A numerical model to study the aerodynamic and aeroelastic bridge deck behavior is presented in this paper. The flow around a rigid fixed bridge cross-section, as well as the flow around the same cross-section with torsional motion, are investigated to obtain the aerodynamic coefficients, the Strouhal number and to determine the critical wind speed originating dynamic instability due to flutter. The two-dimensional flow is analyzed employing the pseudo-compressibility approach, with an Arbitrary Lagrange-Eulerian (ALE) formulation and an explicit two-step Taylor-Galerkin fluid. The finite element method (FEM) is used for spatial discretization. The structure is considered as a rigid body with elastic restraints for the cross-section rotation and displacement components. The fluid-structure interaction is accomplished applying the compatibility and equilibrium conditions at the fluid-solid interface. The structural dynamic analysis is performed using the classical Newmark’s method.

Keywords: Fluid-structure interaction, Finite Element Method (FEM), Large Eddy Simulation (LES), aerelasticity, aerodynamics

Governing Equations for the Flow Simulation

The governing equations, considering the pseudo-compressibility approach in an incompressible process, Large Eddy Simulation (LES) with Smagorinsky’s model for turbulent flows and an Arbitrary Lagrange-Eulerian (ALE) description, are:

a) Momentum equations:

\[ \frac{\partial \mathbf{v}_i}{\partial t} + (\mathbf{v}_j - \mathbf{v}_i) \frac{\partial \mathbf{v}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \mathbf{v}_i}{\partial x_j} + \frac{\partial \mathbf{v}_j}{\partial x_i} \right) + 2 \tau_{ij} \right] \]

\[ (i, j = 1, 2) \text{ in } \Omega \]

being \( \mathbf{v}_i = (c_1 \Delta)^2 (2 S_x S_y)^{1/2} \) with \( S_y = \frac{1}{2} \frac{\partial v_y}{\partial x} + \frac{\partial v_x}{\partial y} \) and

\[ \tilde{\mathbf{A}} = \left( \tilde{A}_x, \tilde{A}_y, \tilde{A}_z \right) \]

\[ \tilde{A}_i = \left( \tilde{A}_x, \tilde{A}_y, \tilde{A}_z \right) \]

b) Mass conservation equation:

\[ \frac{\partial \rho}{\partial t} + (\mathbf{v}_j - \mathbf{v}_i) \frac{\partial \rho}{\partial x_j} + \rho \bar{c}_D \frac{\partial \mathbf{v}_i}{\partial x_j} = 0 \]

\[ (j = 1, 2) \text{ in } \Omega \]

which is obtained considering \( \frac{\partial \rho}{\partial t} = c^2 \).

The boundary conditions of Eqs. (1) and (2) are the following:
\( v_i = w_i \) (i = 1, 2) on the solid boundary \( \tilde{A}_i \),

\( \nu = \tilde{\nu} \) on the boundary \( \tilde{A}_p \),

\( p = \hat{p} \) on the boundary \( \tilde{A}_p \),

\[
\left[ -\frac{p}{\bar{h}} \bar{a}_y + (i + j) \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) + \frac{\partial v_k}{\partial x_k} \right] a_j = \frac{\partial j}{\partial x_j} = S_i
\]

(i, j, k = 1, 2) in \( \tilde{A}_b \) (5)

In these equations, \( v_i \) and \( p \) (the velocity components and the pressure, respectively) are the unknowns. The viscosities \( i = \frac{i}{\bar{h}} \) and \( \bar{h} = \frac{\sqrt{\bar{h}}}{\sqrt{\bar{h}}} \), the specific mass \( \bar{h} \) and the sound velocity \( c_v \) are the fluid properties. The eddy viscosity \( i = \frac{i}{\bar{h}} \) depends of derivatives of the filtered velocity components, of the element dimensions and of the Smagorinsky’s constant \( C_S \). For a purely Eulerian description, the mesh motion velocity \( w \) at each nodal point, with components \( w_j \), is equal to zero. Now, for a purely Lagrangean description, the mesh motion velocity at each nodal point is equal to the fluid velocity, i.e., \( v_j = w_i \) (i = 1, 2). Finally, in an Arbitrary Lagrangean-Eulerian formulation, \( w \neq 0 \) and \( w \neq v \).

On the boundaries \( \bar{A}_w \) and \( \bar{A}_p \), prescribed values for velocity and pressure, \( \tilde{v} \) and \( \hat{p} \), respectively, must be specified, while on \( \bar{A}_s \) the boundary force \( \tilde{f} \) must be in equilibrium with the stress tensor components \( \tilde{\sigma}_y \). In Eq. (5), \( n_j \) is the direction cosine between a vector perpendicular to \( \tilde{A}_b \) and the axis \( x_j \).

Initial conditions for the pressure and the velocity components at \( t = 0 \) must be given.

**The Algorithm for the Flow Simulation**

Expanding the governing equations in a Taylor’s series up to second order terms, the algorithm for the flow simulation contains the following steps (Braun, 2002):

1) Calculate \( \tilde{v}_i^{n+1/2} \) with:

\[
\tilde{v}_i^{n+1/2} = v_i^n + \frac{\Delta t}{2} \left[ -\frac{\partial v_i}{\partial x_j} \frac{1}{\bar{h}} \frac{\partial p}{\partial x_j} + \frac{\partial v_j}{\partial x_j} \right] + \frac{\Delta t}{4} \left[ \frac{\partial ^2 v_i}{\partial x_j \partial x_k} \right] (6)
\]

where \( r_j = (v_j - w_j) \) and \( \bar{v} = (\nu + v_\nu) \).

