# A COUPLED BOUNDARY AND FINITE ELEMENT METHODOLOGY FOR SOLVING FLUID-STRUCTURE INTERACTION PROBLEMS

**Manuel Barcelos, manuelbarcelos@unb.br**
**Carla T. M. Anflor, anflor@unb.br**
Faculdade do Gama, Campus da Universidade de Brasília no Gama
PO Box 8114, 72.405-610, Gama – DF – Brasil

**Luciano Gonçalves Noleto, lucianonoleto@unb.br**
**Éder L. Albuquerque, eder@unb.br**
Universidade de Brasília, Faculdade de Tecnologia, Departamento de Engenharia Mecânica
Campus Universitário Darcy Ribeiro, 70.910-900, Brasília – Distrito Federal – Brasil

***Abstract.*** *This work aims the development of an efficient and robust numerical methodology to study the mechanical behavior of structures under the influence of external flow fields. In order to achieve this goal, it is necessary to simplify and make reliable the exchange of information between two numerical domains. Therefore, two efficient and robust numerical methodologies are coupled over matching meshes to guarantee the quality of the exchanging process. Thus, the external flow field is solved by using a viscous laminar equation model and a finite element method (FEM), while the airfoil structure is solved by using an elastic equation model and a boundary element method (BEM). The coupling between both numerical techniques allow for the simulation of the fluid flow over the airfoil as well as its structural behavior such as a realistic fluid-structure interaction problem. A NACA0012 airfoil with a specific set of mechanical properties and free stream flow configurations is analyzed to illustrate the FSI framework.*

***Keywords****: Boundary Element Method, Fluid-Structure Interaction Problem, Finite Element Method, Aeroelasticity*

## 1. INTRODUCTION

The progress we have experienced in experimental and numerical analysis of fluid-structural interaction (FSI) has not been enough to become it a less complex problem. This is mainly due to the physical phenomenon of interaction of fluid-structure that generates dynamic structural behavior of high complexity, giving highly non-linear responses. Therefore, FSI is an open issue and a research field very active today. On the other hand, the increase in processing power of computer systems in last decade and the use of computer models are presented today as an attractive alternative to analytical and experimental approaches traditionally used for analysis of problems of fluid-structure interaction, FSI in the naval, automotive and aerospace industries. However, a major disadvantage of the approach is its high computational cost and storage requirements, mainly. This is essentially happens because most currently used formulations are based on discretization methods of the whole domain, both for the structural problem and the flow problem.

The computational analysis of FSI represents a fundamental tool in the design of ship and aircraft structures. The increase in recent years of computational processing power has allowed the use of high fidelity computer models. However, computational cost is still important on aeroelastic and hydroelastic problems because fluid domains are, in general, too large. The numerical development of new formulations that reduce the computational cost for aeroelastic and hydroelatic analyses represents one of the most active research areas in engineering, today. These new formulations can reduce the computational resources necessary for its implementation or to increase the accuracy of analysis.

The FSI formulation applied in this work deals with fluid and structure as separate domains. This formulation is called staggered or partitioned scheme, allowing for the fluid and the structures problems to be solved by different numerical methodologies. The shortcoming of this approach is the flux of information over the fluid-structure boundary. For simplicity and robustness, in this work matching meshes domains are chosen. Thus, data is transferred directly through the meshes without the necessity of projection or interpolation procedures. The main advantage of the staggered approach is simplicity and flexibility. Different numerical methods can be used to solve specific problems and few modifications have to be done to previous numerical schemes to adapt them to a FSI framework.

## 2. FLUID-STRUCTURE ANALYSIS

In a general fluid-structure framework, the governing equations are based on the three field formulation (Farhat et al., 2003), as described:

$$F(\mathbf{u}, p, \boldsymbol{\chi}) = 0 \tag{1}$$

$$M(\boldsymbol{\chi}, \mathbf{d}) = 0 \tag{2}$$

$$S(\mathbf{d}, \mathbf{u}, p) = 0 \tag{3}$$

where $F$ is the state equation of the fluid, $M$ is the equation which governs the motion of the fluid mesh and $S$ is the state equation of the structure. The following state variables: $\mathbf{u}$, $p$, $\boldsymbol{\chi}$ and $\mathbf{d}$ represent, respectively, the fluid velocity field, the fluid pressure field, the fluid mesh displacement and the structural displacements. As most of the design requirements are effectively computed by using stead state responses, a quasi-static solution methodology is proposed to solve the coupled problem. Therefore, in order to employ a quasi-static solution methodology some terms related to time derivatives are neglected.

