torsional moment, $\rho$ is the specific mass of the fluid, $U$ is the mean velocity and $A$ is the area of reference. specific Reynolds number, in the wake downstream of the solid an eddy formation arises characterized by the alternate vortex shedding (Blessmann [3]). This phenomenon, known as Von Kármán vortices, occurs with a characteristic frequency and gives rise to periodic and oblique forces in relation to the fluid direction. The components of these forces tend to produce oscillations that occur in a specific detachment frequency of each pair of vortices (Limas [2]). The linear relationship between this vortex shedding frequency $(f_v)$ and the flow velocity defines another dimensionless flow parameter called Strouhal number (St), given by:

$$St = \frac{f_v A}{U} \tag{14}$$

The Strouhal number depends on the geometric characteristics of the solid and the Reynolds number and has an approximate value of $0.2$ for flat plates, such as a bridge deck (Righi [9]).

### 2.4 Initial conditions and boundary conditions

In this work, for the initial conditions, all variables are prescribed at the beginning of the calculations (Dirichlet condition). At the domain inlet, velocities are prescribed and at the domain outlet, it is assumed that the flow is completely developed with static pressure equal to zero Pa. In the upper and lower walls of the domain the condition of no-slip was imposed and on the front and posterior walls the condition of symmetry was imposed.

### 2.5 Numerical methodology

The governing equations, together with the boundary conditions, are solved numerically in the ANSYS CFX® 17.0 simulation environment, which employs a generalized form of the element-based finite volume method.

The numerical solution used by the tool is a partially coupled method. To involve pressure in the mass conservation equation, the software uses the Rhie-Chow interpolation scheme (Rhie and Chow, 1983) for the velocities in this equation. The transient terms of the equations are approximated by the 2nd order Implicit Euler method. The diffusive terms are approximated by central differences while the advective terms are discretized using the 2nd order Upwind scheme.

In addition, the CFX can be used together with the component modules of the ANSYS Workbench platform, whose environment integrates several applications such as the model and geometry generator *Design Modeler*, the ICEM CFD mesh generator, the CFD *pre* configuration and pre-processing tool, the CFX solver and the CFD *post* post processing tool.

## 3 Case study

In order to investigate the turbulence models SST and LES in fluid-structure problems, simulations of a flow around a cross section of the Great Belt East suspension bridge deck were performed.

The bridge section used in the simulations have the same characteristics and configurations presented in the work of Braun [10] in order to establish a comparative study.

Although the flow is considered two-dimensional, elements were used in the generation of the mesh, due to the characteristics of the ANSYS CFX solver, creating an unitary element in the neglected dimension of the domain ($Z$-axis). The computer simulations were carried out in a transient regime for $30s$, with a time step of $\Delta t = 1.8 \times 10^{-4}$.

The bridge has a cellular deck in a box beam with a structure with truss stiffness, as shown in Fig. 1.

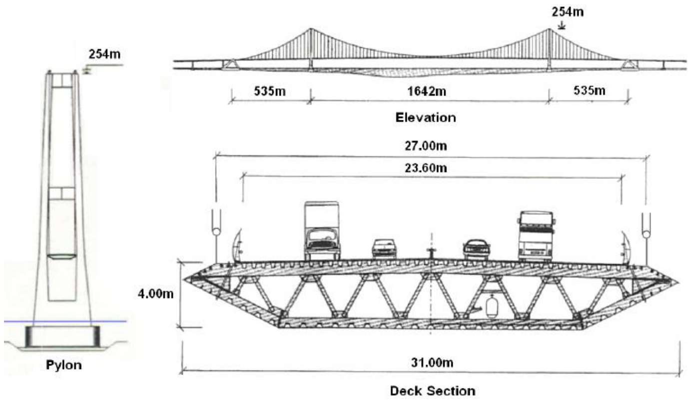


Figure 1. Elevation and cross section of the Great Belt East bridge

The computational domain and conditions for this problem are the same as those used in Braun and Awruch [11], and are illustrated in Fig. 2.

