Analysis of the fluid-structure interaction of a semi-rigid frame

Haridasa F. Carvalho[1], Francisco A. A. Gomes[2], Paulo R. Novak[2], Patricia Grezelle[1], Diego R. Rossetto[2]

[1]*Department of Civil Engineering, Federal University of Technology - Paraná*
[2]*Department of Mechanical Engineering, Federal University of Technology - Paraná*
Via do Conhecimento, KM 01, 85503-390, Pato Branco-Paraná, Brazil
haridasa@alunos.utfpr.edu.br, franciscogomes@utfpr.edu.br,novak@utfpr.edu.br,
patriciagrezelle@alunos.utfpr.edu.br, diegorossetto@utfpr.edu.br

Abstract. Fluid-Structure Interaction (FSI) is characterized as a non-linear multi-physical problem and is present in the most diverse areas of engineering, such as civil, mechanical, naval, aeronautical engineering works, among others. In steel structures, the forces arising from the wind are actions of relevant importance and due to their dynamic characteristics, many uncertainties exist, hindering the work of the structural engineer. Among the types of connections used in practice, the bolt connections stand out for their simplicity and speed in the process of assembling steel structures, both small and large. The large number of variables existing in this type of connection - thickness of the plates involved in the connection, diameter and positioning of the bolts, etc. - however, they make it difficult to analyze their behavior. In this work, a fluid-structure coupling problem is studied considering low velocity flows. Due to the interdisciplinary of the theme, it is necessary to study three different subjects: the Computational Structural Dynamics (CSD), the Computational Fluid Dynamics (CFD), and the coupling problem. The objective of the work is to analyze the physical phenomenon of the fluid-structure interaction in a frame of a floor with semi-rigid connections, adopting a model compatible with this situation. This model includes all the necessary considerations for a bolt connection, specifically: contacts between plate and nut; bolt head and body; contact between plates, as well as friction between them; restressing on the bolt. The coupled problem is solved using Arbitrary Lagrangian-Eulerian (ALE) approach for the fluid domain, while for the solid domain a Lagrangian approach is used. ANSYS® software was used to model and solve the mathematical equations of the coupling problem. This problem consists of analyzing the dynamic behavior of the structure when it is subjected to the action of the load from random winds.

Keywords: Fluid-structure Interaction, Semi-rigid Connection, Finite Element Method, Coupling Problem.

1 Introduction

Fluid-structure interaction (FSI) problems are very common, being present in the most diverse areas of engineering. It extends to aeronautical, hydraulic engineering and even bioengineering in the simulation of blood circulation in the human body, for example. In civil engineering, we can list the action of the wind on the structures, flows in channels and spillways, reservoirs, water interacting with dams and *off-shore* structures, among others. In all the cases mentioned, it is possible to observe flows with a low number of Mach, which can be considered incompressible.

In cases where there is a fluid in motion, when in contact with the structure, it promotes the alteration of its dynamic behavior, in this case there is a classic problem of the fluid-structure coupled problem. In the last decades, advances in materials science have led to the execution of increasingly slender and flexible structures, capable of developing large displacements without reaching rupture. Such structures, however, can have effects that are not adequately predicted in the design stage when exposed to a fluid medium.

According to POTTER [1], in general, structures develop deformations and finite displacements when subjected to external actions, such as loading or forced displacements in their contour. Therefore, a Lagrangian description of the dynamic balance of a solid is commonly used, having as main unknowns the displacements or nodal positions.

According to SANCHES [2], regarding the problems of fluid-structure interaction, the difficulty lies in how to properly couple two problems whose mathematical descriptions are not the same, which means to couple a fixed computational mesh for the fluid, with a deformable mesh for the structure. To get around this problem, DONEA [3] proposed a arbitrary Lagrangian–Eulerian (ALE) method to describe the fluid's governing equations, thus introducing a reference domain that can move with an arbitrary speed field. This type of formulation has been shown to be efficient only in regimes where the displacements or deformations of the geometry are not excessively large, as can be seen in SOULI [4], since the fluid mesh must maintain coherence during the deformation process.

Currently, the partitioned form is the most used to solve FSI problems, being described in detail in the works of PIPERNO [5], MOK [6], TEIXEIRA [7] and IDELSOHN [8].