2) Calculate \( p^{n+1/2} \) with:

\[
p^{n+1/2} = p^n + \frac{\Delta t}{2} \left[ -\frac{\partial p}{\partial x_j} \frac{1}{\bar{h}} \frac{\partial v_i}{\partial x_j} - \frac{\partial ^2 v_i}{\partial x_j \partial x_k} \right] + \frac{\Delta t}{4} \left[ \frac{\partial ^2 v_i}{\partial x_j \partial x_k} \right] (7)
\]

3) Calculate \( \tilde{A} p^{n+1/2} = \tilde{A} p^{n+1/2} - p^n \).

4) Calculate \( v_i^{n+1/2} \) with:

\[
v_i^{n+1/2} = v_i^n + \frac{\Delta t}{4} \frac{\partial ^2 v_i}{\partial x_j \partial x_k} (8)
\]

5) Calculate \( v_i^{n+1} = v_i^n + \tilde{A} \nu_i^n \) with:

\[
\tilde{A} \nu_i^n = \tilde{A} \nu_i^n + \frac{\partial \nu_i}{\partial x_j} + \frac{\partial v_i}{\partial x_j} \left[ \frac{\partial v_j}{\partial x_j} + \frac{\partial v_j}{\partial x_j} \right] + \frac{\partial ^2 v_i}{\partial x_j \partial x_k} (9)
\]

6) Calculate \( \nu_i^{n+1} = \nu_i^n + \tilde{A} \nu_i^n \) with:

\[
\tilde{A} \nu_i^n = \tilde{A} \nu_i^n + \frac{\partial \nu_i}{\partial x_j} + \frac{\partial v_i}{\partial x_j} \left[ \frac{\partial v_j}{\partial x_j} + \frac{\partial v_j}{\partial x_j} \right] + \frac{\partial ^2 v_i}{\partial x_j \partial x_k} (10)
\]

These expressions must be employed after applying the classical Galerkin technique into the finite element method (MEF) context.

As the scheme is explicit, the resulting system is conditionally stable, with a stability condition given by:

\[
\tilde{A} i < \frac{\tilde{A} \nu_i}{\nu_i + \tilde{A} \nu_i} \quad (i = 1, \ldots, NTE) (12)
\]

where \( \nu_i \) (which is a real number less than one) is a safety coefficient, \( \tilde{A} \nu_i \) and \( \nu_i \) are the i-th element characteristic dimension and the velocity, respectively, and \( NTE \) is the total number of elements.

Although variable time step could be adopted (Teixeira & Awrejcewicz, 2001), in this work an unique value of \( \tilde{A} t \) will be used for the whole process, adopting the smallest one from those obtained by Eq. (12).

**The Fluid-Structure Coupling**

In the present work, the structure is idealized as a two-dimensional rigid body. Displacement and rotations take place on the plane formed by the axis \( x_1 \) and \( x_2 \); the body is restricted by dampers and springs, as indicated in Fig. 1.
The structural dynamic equilibrium equation is given by the following matrix expression:

$$\begin{align*}
\dot{U}^c &+ C \ddot{U}^c + K \dot{U}^c = Q^c \\
\sim_S &\sim_S \sim_S \sim_S \sim_S \sim_S \\
\sim_S &\sim_S \sim_S \sim_S \sim_S \sim_S \\
\sim_S &\sim_S \sim_S \sim_S \sim_S \sim_S
\end{align*}$$  \tag{13}

where $M$ is the mass matrix, $C$ the damping matrix, $K$ the stiffness matrix and $\dot{U}^c, \ddot{U}^c$ and $\dot{U}^c$ the acceleration, velocity and generalized displacements, respectively. Finally, $Q^c$ is the load vector.

The subscript $S$ means that these matrices belong to the structure and the superscript $C$ indicates that these values correspond to the gravity center of the solid body. Equation (13) can be written as follows:

$$
\begin{bmatrix}
M_1 & 0 & 0 & m_1 \\
0 & M_2 & 0 & m_2 \\
0 & 0 & M_3 & m_3 \\
K_{11} & 0 & 0 & k_1 \\
0 & K_{22} & 0 & k_2 \\
0 & 0 & K_{33} & k_3
\end{bmatrix}
\begin{bmatrix}
\dot{u}_1 \\
\dot{u}_2 \\
\dot{u}_3 \\
\dot{v}_1 \\
\dot{v}_2 \\
\dot{v}_3
\end{bmatrix}
+ \begin{bmatrix}
C_{11} & 0 & 0 & c_1 \\
0 & C_{22} & 0 & c_2 \\
0 & 0 & C_{33} & c_3
\end{bmatrix}
\begin{bmatrix}
\dot{u}_1 \\
\dot{u}_2 \\
\dot{u}_3 \\
\dot{v}_1 \\
\dot{v}_2 \\
\dot{v}_3
\end{bmatrix}
+ \begin{bmatrix}
Q_{S1} \\
Q_{S2} \\
M_{S3}
\end{bmatrix} = 0
\tag{14}
$$

It must be noticed that the hypothesis of a rigid structure is proper when deformations of the cross-section are small compared to the rotation and displacement components.

