A FLUID-STRUCTURE INTERACTION SCHEME USING BOUNDARY FORMULATIONS

Daniel F. M. Möller, danielmoller@gmail.com
Raul B.V. Pessolani, raul@vm.uff.br
Universidade Federal Fluminense - Depto. de Eng. Mecânica
Rua Passos da Patria, 156 – Niterói – Rio de Janerio

Abstract. *This paper presents an implementation of a Fluid-Structure Interaction Scheme using the Panel and Boundary Element Methods with straight panels applied to a bi-dimensional body immersed in a fluid flow. First, a geometric pre-processor is developed to minimize the discretization errors. In addition, the fluid flow is solved using a Panel Method for potential problems which calculates the velocity and pressure over the body surface. Finally an Elastic Boundary Formulation is applied for computing the stresses and the corresponding displacements on the element surface. These displacements cause an alteration of the body geometry so the flow has to be recalculated. This generates another field of pressure that makes for more changes on the surface. After a few iterations convergence is reached, when the displacements are nearly zero. If the solid is supported by elastic springs, the flow pressure can cause a vibration which is calculated by rigid body movement. The main advantage of this fluid-structure technique is that it needs only a refinement of the boundary, and the meshes for both fluid and elastic problems can be the same. This substantially reduces the dimensions of the problem, and simplifies the treatment of data interchange between the two problems. Two examples show the applicability of this method.*

Keywords*: Fluid-Structure Interaction, Boundary Element Method, Vibration.*

1. INTRODUCTION

Fluid-structure interaction has been developed for studying structures submitted to the action of water or wind, such as oil platforms, risers, wings and airplanes. In these situations, different variable forces appear, inducing deformations and vibrations. The fluid-structure analysis can be used to evaluate the stresses and displacements at different situations, preventing eventual further problems.

The solution of this kind of problems depends on the efficiency of the techniques applied for solving the flow and the structure, separately. It also depends on the manipulation of sub domains at the interface between fluid and structure. In the last years, numerical and experimental techniques have been developed for this analysis, using Finite Element Method (FEM) (Liu, 1980). This technique presents some difficulties, mainly on the interface between the meshes of the structure and the fluid, that have to interchange values of displacements and pressures.

On the other hand, the Boundary Element Method (BEM), that began in the 70's with Lachat and Watson (1975) and Brebbia (1978), has the appeal of needing discretization only of the boundary, substantially reducing system dimension and complications due to mesh coupling on the interface. The BEM is specially indicated for infinite domains, like acoustic and flow problems. Also, it has a very low number of elements and unknowns compared to FEM. However, there are few publications on the application of BEM applied to fluid-structure interaction.

The main objective of this paper is the implementing of a procedure for fluid-structure interaction, using the BEM. The flow is solved by a Panel Method with Sources and Vortices applied to an inviscid and incompressible flow, while the structure analysis is performed by a 2D Elasticity formulation of the BEM.

Straight elements are used. To minimize errors caused by the discretization of curved boundaries, a geometric adaptive procedure is applied. It generates new points along the contour, smoothing the angle between two panels.

The flow analysis calculates the velocities on the surface, and then the pressures needed to perform the elastic analysis, which computes the node displacements and redefines the airfoil geometry. Given a new geometry, the process restarts computing the new distribution of pressure and loads, resulting on increments of displacement. The process is repeated until the displacement increments are smaller than a pre-fixed tolerance.

Another case presented in this paper is the vibration of the structure due to flow pressures. In this case the structure is treated as a rigid body while the supports are elastic.

The elastic and the dynamic fluid-structure interactions are tested in two separated examples.

2. THE PANEL METHOD

According to Anderson (2001), the equation that describes the problem of an incompressible, inviscid and irrotational steady flow around a physical body is the Laplace's equation:

$$\nabla^2 \Phi = 0 \quad \text{or} \quad \frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} = 0 \tag{1}$$

Where Φ is the potential function at the point (x, y).

One way to solve this equation is using the Panel Method described by Katz and Plotkin (2002). This method consists in dividing the surface of the body into panels containing singularities. The value of these singularities are calculated applying boundary conditions, which may be Neumann's or Dirichlet's, at each panel.

Figure 1 shows a discretized airfoil, where V_∞ is the free stream velocity, α is the angle of attack and $\vec{n}$ is the vector normal to the surface.