The use of the partitioned method to couple the fluid domain with that of the structure offers several advantages. According to PIPERNO [5], this method allows using techniques that have already been proven to be effective in discretizing and solving each of the problems individually, thus ensuring greater modularity. Among the connection means used in practice, the bolt connections stand out for their simplicity and speed in the process of assembling steel structures, both small and large ones. The large number of variables existing in this type of connection - thickness of the plates involved in the connection, diameter and positioning of the bolts, etc. - however, make it difficult analyzing your behavior.

The rigidity of a connection profoundly affects the final behavior of the structure. The soliciting efforts on the bars, displacements and rotations may, depending on the type of connection considered, vary from case to case. For this reason, in addition to the bars that make up the structure, the connections must also be properly designed and dimensioned, otherwise the structure may not behave as desired. This is to say that the degree of rigidity of each joint must be properly considered.

One of the objectives of this work is to develop a finite element model, corresponding to the behavior of a bolt connecting two or three square plates, in order to include all the necessary considerations: contacts between the plates and the nut, the head and the body of the bolt; contact between the plates, as well as friction between them; pre-stressing the bolt.

In this way, the behavior of the fluid-structure interaction will act together with the bolt connections properly, using the finite element method to develop a unitary model (a bolt joining two plates) to later insert it into complete connections, and thus, obtain the deformations in the plates. For this, the finite element method and the aid of the ANSYS® [9] software were used for modeling and solving the problem.

2 Governing Equations

2.1 Newtonian incompressible fluid with a mobile domain

According to BAZILEVS et al. [10], the Navier-Stokes equations that represent the incompressible Newtonian fluid are written in terms of the equations of continuity and the amount of motion. These equations can be written as:

$$\rho\left(\dot{u}+\left(\nabla_{\hat{x}} u\right)(u-\hat{v})-F\right)-\nabla_{\hat{x}} \cdot \sigma=0 \qquad \forall(\hat{x}, t) \in \Omega \times D) \tag{1}$$

$$\nabla_{\hat{x}} \cdot u=0 \qquad \forall(\hat{x}, t) \in \Omega \times D \tag{2}$$

Where ρ represents the density of the fluid, F the volume force vector, σ the Cauchy tensor. The time interval of interest is denoted by D = [0, t].

An essential feature of the problems that are addressed in this article is the movement of the fluid boundary in contact with the flexible solid. The geometry of the fluid domain can change significantly during the time domain of interest. Therefore, it is convenient to formulate the problem in the ALE approach, where conservation laws are expressed considering this movement of the border. Thus, the time derivative of velocity u is described as:

$$\frac{\mathrm{d} u}{\mathrm{d} t}=\nabla_{\hat{x}} u(u-\hat{v})+\dot{u} \tag{3}$$

Where $\hat{v}=\partial \hat{x} / \partial t$ is the velocity at that fluid-structure iteration point. The operator $\nabla_{\hat{x}}$ denotes the derivatives in relation to the referential coordinate $\hat{x}$ current. The expression $\dot{u}$ corresponds to the change in particle velocities, observed by an observer traveling with a point in the reference system. The $u - \hat{v}$ speed difference is called the relative speed [10].

For fluid boundary conditions, the boundaries Γ of Ω can be divided into subsets Γ_q, Γ_g and Γ_{fs}, where q, g indices represent, respectively, the boundary at the entrance and exit of the fluid domain. The f - s index represents the fluid boundary in contact with the structure[10]. Boundary conditions can be prescribed in these subsets, as follows:

$$u = q \qquad \forall(\hat{x}, t) \in \Gamma_q \times D \tag{4}$$

$$\sigma\hat{n} = g \qquad \forall(\hat{x}, t) \in \Gamma_g \times D \tag{5}$$

$$u =' d \qquad \forall(\hat{x}, t) \in \Gamma_{f-s} \times D \tag{6}$$

$$(u - \hat{v}) \cdot \hat{n} \qquad \forall(\hat{x}, t) \in \Gamma_{f-s} \times D \tag{7}$$

$$p_f + p_s = \sigma\hat{n} + p_s = 0 \qquad \forall(\hat{x}, t) \in \Gamma_{f-s} \times D \tag{8}$$

The values of q and g are prescribed and represent, respectively, the velocity of the fluid at the entrance and the pressure of the fluid at the exit of the domain through the respective boundary. The boundary condition in the fluid interface structure Γ_{f-s} is shown in eq. (6), and means that there is a non-slip condition there. Also at the Γ_{fs} frontier, there is a need to satisfy the condition prescribed in eq. (7), which means that the Γ_{fs} frontier of the fluid with the structure must coincide with the contour of the deformed structure, for each time step. The pressure balance along the fluid-structure interface is expressed by eq. (8), where the values p_s and p_f represent the pressure vectors exerted by the fluid at the interface with the flexible structure [10].