At the solid-fluid interface, the compatibility condition must be satisfied, or in other words, the fluid velocity and the structure velocity must be the same at the common nodes of both fields. The compatibility condition and the translation of variables evaluated at the center of gravity of the body to a point located at the fluid-structure interface may be written with the following expressions:

$$
\begin{align*}
\dot{U}^I &\sim_S \sim_S \sim_F \sim_S \sim_S \\
\dot{V}^I &\sim_S \sim_S \sim_S \sim_S \sim_S \\
\sim_S &\sim_S \sim_S \sim_S \sim_S \sim_S \\
\sim_S &\sim_S \sim_S \sim_S \sim_S \sim_S \\
\sim_S &\sim_S \sim_S \sim_S \sim_S \sim_S
\end{align*}
$$  \tag{15}

where $S$ and $F$ are referred to the structure and the fluid, respectively, and the superscript $I$ is referred to the interface. It is important to notice that the both vectors $\dot{U}^I$ and $\dot{V}^I$ have two components that correspond to the global axis direction. However, $\dot{U}^c$ has three components, because it includes the rotation around an axis perpendicular to the plane formed by $x_1$ and $x_2$. Values of $\dot{U}^c$ can be transported to the solid-fluid interface (or to nodes belonging to the structure boundary) through a translation matrix $L$, as given by Eq. (15), being $l_1$ and $l_2$ the distance components between the gravity center of the body and the point under consideration, measured in the global system. Considering Fig. 2, it is observed that the distance components from a boundary point to the body gravity center are functions of $\dot{e}$, and it may be written as follows:

$$
\begin{bmatrix}
l_1(\dot{e}) \\
l_2(\dot{e})
\end{bmatrix}
= \begin{bmatrix}
\cos \dot{e} & -\sin \dot{e} \\
\sin \dot{e} & \cos \dot{e}
\end{bmatrix}
\begin{bmatrix}
x_{1g} \\
x_{2g}
\end{bmatrix}
= Rx
\tag{16}
$$

Deriving Eq. (15) with respect to time, taking into account matrix $L$ and equation (16), the following expression is obtained:

$$
\dot{U}^I = L \dot{U}^c + L'(\dot{e}) \dot{U}^c, \text{ where } L'(\dot{e}) = 
\begin{bmatrix}
0 & -l_2 \\
0 & l_1
\end{bmatrix}
\tag{17}
$$

Equation (15) and Eq. (17) are applied to each node at the interface, where the equilibrium condition must be also satisfied, that means that the load $S$ acting on the structure at the interface, must be equal to the load $S$ given by Eq. (5), but with an opposite signal (because here the fluid action on the structure is considered, while Eq. (5) represents the boundary action on the fluid). $S$ can be transported to the center of gravity of the body, obtaining:

$$
\dot{Q}^c = - \int_{S} L^T S dA
\tag{18}
$$

where $L^T$ is the transpose matrix of $L$, given by Eq. (15), and $S$ contains the two components of the fluid boundary force acting on the structure at a point located on the structure surface $A_S$ ($A_S$ represents also the solid-fluid interface); these forces $S$ are given by Eq. (5), but with an opposite signal.

![Figure 2. Rigid body motion. The subscripts “g” and “l” are referred to quantities related to global and local axis, respectively.](image)

To determine the coupling effects between the fluid and the structure, in the finite element method (FEM) context, consider an element belonging to the fluid domain in contact with the solid body, as indicated in Fig. 3, where it can be observed that only points 1 and 2 are in contact with the structure.
The momentum equations in its matricial form, at element level (e), can be obtained by applying the Galerkin method to the Eq. (1), writing:

\[
\begin{bmatrix}
\tilde{MM} & \tilde{MM}^F \\
\tilde{MM}^F & \tilde{MM}^F
\end{bmatrix}
\begin{bmatrix}
\tilde{V}^I \\
\tilde{V}^F
\end{bmatrix}
\begin{bmatrix}
\tilde{AD} & \tilde{AD}^F \\
\tilde{AD}^F & \tilde{AD}^F
\end{bmatrix}
\begin{bmatrix}
\tilde{V}^I \\
\tilde{V}^F
\end{bmatrix}
\end{equation}

where \( \tilde{MM} \) contains the time derivative coefficients from the velocity components \( \tilde{V} \). \( \tilde{AD} \) contains the coefficients of advective and diffusive terms, \( \tilde{GP} \) contains the coefficients of pressure derivative terms with respect to \( x_1 \) and \( x_2 \) and, finally, \( \tilde{S} \) is a vector containing the boundary integrals resulting from the integration by parts of pressure and diffusive terms.

In Eq. (19), \( \tilde{V}^I \) and \( \tilde{V}^F \) contain, respectively, acceleration and velocity components corresponding to nodes 1 and 2 of Fig. 3, while \( \tilde{V}^P \) and \( \tilde{V}^F \) contain variables corresponding to nodes 3 and 4 of the same figure. A similar remark can be made with respect to the vectors of pressure gradients \( \tilde{GP} \) and boundary forces \( \tilde{S} \). Matrix \( \tilde{MM}^H \) contains elements coming from the connection of node 1 with itself and with node 2, and the connection of node 2 with itself and with node 1. Matrix \( \tilde{MM}^{IF} \) reflects the connection between the nodes 1 with 4 and 2 with 3. Similar commentaries can be made with respect to matrices \( \tilde{AD}^H \) and \( \tilde{AD}^{IF} \).