Equations (1) to (3) are coupled through the transmission conditions on the fluid-structure interface $\Gamma_{fs}$, which represent the equilibrium of forces, equation (4), the compatibility of the displacement, equation (5), and the velocity, equation (6), between the fluid and the structure domains,

$$\sigma_S \cdot \mathbf{n} = \tau \cdot \mathbf{n} - p \cdot \mathbf{n} \text{ on } \Gamma_{fs} \tag{4}$$

$$\boldsymbol{\chi} = \mathbf{d} \text{ on } \Gamma_{fs} \tag{5}$$

$$\frac{\partial \boldsymbol{\chi}}{\partial t} = \frac{\partial \mathbf{d}}{\partial t} \text{ on } \Gamma_{fs} \tag{6}$$

where $p$ and $\tau$ are, respectively, the flow pressure and shear stress tensor, $\sigma_s$ is the structure stress tensor and $\mathbf{n}$ is the normal at a point on $\Gamma_{fs}$.

## 3. FLOW PROBLEM

By considering a time dependent viscous laminar and incompressible flow problem, the flow governing equations $F$ are described as:

$$\nabla.\mathbf{u} = 0 \tag{6}$$

$$\frac{\partial^2 \mathbf{u}}{\partial t} + (\nabla \mathbf{u})\hat{\mathbf{u}} = -\frac{1}{\rho_f}\nabla p + \nabla.(\nu_f \nabla \mathbf{u}) + \mathbf{f}_f \tag{7}$$

where $\mathbf{u}(\mathbf{x},t)$ e $p(\mathbf{x},t)$, respectively, the velocity and the pressure fields of the flow are function of the mesh position and the time, $\rho_f$ and $\nu_f$ are in this order the density and the viscosity of the fluid problem and $\mathbf{f}_f$ is a given force function. Due to the moving boundary problem the continuity and momentum equations are modified to satisfy the arbitrary Lagrangian-Eulerian (ALE) formulation. Therefore, the relative velocity term $\hat{\mathbf{u}}$ is introduced and represents the difference between flow and mesh velocities. The boundary conditions are defined on $\Gamma_f = \Gamma_d \bigcup \Gamma_{fs} \bigcup \Gamma_o$ by:

$$\mathbf{u}(\mathbf{x},t) = \mathbf{u}_{\Gamma_d} \text{ on } \Gamma_d \tag{8}$$

$$\mathbf{u}(\mathbf{x},t) = \mathbf{u}_{\Gamma_{fs}} \text{ on } \Gamma_{fs} \tag{9}$$

$$p(\mathbf{x},t) = p_{ref} \text{ on } \Gamma_o \tag{10}$$

The subset $\Gamma_d$ represents the boundary values that are constant where the Dirichlet boundary condition for the velocity field is prescribed, $\mathbf{u}_{\Gamma_d}$. $\Gamma_{fs}$ is the moving boundary, where the fluid velocity is equivalent to the domain velocity, $\mathbf{u}_{\Gamma_{fs}}$. $\Gamma_o$ is the outlet boundary condition where a reference pressure is prescribed, $p_{ref}$.

The flow solution methodology is based on the semi-explicit iterative solution of the systems of equations (6) and (7) after a time and a spatial discretization, and considering a projection method framework (Goldberg and Ruas, 1999, Guermond et al., 2006, Lohner et al., 2006).

For a given $\Delta t > 0$ and considering the set of variables $\mathbf{u}^n$, $p^n$ and $\mathbf{x}^n$ known from the previous time step $t$. The solution $\mathbf{u}^{n+1}$, $p^{n+1}$ and $\mathbf{x}^{n+1}$ at the time $t+\Delta t$ is computed by using a staggered approach such as:

$$\frac{1}{\Delta t}\left(\mathbf{u}^* - \mathbf{u}^n\right) = -\frac{1}{\rho_f}\nabla p^n + \nabla.\left(\nu_f^n \nabla \mathbf{u}^n\right) + \mathbf{f}_f \tag{11}$$

$$\frac{1}{\Delta t}\left(\mathbf{u}^{n+1} - \mathbf{u}^*\right) = -\frac{1}{\rho_f}\nabla\left(p^{n+1} - p^n\right) \tag{12}$$