2.2 Structural dynamics

The conservation of energy in a continuous solid can be expressed in its spatial condition as follows:

$$\rho(''d - F) - \nabla \cdot \sigma = 0 \tag{9}$$

Where ρ is the density of the deformed solid, the vector ”d represents the displacement of the structure, while the body forces are given by the vector F. The Cauchy tensor here is also represented by σ. For simplification, this work deals with a structure with linear elastic behavior [10].

As in the fluid domain boundary, the structure outline can be subdivided into three subsets Γ_q, Γ_g and Γ_{fs}, with their boundary conditions being those that follow:

$$d = q \qquad \forall(\hat{x}, t) \in \Gamma_q \times D \tag{10}$$

$$\sigma n = g \qquad \forall(\hat{x}, t) \in \Gamma_q \times D \tag{11}$$

$$"d = u \qquad \forall(\hat{x}, t) \in \Gamma_q \times D \tag{12}$$

$$p_f + p_s = \sigma\hat{n} + p_s = 0 \qquad \forall(\hat{x}, t) \in \Gamma_{f-s} \times D \tag{13}$$

The values q, g and n are prescribed and mean, respectively, the displacement, the traction vector and the unitary vector normal to the surface of the structure boundary. The conditions of eq. (12) and eq. (13), clearly come according to the conditions of eq. (6) and eq. (7) of the fluid, respectively [10].

3 Methodology

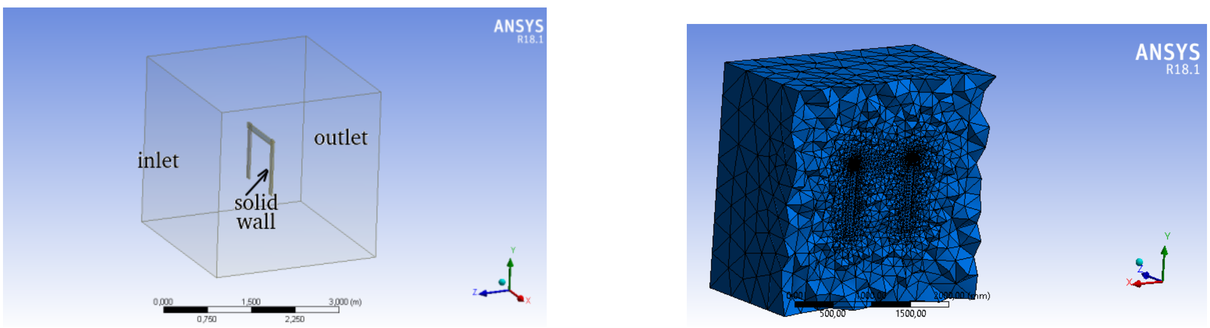
In the fluid environment, the CFX module is used, and the equations are developed according to Section 2 and BLAZEK [11]. In order to analyze the physical phenomenon involved in the fluid-structure interaction, the numerical methods used to solve the problem will not be discussed here.

In a first step, the governing equations for the fluid are solved, and then the pressure value is transferred at the fluid-solid interface. In this way, the bolt connection is modeled, with the respective contact and preload regions. Such pressures generate a certain displacement in the structure, being one of the results of the present work.

A 3000 mm cube on the side around the frame to represent the fluid domain is shown in Fig. 1(a). To determine these values, a mesh with 93,029 finite elements and 16,356 nodes was assembled, as shown in Fig. 1(b). It is noticed that the mesh is more accurate close to the frame, as it is the region of greatest interest in the case analyzed. The greater the number of elements to discretize the fluid's contact region with the frame, the more accurate the result will be.

For the boundary conditions of the fluid, we have the boundary conditions of the inlet, outlet, and no-slip for the solid walls. The other surfaces of the control volume, upper, bottom, and laterals were enforced the slip boundary condition, BLAZEK [11]. The inlet has normal speed equal to 50 m/s and turbulence medium with intensity equal do 5%. The outlet has relative pressure equal to 0 Pa.

All 93029 elements are tetrahedral. According to ANSYS [12], trilinear interpolation functions were used for pressure and velocity.