Regarding the structural analysis, only the first matricial expression of Eq. (19) is necessary, because only this equation contributes to the assembling of the overall dynamic equilibrium equation. On the other hand, as the structural and the flow analysis are performed in a sequential form in this work, the system constituted by the solid body and fluid elements with one or more sides common to the solid-fluid interface have prescribed values of \( \tilde{V} \) and \( \tilde{P} \) at nodes that do not have any contact with the structure

(they were calculated previously in the flow analysis). Referring to Fig. 3, at nodes 3 and 4, the values \( \tilde{V}^I \) and \( \tilde{P}^F \) are known. All these considerations lead to the elimination of the second expression of Eq. (19) when the governing equations which describe the solid body motion are built, taking into account the solid-fluid coupling effect. The first expression of Eq. (19) can be re-written as:

\[
\tilde{MM}^H \tilde{V}^I + \tilde{AD}^H \tilde{V}^I + \tilde{MM}^{IF} \tilde{P}^F + \tilde{AD}^{IF} \tilde{V}^F - \frac{1}{\tilde{g}} \tilde{GP}^I = \tilde{S}^I
\]

Equation (15) with matrix \( \tilde{L} \) and Eq. (17) with matrix \( \tilde{L}'(\dot{\tilde{e}}) \) are considered for each node at the interface. Then, when an element side with two nodes and lying on the fluid-structure interface is considered, Eq (15) and Eq (17) are written in the following form:

\[
\tilde{U}^I = \tilde{V}^I = \tilde{T}\dot{\tilde{U}}^e \tilde{S}^I \quad \tilde{U}^I = \tilde{V}^I = \tilde{T}\dot{\tilde{U}}^e \tilde{S}^I
\]

Referring again to Fig. 3, the matrices \( \tilde{T} \) and \( \tilde{T}' \) are given by:

\[
\begin{bmatrix}
1 & 0 & -\tilde{I}_1^2 \\
0 & 1 & \tilde{I}_1^2 \\
0 & 0 & \tilde{I}_1^2
\end{bmatrix}
\begin{bmatrix}
\tilde{L} \\
\tilde{L} \\
\tilde{L}
\end{bmatrix}
= \begin{bmatrix}
0 & 0 & -\tilde{I}_1^2 \\
0 & 0 & -\tilde{I}_1^2 \\
0 & 0 & -\tilde{I}_1^2
\end{bmatrix}
\begin{bmatrix}
\tilde{L}'(\dot{\tilde{e}}) \\
\tilde{L}'(\dot{\tilde{e}}) \\
\tilde{L}'(\dot{\tilde{e}})
\end{bmatrix}
\]

The contribution from \( \tilde{S}^I \), on the side 1-2 of the element (e), to the total load acting at the gravity center of the body, can be calculated as:

\[
\tilde{Q} = -\tilde{T}^\top \tilde{S}^I
\]

Considering Eqs. (14), (15), (17) and (20), with the last one multiplied by \( \tilde{g} \), the structural dynamic equilibrium equation, taking into account the solid-fluid coupling effect, is given by:

\[
\begin{bmatrix}
\tilde{M} + \sum_{i=1}^{NL} \tilde{C}^T \tilde{T} \rho \tilde{MM}^H + \sum_{j=1}^{NL} \tilde{C}^T \tilde{T} \rho \tilde{MM}^{IF} \\
\sum_{i=1}^{NL} \tilde{T}^T \rho \tilde{AD}^H \tilde{T} + \sum_{j=1}^{NL} \tilde{T}^T \rho \tilde{AD}^{IF} \tilde{T} + \tilde{C} + \sum_{i=1}^{NL} \tilde{T}^T \rho \tilde{MM}^H \tilde{P}^F + \sum_{j=1}^{NL} \tilde{T}^T \rho \tilde{MM}^{IF} \tilde{P}^F
\end{bmatrix}
\begin{bmatrix}
\tilde{U}^e + \tilde{K} \tilde{U}^e \\
\tilde{U}^e \tilde{S}^I \tilde{U}^e \tilde{S}^I \tilde{U}^e \tilde{S}^I
\end{bmatrix}
= \tilde{Q}
\]

where \( NL \) is the total number of fluid elements in contact with the structure, having at least one straight segment common to the solid body surface, forming the solid-fluid interface. The matricial Eq. (24) is re-written as:

\[
\begin{bmatrix}
\tilde{M} \tilde{U}^e + \tilde{C} \tilde{U}^e + \tilde{U}^e \tilde{K} \tilde{U}^e \\
\sum_{i=1}^{NL} \tilde{T}^T \rho \tilde{MM}^H \tilde{V}^F + \sum_{j=1}^{NL} \tilde{T}^T \rho \tilde{MM}^{IF} \tilde{V}^F + \tilde{C} + \sum_{i=1}^{NL} \tilde{T}^T \rho \tilde{AD}^H \tilde{V}^F + \sum_{j=1}^{NL} \tilde{T}^T \rho \tilde{AD}^{IF} \tilde{V}^F + \tilde{C} + \sum_{i=1}^{NL} \tilde{T}^T \rho \tilde{MM}^H \tilde{P}^F + \sum_{j=1}^{NL} \tilde{T}^T \rho \tilde{MM}^{IF} \tilde{P}^F
\end{bmatrix}
\begin{bmatrix}
\tilde{U}^e + \tilde{K} \tilde{U}^e \\
\tilde{U}^e \tilde{S}^I \tilde{U}^e \tilde{S}^I \tilde{U}^e \tilde{S}^I
\end{bmatrix}
= \tilde{Q}
\]