$$\nabla.\mathbf{u}^{n+1} = 0 \tag{13}$$

The fractional step method introduces a predicted velocity $\mathbf{u}^*$ which is corrected at the end of the block. Also, taking the divergence of the Eq. (12) and using Eq. (11) the Poisson equation is obtained:

$$\nabla^2\left(p^{n+1} - p^n\right) = \frac{\rho_f}{\Delta t}\nabla.\mathbf{u}^* \tag{13}$$

and the boundary condition for Eq. (13) is:

$$\nabla^2\left(p^{n+1} - p^n\right).\mathbf{n} = \frac{\rho_f}{\Delta t}\nabla.\mathbf{u}^*.\mathbf{n} \quad \text{on } \Gamma_f \tag{14}$$

The projection method here described is called Incremental Projection Scheme. This method is a modification of the version proposed by Chorin (1968) which improves convergence as reported in the literature (Codina and Blasco, 2000, Codina, 2000, Guermond et al., 2006).

By considering the dimension of the finite element space equals to $N$ and defining the base function as $\{\mathbf{N}_i : i = 1, \ldots, N\}$ and $\{\mathbf{N}_j : j = 1, \ldots, N\}$. The matrix form of the discrete finite element problem is:

**Step 1**: Predict velocity through the Momentum equation

$$\mathbf{M}\,\Delta\mathbf{u}^* = \mathbf{F}_{\mathbf{u}}^*\left(\mathbf{u}^n, p^n, \mathbf{x}^n\right) \tag{15}$$

**Step 2**: Poisson problem

$$\mathbf{A}\,\Delta p^{n+1} = \mathbf{F}_p^*\left(\mathbf{u}^*, \mathbf{x}^n\right) \tag{16}$$

**Step 3**: Velocity correction – projection on the divergence free space

$$\mathbf{M}\,\Delta\mathbf{u}^{n+1} = \mathbf{F}_{\mathbf{u}}\left(p^{n+1}, \mathbf{x}^n\right) \tag{17}$$

For equations (15) to (17), $\mathbf{M}$ and $\mathbf{A}$ are the mass and the Laplacian matrices which are given by:

$$\mathbf{M}_{ij} = \frac{1}{\Delta t}\left(\mathbf{N}_i, \mathbf{N}_j\right) \tag{18}$$

$$\mathbf{A}_{ij} = \left(\nabla\mathbf{N}_i, \nabla\mathbf{N}_j\right) \tag{19}$$

The vectors $\mathbf{F}_{\mathbf{u}}^*$, $\mathbf{F}_p$ and $\mathbf{F}_{\mathbf{u}}$ are related to the discretization of the right-hand side of the equations through steps 1 to 3, and the boundary integral terms referent to the boundary conditions are also included to these vectors.

The greatest advantage of the presented numerical scheme at the steps 1 to 3 is the mass matrix. In order to enhance convergence and time efficiency the mass matrix is lumped in a diagonal form, and its construction is performed only if the mesh position is updated. Once a flow state $\mathbf{u}^{n+1}$ and $p^{n+1}$ is computed, the fluid forces acting over the structure is determined by:

$$f_f^{n+1} = \tau^{n+1} \cdot \mathbf{n} - p^{n+1} \cdot \mathbf{n} \text{ on } \Gamma_{fs} \tag{20}$$

## 4. MESH MOTION PROBLEM

The governing equations of the motion of the fluid mesh $M$ are built considering the fluid mesh as a pseudo-elastic structure (Batina, 1991, Koobus et al., 1998, Degand and Farhat, 2002):

$$\bar{\mathbf{M}}\ddot{\chi} + \bar{\mathbf{C}}\dot{\chi} + \bar{\mathbf{K}}\chi = \bar{\mathbf{R}} \tag{21}$$

where the vector of mesh displacement $\chi(\mathbf{x},t)$ is a function of the mesh position and the time. $\bar{\mathbf{M}}$, $\bar{\mathbf{C}}$ and $\bar{\mathbf{K}}$ are respectively the fictitious mass, damping and stiffness matrices associated to the fluid grid and $\bar{\mathbf{R}}$ is the reaction force. For a quasi-static model the fictitious mass $\bar{\mathbf{M}}$ and damping $\bar{\mathbf{C}}$ matrices are neglected (Farhat et al., 1998), resulting in