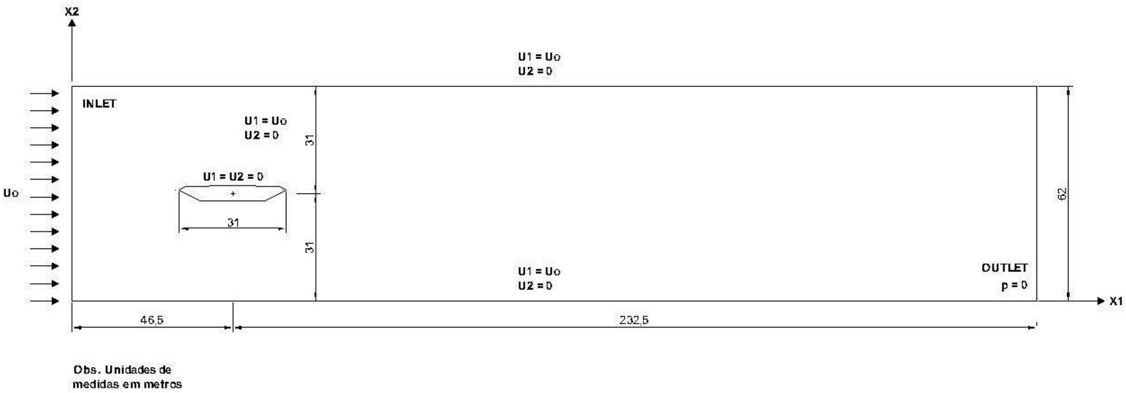


Figure 2. Schematic illustration of the computational domain of the flow around the cross section of the bridge

The angle of attack of the wind used for the simulations of this case was $\alpha = 0°$. In addition, an unstructured mesh was used, composed of 465735 elements and 141030 knots. The mesh contains prismatic elements with a quadrangular base (type *Hex* 8) and prismatic elements with a triangular base (type *Wed* 6) as shown in Fig. 3. The Reynolds number adopted was $3 \times 10^5$ with a time step of $1.15 \times 10^{-4}$ seconds. The flow occurs in a transient regime and the total simulation time is 30 seconds.

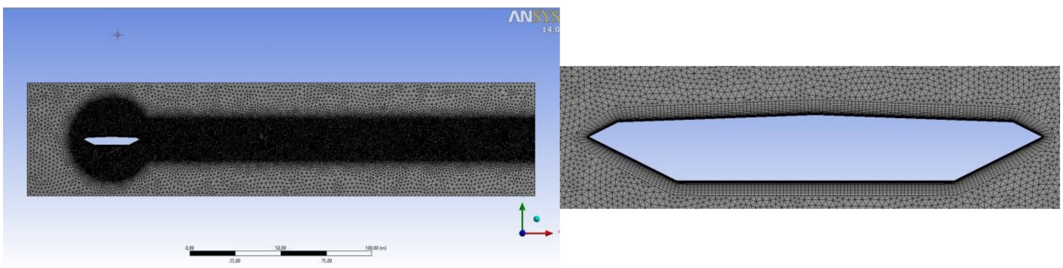

Figure 3. Mesh generated with *ICEM CFD* with refinement details on the edges of the deck

Other flow parameters are: fluid specific mass $(\rho) = 1.32\ kg/m^3$; dynamic viscosity $(\mu) = 5.78 \times 10^{-4}\ Ns/m^2$; inlet velocity $(U_0) = 40.0\ m/s$ and the characteristic length of the cross section $(D) = 31.0\ m$.

## 4 Numerical Results

The results obtained using LES and SST turbulence models for the drag coefficient $(C_D)$, lift coefficient $(C_L)$, torsional moment coefficient $(C_M)$ and Strouhal number $(St)$ were compared with the results of Braun [10] in Tab. 2. Pressure fields and velocity fields are also presented to characterize the phenomenon vortex shedding. These results are shown in Fig. 4 and Fig. 5.

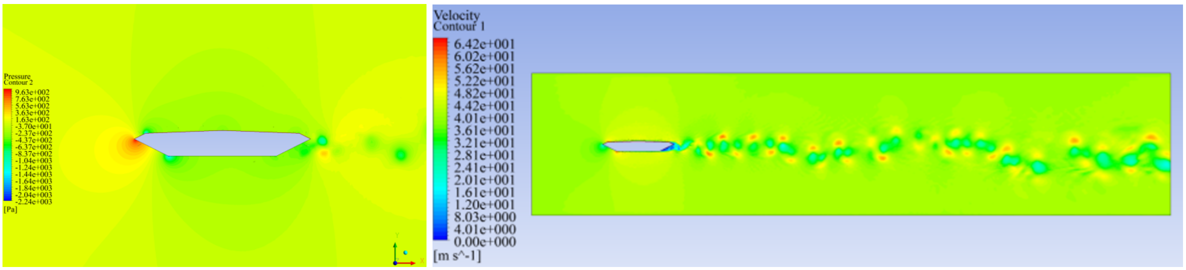


Figure 4. Pressure field (left) and Velocity field (right) in the region close to the bridge cross section for $Re{=}3{\times}10^5$ and using LES.