Figure 3. Element of the fluid domain in contact with the solid body.

\[\text{J. of the Braz. Soc. of Mech. Sci. & Eng.}
\]
As can be noticed, $\tau$ is a non-symmetric matrix, because it contains the advective terms and $[\tau^T_{MM} \tau^T_{11} \tau^T_{12} \tau^T_{13}]$. This last term leads to the non-linearity of matrix $\tau$.

In this work, a monolithic coupling between fluid and structure was not considered. The analysis for both fields is made in a sequential way. Firstly, Eq. (6) to Eq. (11) are solved, with the smallest $\Delta t$ calculated with Eq. (12) and applying the boundary conditions given by Eq. (3) to Eq. (5). After, Eq. (25) is solved using the Newmark’s method (Bathe, 1996). Although different time steps may be used for the fluid and the structure, here the same time step was adopted, because the computer time required by the structure analysis is negligible with respect to the processing time demanded by the flow analysis. Furthermore, compatibility and equilibrium conditions are more accurately imposed if the same time intervals are employed.

**Strouhal Number and Aerodynamic Coefficients Calculation**

The Strouhal number ($S_t$) can be calculated with $S_t = \frac{f_s L_0}{V_0}$, where $V_0$ is a reference velocity, $L_0$ a reference dimension and $f_s$ is the shedding frequency of a pair of vortices. It depends of the immersed prism cross-section, its oscillations, its superficial details, the Reynolds number and the flow characteristics. Formulation to calculate this number is presented in many publications and texts (for example, Schlichting, 1979). When the Strouhal number of a flow with a given immersed structure is known, it is possible to obtain the velocity $V_{0,th}$, which will produce the resonant phenomenon on the vibrating body. It occurs when the shedding frequency of a pair of vortices is approximately equal to the structural natural frequency.

The drag coefficient $C_D$ is related to the acting forces on the structure in the flow direction, while the lift coefficient $C_L$ is related to the acting forces on the structure in the transversal-to-flow direction. Finally, the pitching moment coefficient $C_M$ is related to the torsional moment acting at the gravity center of the immersed prism. The three coefficients can be calculated using the following expressions:

$$C_D = \frac{\sum_{i=1}^{NTN} S_{yi}}{2 \bar{V}_0^2 L_0}; C_L = \frac{\sum_{i=1}^{NTN} S_{zi}}{2 \bar{V}_0^2 L_0}; C_M = \frac{\sum_{i=1}^{NTN} \left(-S_{yi} I_{yi} + S_{zi} I_{zi}\right)}{2 \bar{V}_0^2 L_0^2}$$

(26)

where $S_{yi}$ and $S_{zi}$ are the forces in the directions $x_1$ and $x_2$, respectively, acting on the structure at node $i$, located on the interface. $I_{yi}$ and $I_{zi}$ are the projections in the directions $x_1$ and $x_2$, respectively, of the distance between the gravity center and node $i$. $NTN$ is the total number of nodes located on the solid-fluid interface. The forces $S_{yi}$ and $S_{zi}$ are the components of the force vector $S_i$, given by Eq. (20). These forces are applied to the structure on each fluid element side belonging to the interface.

The pressure coefficient at a point $i$, $C_{p,i}$, located on the interface, is related to the pressure acting at that point. This coefficient can be calculated using the following expression:

$$C_{p,i} = \frac{P_i - P_0}{\frac{1}{2} \bar{V}_0^2}$$

(27)

where $P_i$ is the pressure at node $i$ and $P_0$ is a reference pressure (for example, the pressure in an undisturbed area of the flow). With the instantaneous values of pressure coefficients, the time history may be obtained and then the mean pressure distribution on the body surface, for a given time interval, may be calculated.

**The Automatic Mesh Motion Scheme**

Taking into account that the immersed body in the fluid can move and rotate in its plane and that the flow is described by an Arbitrary Lagrange-Eulerian (ALE) formulation, a scheme for the mesh motion is necessary, establishing the velocity field $w$ in the fluid domain, such that the element distortion will be as smaller as possible, according to the following boundary conditions:

$$w|_{interface} = \frac{V}{V_0} = \frac{U}{U_0} \text{; } w|_{external boundaries} = 0$$

(28)

In the present work, the mesh motion scheme is similar to that used by Teixeira & Awruch (2001). Considering that $i$ is an inner point in the fluid field and $j$ is a boundary node, the mesh velocity components at node $i$, in the direction of the axis $x_k$, are given by:

$$w_k = \sum_{j=1}^{NS} a_{ij} w_j (k = 1, 2)$$

(29)

where $NS$ is the total number of nodes belonging to the boundary lines and $a_{ij}$ are the influence coefficients between the inner points and the boundary lines of the flow field, being $a_{ij} = \frac{1}{(d_{ij})^n}$, where $d_{ij}$ is the distance between $i$ and $j$, and $n \geq 1$. The exponent $n$ can be adjusted by the user. Although regions with purely Eulerian and purely Lagrangean descriptions mat be used simultaneously with the ALE formulation, this alternative may result in more complex and less efficient codes. It may also lead to more difficulties to control mesh distortions.