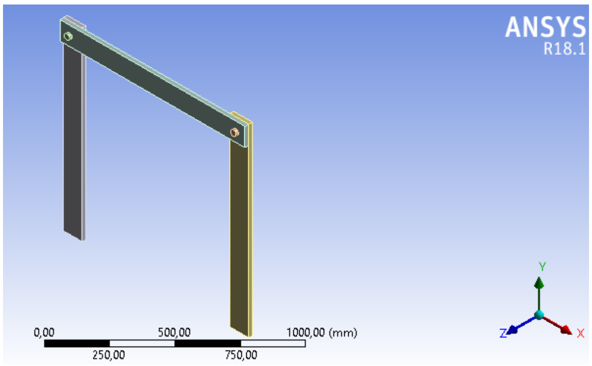
(a) (b)

Figure 1. a) Control volume for the numerical solution; b) Mesh generated for the fluid domain.

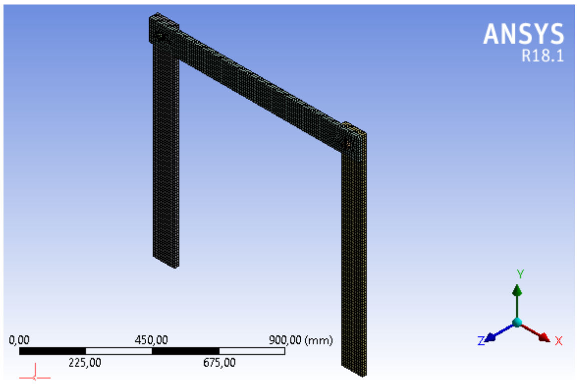
To verify the behavior of a structure submitted to the action of the wind, a steel structure was chosen as shown in Fig. 2(a). The portico geometry used in the simulation has three 1000 mm x 100 mm x 20 mm metal plates. In addition, the frame has fixed support at its bottom. The bolts are 20 mm in diameter. It was used as Structural Steel material, software default. A 50 N bolt pretension was added to the bolt.

Four finite elements are used to discretize the structure, giving the problem its own characteristics. A 20-node 3D solid element that exhibits quadratic displacement behavior is used, has three degrees of freedom per node: translations in the x, y and z nodal directions. Another element used is to represent several 3D "target" surfaces for the contact element. It is a shell element located on the surface of the solid. There is also an element that has the objective of defining a 3D pre-tension section within a mesh structure, has 3 nodes and a degree of freedom of translation. Finally, an element is used to represent the contact and sliding between the 3D target surfaces and a deformable surface defined by this element. The element is defined by eight nodes and is located on the surface of the 3D solid element. The Frictional Model used takes into account Coulomb's Law, in which two contact surfaces can carry shear stresses [13]. For both bolts, frictional contact was considered with a frictional coefficient of 0.2 in the contacts: bolt to late, plate to bolt and plate to nut. The bolded contact was considered for bolt to the nut.

For the simulation performed in this work, the frame finite element mesh is shown in Fig. 2(b), and has 6781 finite elements and 37973 nodes.



(a)

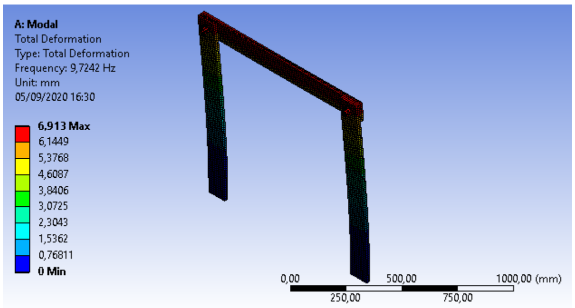


(b)

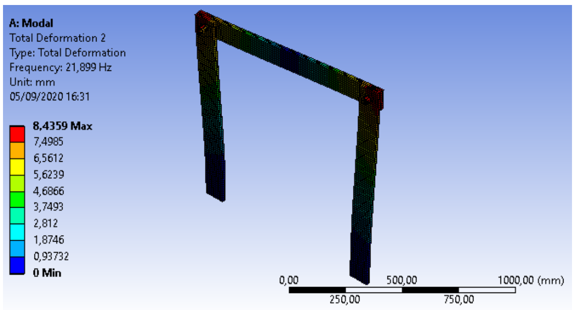
Figure 2. a) Frame geometry; b) Frame mesh.