$$\bar{\mathbf{K}}\chi = \bar{\mathbf{R}} \tag{22}$$

The quasi-static approach, due to its simplified structure and solution strategy, is usually preferred to model fluid-structure interaction problems. The quasi-static fluid mesh motion equations obey the kinematic compatibility between fluid and structure. The kinematic compatibility dictates how the position $\chi_{\Gamma_{fs}}$ of fluid mesh on the boundary $\Gamma$ is related to the fluid mesh boundary velocity $\mathbf{u}_{\Gamma_{fs}}^{n+1}$ by:

$$\chi_{\Gamma_{fs}} = \Delta t \, \mathbf{u}_{\Gamma_{fs}}^{n+1} \tag{23}$$

where $\Delta t$ is fluid problem time step. For this problem the fluid mesh boundary points follow the displacement field of the structure. Therefore, the fluid boundary moves as if there was a boundary velocity:

$$\mathbf{u}_{\Gamma_{fs}}^{n+1} = \frac{\mathbf{d}_{\Gamma_{fs}}^{n+1}}{\Delta t} \tag{24}$$

The fictitious stiffness matrix $\bar{\mathbf{K}}$ and the reaction force vector $\bar{\mathbf{R}}$ in equation can be divided in subsets related to the internal ($\Gamma$) and external ($\Omega$) degrees of freedom of the fluid mesh.

$$\begin{bmatrix} \bar{\mathbf{K}}_{\Omega_{fs}\Omega_{fs}} & \bar{\mathbf{K}}_{\Omega_{fs}\Gamma_{fs}} \\ \bar{\mathbf{K}}_{\Omega_{fs}\Gamma_{fs}} & \bar{\mathbf{K}}_{\Gamma_{fs}\Gamma_{fs}} \end{bmatrix} \begin{bmatrix} \chi_{\Omega_{fs}} \\ \chi_{\Gamma_{fs}} \end{bmatrix} = \begin{bmatrix} \bar{\mathbf{R}}_{\Omega_{fs}} \\ \bar{\mathbf{R}}_{\Gamma_{fs}} \end{bmatrix} \tag{25}$$

Equation (25) is solved with the condition of null force on the internal grid points, $\bar{\mathbf{R}}_{\Omega_{fs}} = 0$. Also, in this work for the fluid mesh motion problem, a quasi-static model is used as described in equation (22), and the fictitious stiffness matrix $\bar{\mathbf{K}}$ is constructed by an improved spring analogy method (Koobus et al., 1998, Farhat et al., 1998, Degand and Farhat, 2002). As the main goal is to deal with fluid-structure interaction problems of small to medium scale, the solution strategy employs a direct method to solve the mesh motion linear system of equations. The direct method is preferred because of its simplicity and robustness. The mesh position $\mathbf{x}^{n+1}$ is updated by summing the mesh displacement vector to the previous configuration of the mesh position vector $\mathbf{x}^n$, such as follows:

$$\mathbf{x}^{n+1} = \mathbf{x}^n + \begin{bmatrix} \boldsymbol{\chi}_{\Omega_{fs}} \\ \boldsymbol{\chi}_{\Gamma_{fs}} \end{bmatrix} \tag{26}$$

## 5. STRUCTURE PROBLEM

Now consider an elastic plate of thickness $h$ occupying the area $\Omega_S$, bounded by the contour $\Gamma_S$ , in the $x_1x_2$ plane. The governing equations for the structure quasi-static response, considering plane stress conditions, are given by [8]:

$$Gd_{i,jj} + \frac{G}{1-2\nu} d_{i,ji} = -f_i \tag{27}$$

where $G$ is the shear modulus, $\nu$ is the Poisson ratio and $d_i$ represents in-plane displacements and $f_i$ in-plane body forces. The boundary integral formulation for equation (27) is given by:

$$c_{ij}d + \int_{\Gamma_s} T_{ij}^* d_j \, d\Gamma_S = \int_{\Gamma_s} D_{ij}^* l_j \, d\Gamma_S + \frac{1}{h}\int_{\Omega_s} D_{ij}^* f_j \, d\Omega_S \tag{28}$$

where $T_{ij}^*$ and $D_{ij}^*$ are the fundamental solutions for plane stress elasticity, respectively [8], $l_j$ is the load at the boundary and $c_{ij}$ is a constant that depends on the geometry at the collocation point. Applying the boundary element method and considering discontinuous quadratic elements to discretize the boundary of the airfoil, the following equation is obtained:

$$\mathbf{Hd}^{n+1} = \mathbf{Gl}^{n+1} + \mathbf{Bf}^{n+1} \tag{29}$$

In this work, a matching mesh problem approach is chosen to illustrate the fluid-structure coupling concept. Therefore, the exchange of force and displacement over the fluid and structure boundaries is executed through every pair of matched points between the two mesh domains, without the need of projection and interpolation procedures. The exchange of force and displacement over matching meshes is represented by:

$$\mathbf{l}^{n+1} = \mathbf{I}\mathbf{f}_{fs}^{n+1} \text{ on } \Gamma_{fs} \tag{30}$$

$$\boldsymbol{\chi}_{fs}^{n+1} = \mathbf{I}\,\mathbf{d}^{n+1} \text{ on } \Gamma_{fs} \tag{31}$$

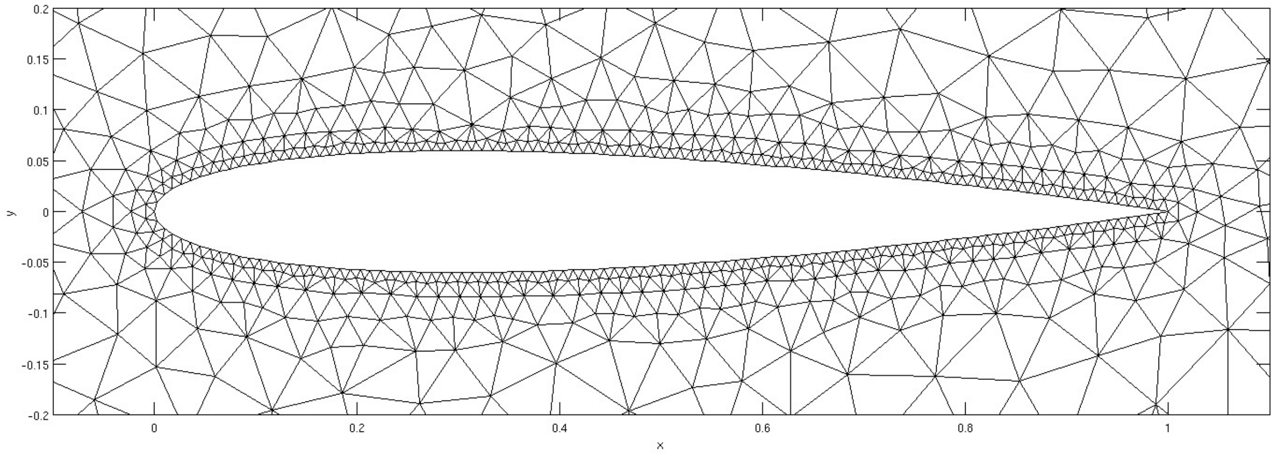
where $\mathbf{I}$ is the identity matrix.

## 6. RESULTS

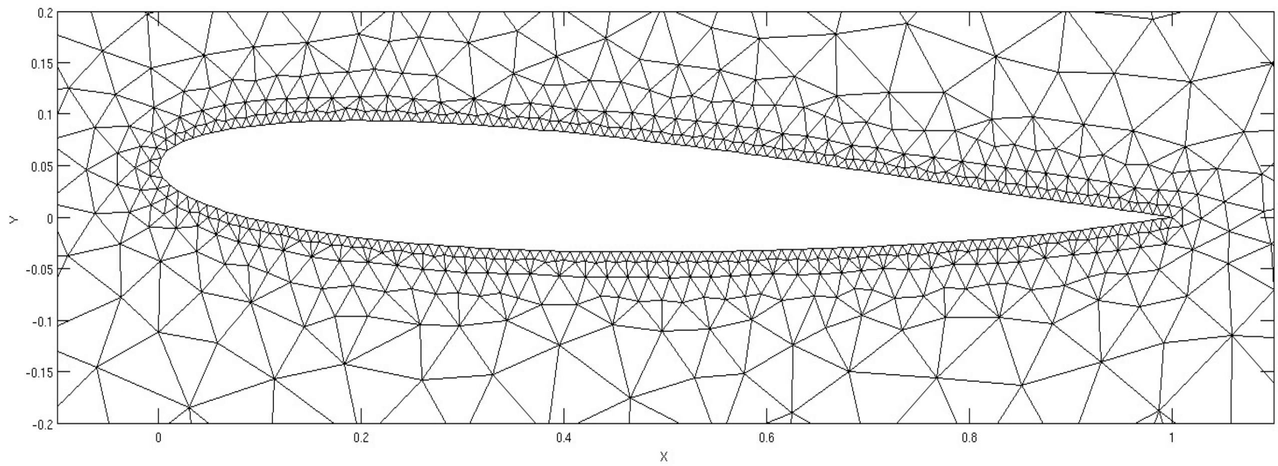
The test case studied is a NACA0012 airfoil of 1 m of chord. The fluid problem is solved by a finite element code for laminar and incompressible flows. The finite element mesh has 5,712 nodes and is composed by 10,952 triangular elements. The mesh region close to the boundary layer has a high ratio of refinement. As a first step, a preliminary flow solution is computed in order to use it for starting point to solve the coupled problem. Thus, a flow air with Reynolds number of 1,000, angle of attack of 5 degrees and free stream kinematic viscosity of 1.33 $10^{-5}$ $m^2/s$ and density of 1.293 $kg/m^3$ was set to run with a time step of 0.01s and a maximum number of flow solution iterations of 10,000. A preliminary steady state flow solution was obtained with zero velocity boundary condition on the airfoil and a velocity error target of $10^{-4}$.