**Examples**

**Analysis of the Flow Around a Rectangular Prism**

This example presents a prism with a rectangular cross-section free to oscillate in the transversal-to-flow and rotational directions. Through this problem, the program performance for large and coupled motion is observed, even that this cross-section form is not usually employed in bridge structures. In this study, special attention is given to the structural dynamic response in the two degrees of freedom of the cross-section and to the finite element mesh motion, remembering that a special scheme is employed for the non-linear dependence with respect to the cross-section rotation in the compatibility condition at the solid-fluid interface.

In the present section, results obtained for a flow around a rectangular prism with a Reynolds number equal to 1000 are presented. The rectangular cross-section exhibits a height/width relation ($h/B$) of 0.2. The geometry and the boundary conditions, in
a non-dimensional form, are shown in Fig. 4. In addition, initial velocity and pressure field for the fluid-structure interaction problem are those of a developed flow obtained with a fixed body.

The finite element mesh has 5865 nodes and 5700 quadrilateral bi-linear isoparametric elements and is shown in Fig. 5. A non-dimensional time step $\Delta t^* = 1.0 \times 10^{-4}$ was adopted. The fluid and structural data are presented in Table 1.

### Table 1. Rectangular prism: dimensionless data for the fluid and the structure.

<table>
<thead>
<tr>
<th>Fluid data</th>
<th>Rectangular Prism - Reynolds 1000</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific mass ($\rho$)</td>
<td>1.0</td>
</tr>
<tr>
<td>Volumetric viscosity ($\mu$)</td>
<td>0.0</td>
</tr>
<tr>
<td>Reynolds number ($Re$)</td>
<td>1000</td>
</tr>
<tr>
<td>Mach number ($M$)</td>
<td>0.06</td>
</tr>
<tr>
<td>Reference/inflow velocity ($V_0$)</td>
<td>1.0</td>
</tr>
<tr>
<td>Characteristic dimension ($D$)</td>
<td>1.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Structural data</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensionless longitudinal stiffness ($K_{11}^*$)</td>
<td>$3 \times 10^4$</td>
</tr>
<tr>
<td>Dimensionless transversal stiffness ($K_{22}^*$)</td>
<td>0.7864</td>
</tr>
<tr>
<td>Dimensionless torsional stiffness ($K_{33}^*$)</td>
<td>17.05</td>
</tr>
<tr>
<td>Dimensionless longitudinal mass ($M_{1}^*$)</td>
<td>195.57</td>
</tr>
<tr>
<td>Dimensionless torsional mass ($M_{3}^*$)</td>
<td>105.94</td>
</tr>
<tr>
<td>Dimensionless longitudinal damping ($C_{11}^*$)</td>
<td>$1 \times 10^7$</td>
</tr>
<tr>
<td>Dimensionless transversal damping ($C_{22}^*$)</td>
<td>0.0325</td>
</tr>
<tr>
<td>Dimensionless torsional damping ($C_{33}^*$)</td>
<td>0.0</td>
</tr>
</tbody>
</table>

In Fig. 6 the time histories related to angular and vertical displacements, velocities and accelerations are presented. It is important to notice that the time used in these figures is dimensionless. These results are very similar to those obtained by Sarrate et al. (2001), using a different method.
The streamlines and the pressure field are shown in Fig. 7, in three instants ($t^* = 439$, $t^* = 442$ and $t^* = 448$). It can be observed the presence of high pressure gradients and large vortices alternating between the lower and the higher surfaces. The streamlines show that the cross-section orientation with respect to the free flow direction modifies the boundary layer form. This conclusion is the same observed in bluff bodies, where the flux-ward dimension is one of the parameters that determine the forms of the boundary layer and wake. In Fig. 8 it is verified the distortion of the mesh in an instant where extreme structural rotation is reached.

Figure 7. Rectangular prism: (I) pressure contours and (II) streamlines contours; (a) $t^* = 439$; (b) $t^* = 442$ and (c) $t^* = 448$. 
Numerical Study of the Great Belt East Bridge Cross-section

In this section, results of the numerical simulation of the wind action on a cross-section belonging to the Great Belt East Bridge are presented, including the aerodynamic and the aeroelastic behavior. The studies are accomplished by fixed and oscillating sectional models, according to the usual wind tunnel techniques.

The Great Belt East Bridge is located in Denmark, precisely in the Great Belt Channel, an important international shipping route. The design phase was initiated in 1989, being it opened to the traffic in 1998. It is a suspension bridge, with a superstructure constituted by two approaching spans of 535 m (each one) and a central span of 1624 m, which will be studied in this work. In Fig. 9, general aspects of the bridge are shown. The pictures were taken from Larsen & Walther (1997).