4 Results

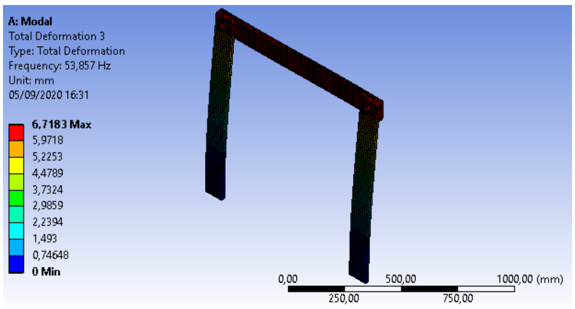
Initially, modal analysis was performed on the bolted frame. The first 3 natural frequencies are, in Hertz: 9.7242, 21.899 and 53.857, as shown in the Fig. 3.



(a)



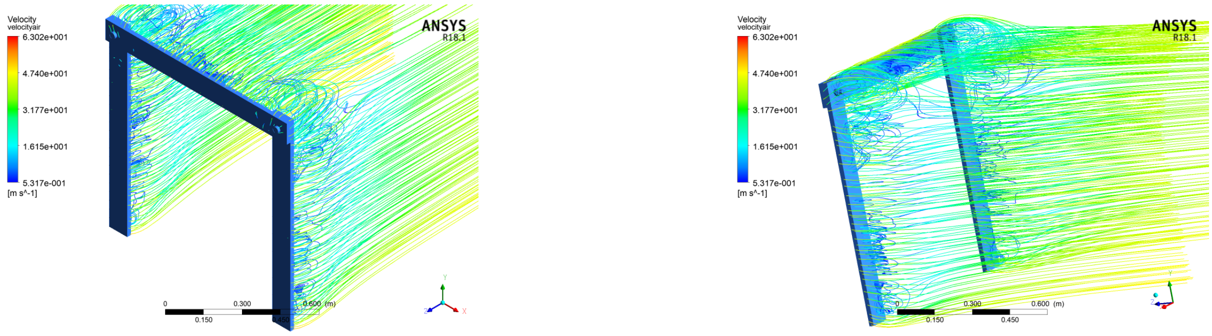
(b)



(c)

Figure 3. Frame vibration modes a) Mode 1; b) Mode 2; c) Mode 3.

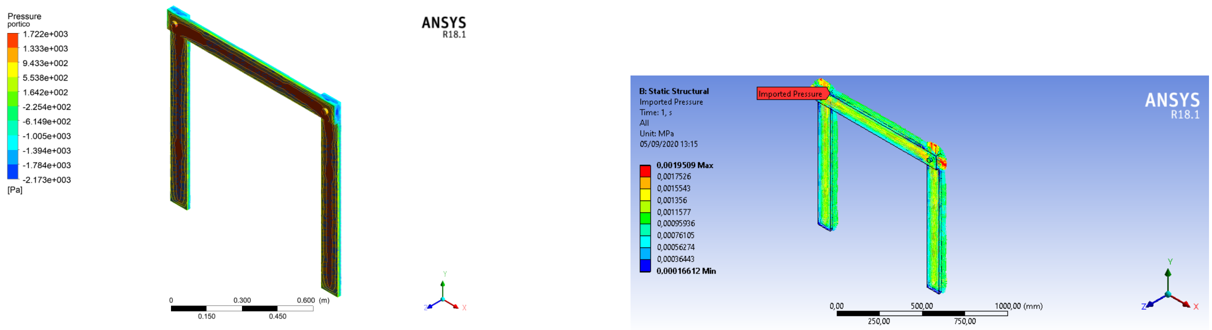
The air flow is constant at the entrance. The behavior and speed of the gust of wind exiting the frame are shown in Fig. 4.



(a) (b)

Figure 4. a) Front view of the frame; b) Posterior view of the frame.

Still as a result of fluid-dynamics, we have the pressures that will be transferred to the structure, as shown in Fig. 5(a). In this way, these pressures are imported into the static part as shown in Fig. 5(b).



(a) (b)

Figure 5. a) Stresses caused by the fluid; b) Imported pressure.

Thus, we have the total deformation of the bolted frame after the 50 m/s wind gust, as shown in Fig. 6.