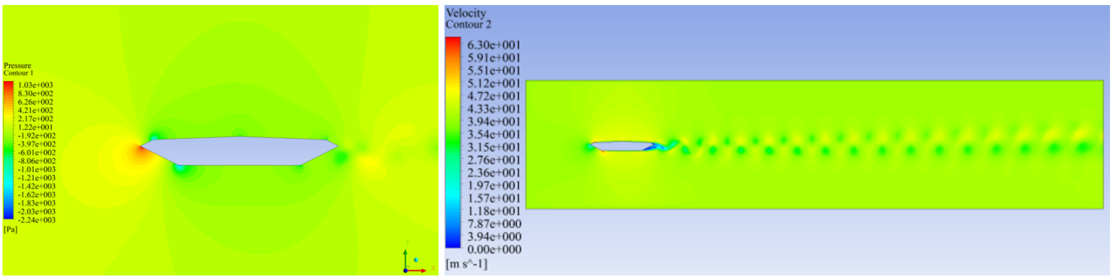


Figure 5. Pressure field (left) and Velocity field (right) in the region close to the bridge cross section for $Re{=}3{\times}10^5$ and using SST.

It is possible to observe in Fig. 4 the formation of a vortex wake, in which the detachment occurs alternately for the LES model.

Table 2. Results of the aerodynamic coefficients and the Strouhal number for different turbulence models

| **Model** | **Results** | | | | |
|---|---|---|---|---|---|
| | $Re = 3 \times 10^5$ | | | | |
| | $C_D$ | $C_L$ | $C_M$ | $St$ | Courant number |
| **LES (Present work)** | 0.63 | 0.05 | 0.05 | 0.179 | 2.80 |
| **SST (Present work)** | 0.63 | 0.05 | 0.04 | 0.189 | 1.40 |
| **LES (Braun and Awruch [12])** | 0.63 | 0.05 | 0.05 | 0.180 | - |

Table 2 has shown that the results of the aerodynamic coefficients obtained with both turbulence models are in excellent agreement with the results of Braun and Awruch [12]. In addition, it is possible to notice that the Strouhal number obtained by the LES is also in excellent agreement. The result obtained with the LES model is the one that best approximates to Braun and Awruch [12] results.

## 5 Conclusions

The case study presented in this work aimed to investigate and verify the behavior of two turbulence models in flows around blunt bodies.

The profile of the Great Belt East bridge, whose section resembles a flat plate, was analyzed. It was possible to observe the vortex shedding with both models. However, qualitatively, the SST model has shown a better result. The LES model presented a certain instability in the von Kármán vortex street due to the convective scheme used.

## References

[1] Zhang, W., Ge, Y., & Cai, C., 2013. Evaluating wind loads on bridge decks using velocity fields. *Journal of Engineering Mechanics*, vol. 139, n. 3, pp. 339–346.
[2] Limas, L. F., 2007. Determinação da influência de seções transversais na resposta dinâmica de pontes através de ensaios em túnel de vento e identificação de sistemas.
[3] Blessmann, J., 2005. Introdução ao estudo da ação dinâmica do vento.
[4] Menter, F., 1993. Zonal two equation kw turbulence models for aerodynamic flows. In *23rd fluid dynamics, plasmadynamics, and lasers conference*, pp. 2906.
[5] Smagorinsky, J., 1963. General circulation experiments with the primitive equations: I. the basic experiment. *Monthly weather review*, vol. 91, n. 3, pp. 99–164.
[6] Spalding, D., 1974. The numerical computation of turbulent flow. *Comp. Methods Appl. Mech. Eng.*, vol. 3, pp. 269.
[7] Yakhot, V., Orszag, S., Thangam, S., Gatski, T., & Speziale, C., 1992. Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A: Fluid Dynamics*, vol. 4, n. 7, pp. 1510–1520.
[8] Mannini, C., 2006. *Flutter vulnerability assessment of flexible bridges*. PhD thesis, PhD thesis, Technische Universitaet Carolo-Wilhelmina Braunschweig and University of Florence.
[9] Righi, M., 2003. Aeroelastic stability of long span suspended bridges: Flutter mechanism on rectangular cylinders in smooth and turbulent flow. *DIC, Universitadegli Studi di Firenze*.
[10] Braun, A. L., 2007. Simulação numérica na engenharia do vento incluindo efeitos de interação fluido-estrutura.
[11] Braun, A. L., 2002. Um modelo para a simulação numérica da ação do vento sobre seções de ponte.
[12] Braun, A. L. & Awruch, A. M., 2003. Numerical simulation of the wind action on a long-span bridge deck. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, vol. 25, n. 4, pp. 352–363.