Firstly, the fixed cross-section was analyzed and the aerodynamic coefficients were obtained as functions of the angle of attack of the wind direction. The Strouhal number was also calculated. Finally, free oscillations of the cross-section in the vertical and the rotational degrees of freedom were allowed in order to carry out dynamic instability investigations.

Analysis of the Flow Around the Fixed Cross-Section

The computational domain and the boundary conditions used in this example, are illustrated in Fig. 10. As can be noticed, the inflow boundary conditions are functions of the angle of attack of the wind direction. Four different values of the angle of attack were studied: -10°, -5°, 0° e +5°. The initial pressure and velocity were assumed equal to zero.

The finite element mesh employed in this problem has 8175 bilinear isoparametric elements with 8400 nodes, and is shown in Fig. 11.
The Reynolds number used in the four cases is $3.0 \times 10^5$. The other constants used in the analysis are presented in Table 2. From the well-known Courant stability condition, the time step is $\Delta t = 1.15 \times 10^{-4}$ s.

Table 2. Great Belt East Bridge: data used to determine aerodynamic coefficients.

<table>
<thead>
<tr>
<th>Constants</th>
<th>Great Belt East Bridge - Reynolds $3 \times 10^5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific mass ($\beta_1$)</td>
<td>1.32 Kg/m$^3$</td>
</tr>
<tr>
<td>Volumetric viscosity ($\beta_2$)</td>
<td>0.0 m$^2$/s</td>
</tr>
<tr>
<td>Kinematic viscosity ($\beta_3$)</td>
<td>$5.78 \times 10^{-4}$ m$^2$/s</td>
</tr>
<tr>
<td>Sound velocity ($c$)</td>
<td>337.0 m/s</td>
</tr>
<tr>
<td>Reference/inflow velocity ($V_0$)</td>
<td>40.0 m/s</td>
</tr>
<tr>
<td>Smagorinsky's constant ($C_S$)</td>
<td>0.2</td>
</tr>
<tr>
<td>Character dimension/cross-section(D)</td>
<td>4.40 m</td>
</tr>
</tbody>
</table>

The investigated mean coefficients, obtained from the time histories, are plotted in Fig. 12 as functions of the angle of attack, and compared with the experimental results given by Reinhold et al. (1992) and the numerical results obtained by Kuroda (1997).

![Figure 12. Great Belt East Bridge: numerical and experimental results for aerodynamic coefficients as functions of the angle of attack.](image)

The Strouhal number, obtained from the vertical velocity component time history $V_2$ at a point located a distance 0.2 $B$ behind the cross-section (with zero angle of attack), is 0.18. Comparisons of some of the results obtained for Strouhal number of the referred bridge are shown in Table 3.

Table 3. Strouhal number for the Great Belt East Bridge.

<table>
<thead>
<tr>
<th>Reference</th>
<th>Strouhal number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present work</td>
<td>0.180</td>
</tr>
<tr>
<td>Larsen et al. (1998) (numer.)</td>
<td>0.170</td>
</tr>
<tr>
<td>Wind tunnel tests (from: Larsen et al. (1998))</td>
<td>0.160</td>
</tr>
</tbody>
</table>

The streamlines observed for the different angles of attack are presented in Fig. 13 and are similar to those obtained by Kuroda (1997).

![Figure 13. Great Belt East Bridge: streamlines contours for different angles of attack.](image)

**Aeroelastic Analysis: Flutter**

In this work, Flutter analysis is accomplished in two different ways: a) direct method, presented by Selvam et al. (2002), and b) using the flutter derivative $A_2^*$, introduced by Scanlan & Tomko (1971), being this coefficient related to the aerodynamic damping due to torsional rotations. Both methods have the same experimental procedure and are based on the observation of the structural response to cross-section rotations for various wind velocity values.
In Selvam et al. (2002), it is shown a method in which the
growth/decay rate is determined from the structural response,
oberved in several reduced wind velocities given by \( \nu^* = \frac{V^*}{g} \),
where \( V^* \) is the inflow velocity, \( B \) is the bridge deck width and \( f \)
is the natural structural frequency. These values of the growth/decay
rate are calculated with \( \dot{\alpha} \exp = \left( y^k - y^{k+1} \right) / y^k \), where \( y^k \) and \( y^{k+1} \) are
the peak values in the same oscillation period. After, they are
transported to a chart in function of the reduced wind velocity, and
the critical velocity corresponds to the point where the curve crosses
the velocity axis (growth/decay rate = 0).

In the flutter derivatives method (Scanlan & Tomko, 1971), the
experimental damping \( \alpha_\exp \) and the natural frequency \( \dot{\omega}_\exp \) for
each reduced wind velocity are obtained from the structural
response. These values are introduced into an expression,
representing the aerodynamic damping and given by:

\[
A_2^* (\nu^*) = \frac{4 I}{B} \left[ \dot{\alpha} \exp \dot{u} \exp - \dot{u} \exp \right]
\]

where \( I \) is the mass moment of inertia, \( B \) is the specific mass of the
fluid, \( B \) is the bridge deck width, \( \alpha \) is the structural critical
damping and \( \dot{\omega} \) the structural natural frequency. Eq. (30) may be
also written, by experimental considerations, in a reduced
expression in terms of the logarithmic decrement \( \dot{\alpha} \exp \equiv 2 \delta \dot{\omega} \exp \) as follows:

\[
A_2^* (\nu^*) = - \frac{1}{\dot{\omega}} \frac{2 \dot{\alpha} \exp}{\delta} \exp
\]

Thus, a curve of this coefficient \( A_2^* \) in function of the reduced
wind velocities is built. The critical flutter velocity is obtained by a
critical condition expressed by:

\[
A_2^* = \frac{4 I \alpha_\exp}{B} \exp
\]

So, when \( A_2^* > 4 I \alpha_\exp / \dot{\omega} \) the aerodynamic damping is greater
than the structural damping, originating negative damping and
oscillations with growing amplitudes.