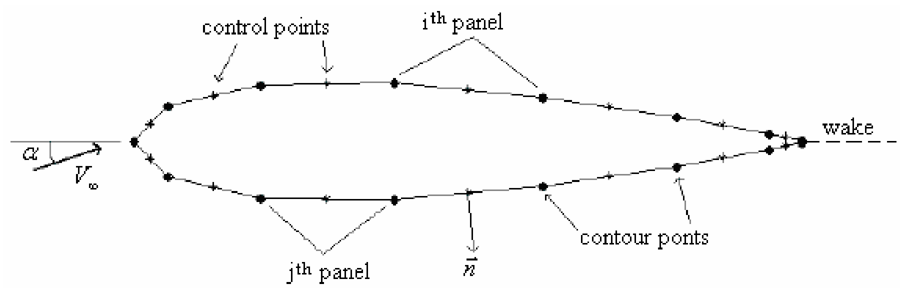

Figure 1. Panels on the surface of an airfoil.

The method uses analytical solutions of the Laplace's equation combining elementary flows (sources, doublets and vortices) at each panel. There are various formulations depending on the type of singularities, the functions of distribution order and the boundary conditions. The potential at a point P of the flow is given by Eq. (2).

$$\Phi(P) = -\frac{1}{4\pi}\int_{S_B}\left(\frac{1}{r}\sigma - \mu\cdot\vec{n}\cdot\nabla\frac{1}{r}\right)dS + \frac{1}{4\pi}\int_{S_W}\left(\mu\cdot\vec{n}\nabla\frac{1}{r}\right)dS + \Phi_\infty \tag{2}$$

S_B and S_W are respectively the body and wake surfaces, r is the distance between P and the surface, μ and σ the strength of the doublet and source singularities respectively.

2.1 GEOMETRIC PRE-PROCESSOR

As with any other discrete method, the Panel Method's results are affected by the discretization of the geometry. In order to improve the quality of the solution, a geometrical pre-processor was developed, subdividing the panels and generating new points at positions that reduce the sharp corners.

The sharp corners are identified by the angle between two adjacent panels. A tolerance is set and all the panels that don't fit the conditions are divided. The new points are generated with operations involving the tangent vectors of the panels and their bisectors, as shown in Fig. 2.

Three adjacent panels are used to divide the center one. Bisectors of the angles between them are calculated (Fig. 2a). The angles formed between these bisectors and the center panel has new bisectors (Fig. 2b), which are straight lines that will create a new point at their intersection (Fig. 2c). It is important to calculate the position of all new points before adding them to the contour, as this avoids loosing the body's symmetry, were there any.

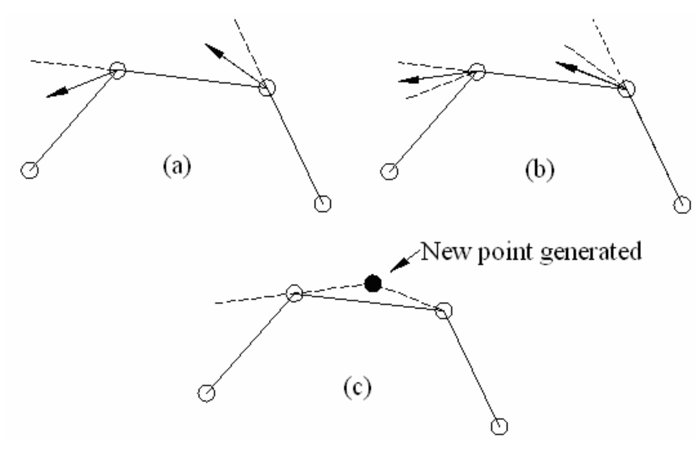

Figure 2. (a) Bisectors of the angles between panels. (b) More bisectors. (c) New point.

When applied to a square, such operations result in a circle as shown in Fig. 3, as was the expected result. This pre-processing is not done at the trailing edge, where the sharpness of the airfoil must be preserved.

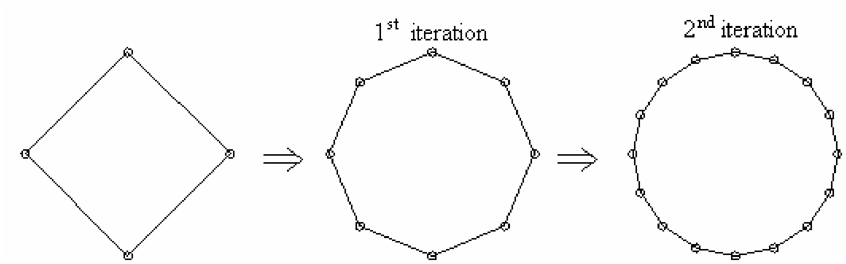

Figure 3. Pre-processing a square turns it into a circle.