In the second step, the coupled problem was set to start from the preliminary steady state solution and the flow properties remained the same. The structure problem is solved by a boundary element method with a mesh of 204 nodes over the airfoil contour line. As the resulting boundary force computed from the flow solution has a small magnitude, the structure properties were chosen to make it flexible when under the influence of this type of load. So, pseudo material was defined with an elasticity modulus of 700Pa and a Poisson ration of 0.3 to satisfy these conditions. In order to have a smooth solution, the structure code employs a numerical integration scheme with 10 Gauss points over each element. For structural boundary condition, the last 20 points on the airfoil trailing, 10 on the upper surface and 10 on the lower surface, were clamped.

The staggered solver was set to run for 10 coupled problem iterations to allow it to converge to the most reasonable fluid-structure response and for 1000 fluid iterations in order to have a converged steady state flow solution (with a velocity error target of $10^{-6}$) at each coupled problem iteration. Despite the flow problem is running a time dependent solver to determine a steady state solution, the fluid mesh motion and the structure problems are running in a quasi-static mode to simplify the whole FSI modeling. The exchange of force and displacement data over the fluid-structure boundary is done directly through the meshes, because the fluid and the structure domains have matching meshes.



(a)

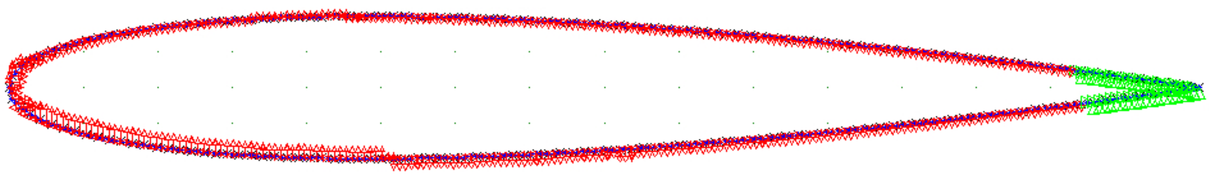


(b)

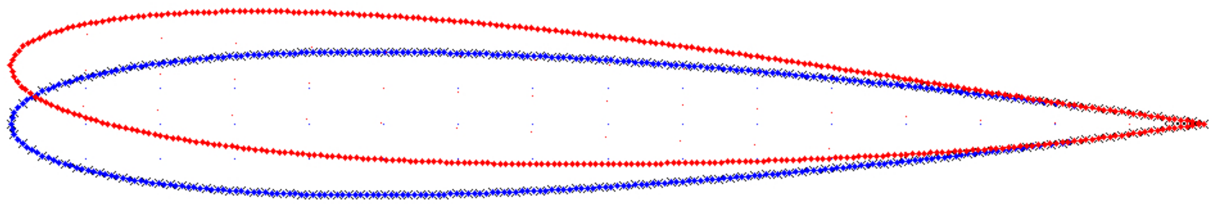
Figure 1: Fluid mesh: original configuration (a) and deformed configuration (b).