The geometry as well as the finite element mesh employed in
the determination of the critical velocity of flutter is the same that
was used previously (Fig. 10 and Fig. 11). Initially, a fixed cross-
section with an inclination of 1.8º was taken. After 30000 time
steps, the load boundary conditions at the body surface were
computed, and then, the body motion was allowed. The outflow
boundary conditions were kept identical to the case where the body
remains fixed with zero angle of attack, with exception to the inflow
velocity, which changes in order to obtain the desired curves.

The physical properties and design values of the structure,
employed in the experiments, are found in Table 4. The structure is
idealized such that only torsional rotations are allowed (because it
was verified that coupling vertical displacements and rotations will
not modify significatively the critical velocity of flutter).

The problem was analyzed for four reduced velocities: 2, 4, 6
and 10. These values correspond to the following inflow velocities:
16.86 m/s, 33.73 m/s, 50.59 m/s and 84.32 m/s, respectively. The
flow was analyzed with \( Re = 10^5 \).

<table>
<thead>
<tr>
<th>Great Belt East Bridge – Reynolds 10^5 – Structural data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longit. and transv. stiffness (K_{11}, K_{22})</td>
</tr>
<tr>
<td>Torsional stiffness (K_{33})</td>
</tr>
<tr>
<td>Longit. and transv. mass (M_{1}, M_{2})</td>
</tr>
<tr>
<td>Torsional mass (M_{3})</td>
</tr>
<tr>
<td>Longit. and transv. damping (C_{11}, C_{22})</td>
</tr>
<tr>
<td>Torsional damping (C_{33})</td>
</tr>
<tr>
<td>Vertical natural frequency (f_k)</td>
</tr>
<tr>
<td>Angular natural frequency (f_{k})</td>
</tr>
<tr>
<td>Critical damping (\alpha)</td>
</tr>
</tbody>
</table>

In Fig. 14, time histories related to the angular displacement are
presented for each reduced wind velocity. From the rotation time
histories, the growth/decay rate as well as the logarithmic decrement
for each reduced velocity were obtained. In Table 5 all these values
are presented.

![Figure 14. Great Belt East Bridge: angular displacement time histories for the studied reduced wind velocities.](image-url)
In Fig. 15 the curves to obtain the critical velocity of flutter by the direct method of Selvam et al. (2002) and by the flutter derivative $A^*$ (Scanlan & Tomko, 1971), respectively, are presented.

From Fig. 15, the reduced critical velocity obtained by the direct method is 8.18, corresponding to a velocity of 69 m/s. By the flutter derivative method, considering the critical condition as $A^*_2 \geq 1.62 \times 10^2$, yields a reduced velocity equal to 8.66 which corresponds to a critical velocity equal to 73 m/s. In Table 6 below, comparisons of the critical velocity obtained by this work and by other authors (through numerical and experimental works) are shown.

<table>
<thead>
<tr>
<th>Reference</th>
<th>$V_{cr}$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Present work – fundamental</td>
<td>69</td>
</tr>
<tr>
<td>Present work – fundamental</td>
<td>73</td>
</tr>
<tr>
<td>Present work – flutter derivative</td>
<td>65-72</td>
</tr>
<tr>
<td>Larsen et al. (1999) (num.)</td>
<td>74</td>
</tr>
<tr>
<td>Enevoldsen et al. (1999) (num.)</td>
<td>70-80</td>
</tr>
</tbody>
</table>

Table 5. Great Belt East Bridge: numerical results for flutter analysis.

<table>
<thead>
<tr>
<th>Great Belt East Bridge - Reynolds</th>
<th>$V^*$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V^*$ = 2</td>
<td>0.131</td>
</tr>
<tr>
<td>$V^*$ = 4</td>
<td>0.270</td>
</tr>
<tr>
<td>$V^*$ = 6</td>
<td>0.311</td>
</tr>
<tr>
<td>$V^*$ = 10</td>
<td>-0.500</td>
</tr>
</tbody>
</table>

Logarithmic decrement: 0.176 0.205 0.291 -0.403

Conclusions

A model for the numerical simulation of the wind action on bridges was described. The computational code was validated through the analysis of a rectangular cross-section and studies of the aerodynamic and aeroelastic behavior of the Great Belt East Bridge cross-section. The results show good agreement with those obtained by other authors. In future works, it is expected to explore other bridge decks, considering details such as cables, guardrails and aerodynamic appendages. It is also expected improvements in the efficiency in processing time, mainly in turbulent flows. An alternative is to use time integration with subcycles (Teixeira & Awruch, 2001), optimizing the time step on the computational domain. Another possibility is to use semi-implicit schemes for the flow analysis.

References