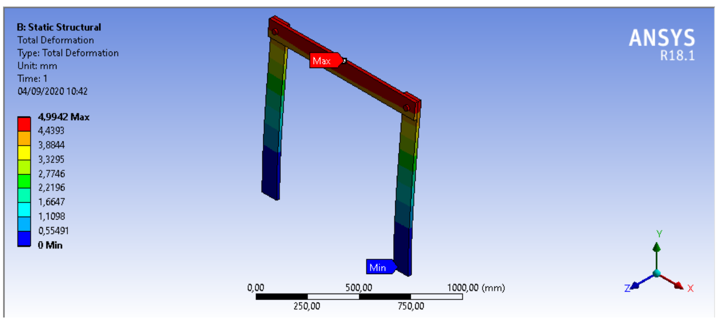


Figure 6. Total deformation

5 Conclusions

In this present work, a study was presented to assist in the understanding of fluid-structure coupling problems. These results are partial and help to understand how the coupling phenomenon happens. The values of the first three modes of vibration and natural frequencies were verified, considering a semi-rigid frame. Following the results found in this article, the results can be validated by performing the analysis experimentally. The main purpose of using semi-rigid connections in metal structures is to redistribute the bending moments in beams and columns and, consequently, to reduce steel consumption.

The treatment of the connections in their real behavior, that is, as semi-rigid, considering problems such as stability of the structure (effective length, buckling, etc.) and lamination of the bars, allows to obtain, at the end of the project, a lighter structure and in lower cost, without loss of resistance. This leads to more optimized designs and with the same level of security to collapse as designs with connections treated as rigid or flexible.

It is important to note that the values can increase the accuracy using a finer mesh. However, greater refinement of the meshes will require more computational effort. The results presented are partial and are part of a study about the fluid-structure coupling. The study of the dynamic properties of semi-rigid connections is of extreme importance to knowledge due to its complexity in relation to welded connections and, as discussed, because they are a more economical type of connection to buildings.

Acknowledgements. The first author was financed in part by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior - Brasil (CAPES) -Finance Code 001. The authors would like to acknowledge Fundação Araucária de Apoio ao Desenvolvimento Científico e Tecnológico, which provide a financial support under Grant 376/2014.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] POTTER, M. C.; WIGGERT, D. C., 2009. *Mecânica dos Fluidos*. 3. ed. São Paulo. Thomson.

[2] SANCHES, R. A. K., 2011. *Sobre o acoplamento fluido-casca utilizando o método dos elementos finitos*. PhD thesis, SET-EESC-USP, São Carlos, São Paulo, Brasil.

[3] DONEA, J.; GIULIANI, S. H. J. P., 1982. An arbitrary lagrangian-eulerian finite element method for transient dynamic fluid-structure interactions. *Computer Methods in Applied Mechanics and Engineering*, vol. 33, pp. 689–723.

[4] SOULI, M.; OUAHSINE, A. L. L., 2000. Ale formulation for fluid-structure interaction problems. *Computer Methods in Applied Mechanics and Engineering*, vol. 190, pp. 659–675.

[5] PIPERNO, S.; FARHAT, C. L. B., 1995. Partitioned procedures for the transient solution of coupled aroelastic problems .1. model problem, theory and 2-dimensional application. *Computer Methods in Applied Mechanics and Engineering*, vol. 124, pp. 79–112.

[6] MOK, D. P.; WALL, W. A., 2001. Partitioned analysis schemes for the transient interaction of incompressible flows and nonlinear flexible structures. *Trends in Computational Structural Mechanics*, pp. 689–698.

[7] TEIXEIRA, P. R. F. E.; AWRUCH, A. M., 2005. Numerical simulation of fluid-structure interaction using the finite element method. *Computers and Fluids*, vol. 34, pp. 249–273.

[8] IDELSOHN, S.; OÑATE, E. D. P. F. C. N., 2006. Fluid-structure interaction using the particle finite element method. *Computer Methods in Applied Mechanics and Engineering*, vol. 195, pp. 2100–2123.

[9] ANSYS®, 2017. Academic Research Mechanical and CFD. 18.2. Licence Pak Universidade Tecnologica Federal do Parana.

[10] BAZILEVS, Y., TAKIZAWA, K., & TEZDUYAR, T., 2013. *Computational Fluid-Structure Interaction: Methods and Applications*. Wiley Series in Computational Mechanics. Wiley.

[11] BLAZEK, J., 2015. *Computational Fluid Dynamics: Principles and Applications*. Oxford, UK. Butterworth-Heinemann; 3rd Edition.

[12] ANSYS, I., 2013. *ANSYS CFX Theory Guide*.

[13] CHANDRUPATLA, T., 2004. *Finite Element Analysis For Engineering & Tech.* Orient BlackSwan.