The steady state flow solution reached a lift, $C_L$, and drag, $C_D$, coefficients of $0.031$ and $0.011$, respectively. The maximum displacement happened at the leading edge, about 0.044 m. Figures (1) and (2) show the fluid and structure meshes before and after the FSI solution strategy. One can remark that the resulting lift force deformed the airfoil as if was a clamped beam. This behavior happened as expected, because an airfoil under the flow influence is a device that generates lift and drag forces. If there are forces over a structure, then its most natural mechanical response is to deform. This deformation changes the flow path, and after a set of cyclic interactions, a steady state is reached and a final deformed state is defined.

The main goal of this work was achieved, because a numerical FSI framework was built and tested with success. This is the proof that is possible to improve the simulation fidelity with a few set of steps and reorganizing preexistent numerical schemes. Now, by the application of isolated numerical schemes is possible to tackle more complex coupling problems.



(a)



(b)

Figure 2: Structure mesh: original configuration with boundary conditions (a) and deformed configuration with internal point displacements (b).

## 7. CONCLUSION

The work here presented has as goal to show the great potential of application of staggered numerical schemes to solve coupled fluid-structure problems. The main advantage of the staggered scheme is its simplicity and flexibility, once the fluid and structure problems are solved in a separate fashion. One can use different methodologies and numerical algorithms to tackle each individual problem. This allow for applying numerical methods with the best characteristics to solve specific fluid-structure coupling problems.

The bottleneck of the staggered approach lies on the fact that is necessary the development of preprocessing tools to project, interpolate and transfer force and displacement data over the fluid-structure boundary. The methodology employed in this work avoided this difficulty by using matching meshes that allowed for transferring information between domains directly, without the need of a preprocessing step.

The solution of the coupling problem by employing a finite element method for the fluid domain and a boundary element method for the structure domain was remarkably successful. Few modifications had to be executed to prepare the exchange of information between de codes. Also, as the fluid code had the ALE and mesh motion featured already installed and tested, there is no difficulty of adapting it to fit in a fluid-structure coupling framework.

## 8. REFERENCES

Batina, J. T., 1991. Unsteady Euler algorithm with unstructured dynamic mesh for complex-aircraft aerodynamic analysis. AIAA Journal, vol. 29, n. 3, pp. 327–333.

Chorin, A. J., 1968. Numerical solution of the Navier-Stokes equations. Math. Compt., vol 22, pp. 745-762.

Codina, R., & Blasco, J., 2000. Stabilized finite element method for transient Navier-Stokes equations based on a pressure gradient projection. Comput. Methods Appl. Mech. Engrg., vol 182, pp. 277-300.

Codina, R., 2000. Pressure stability in fractional step finite element methods for incompressible flow. J. for Compt. Physics, vol. 170, pp. 112-140.

Degand, C., & Farhat, C., 2002. A three-dimensional torsional spring analogy method for unstructured dynamic meshes. Computers and Structures, vol. 80, pp. 305–316.

Farhat, C., Degand, C., Koobus, B., & Lesoinne, M., 1998. Torsinal springs for two dimensional dynamic unstructured fluid meshes. Comput. Methods Appl. Mech. Engrg., vol 163, pp. 231-245.

Farhat, C., Geuzaine, P. and Brown, G., 2003. Application of a three-field nonlinear fluid-structure formulation to the prediction of the aeroelastic parameters of an F-16 fighter. Computers and Fluids, vol 32, pp. 3–29.

Goldberg, D., & Ruas, V., 1999. A numerical study of projection algorithms on finite element simulation of three-dimensional viscous incompressible flows. Int. J. Numer. Meth. Fluids, vol 30, pp. 233-256.

Koobus, B., Farhat, C., Degand, C., & Lesoinne, M., 1998. An improved method of spring analogy for dynamic unstructured fluid meshes. In Proceedings of the 39th AIAA/ASME/ASCE/AHS/ASC structures, structural dynamics, and materials conference and exhibit, April 20-23, Long Beach, California.

Lohner, R., Yang, C., Cebral, J., Camelli, F. Soto, O., & Walts, J., 2006. Improving the speed and accuracy of the projection-type incompressible flow solvers. Comput. Methods Appl. Mech. Engrg., vol. 195, 3087-3109.

Brebbia, C. A., & Dominguez, J., 1993. Boundary Elements: An Introductory Course. WIT Press, Southampton.

## 9. RESPONSIBILITY NOTICE

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