2.1.1 Particular case – Collinear panels

When it occurs that two adjacent panels are in the same straight line, the new point will be generated at the same coordinates of an old one. This causes errors in the panel method. Because of that, collinear panels can not be divided by this pre-processor. If necessary, they are divided by simply adding points over their line.

One advantage is that the straightness of part of the contour can be forced by placing two collinear panels.

2.1.2 Particular case – Panels on curvature inflection

When dividing a panel that is at the inflection of the surface's curvature, the new point is generated far away from the body. To solve this problem, a method shown in Fig. 4 is used.

A new vector $\vec{t}_{rm}$ is placed at the middle of the center panel (Fig. 4a). Where $\vec{t}_{rm}$ is a bisector between $\vec{t}_{ri}$ and $\vec{t}_{ri+1}$, which are the directions of the lines that would generate the new point in an ordinary case. $\vec{t}_{rm}$ is then mirrored across the panel (Fig. 4b). Two new points are generated at the intersections of the straight lines orientated by $\vec{t}_{rm}$, $\vec{t}_{ri}$ and $\vec{t}_{ri+1}$ (Fig. 4c).

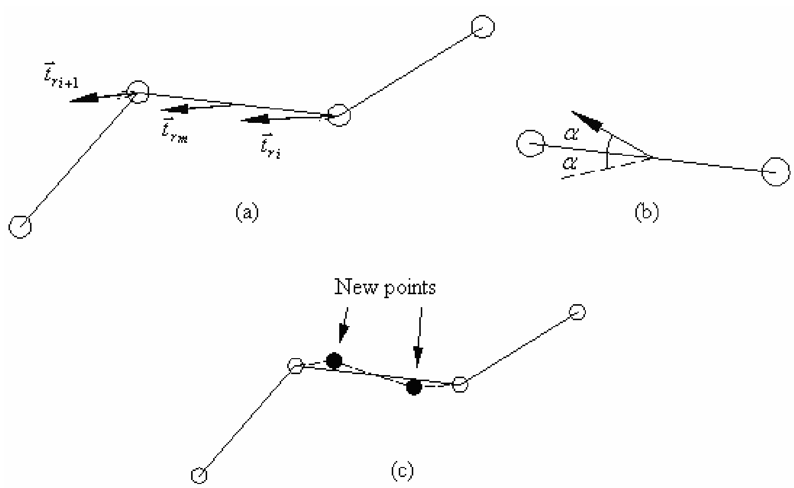

Figure 4. Panel on curvature inflection. (a) Middle vector $\vec{t}_{rm}$. (b) Mirrored $\vec{t}_{rm}$. (c) New points.

3. THE BOUNDARY ELEMENT METHOD FOR ELASTIC PROBLEMS

The direct BEM formulation (Brebbia, 1978) for linear bi-dimensional elasticity problems is given by:

$$C_{ij}(\xi)U_i(\xi) + \int_{\Gamma u} T^*_{ij}(\xi,\chi).U_j(\chi).d\Gamma(\chi) = \int_{\Gamma t} U^*_{ij}(\xi,\chi).T_j(\chi).d\Gamma(\chi) \tag{3}$$

With the boundary conditions: $U_j = \bar{u}_j$, in Γ_u e $T_j = \bar{t}_j$, in Γ_t. Where Γ is the boundary divided into Γ_u and Γ_t. Subscripts *i* an *j* represents one of the three orthogonal directions, ξ and χ are respectively the source and field points, *U* is the displacement of the point, *T* is the stress and *C* is a coefficient which depends on body geometry and is equal to ½ for smooth surfaces. U^* and T^* are the fundamental solutions of the Laplace's equation, given by:

$$T_{ij}^*(\xi,\chi) = \frac{-1}{4\pi(1-\nu)r}\left[\left((1-2\nu)\delta_{ij} + 2r_i \cdot r_j\right)\frac{\partial r}{\partial n} - (1-2\nu)(r_i \cdot n_j - r_j \cdot n_i)\right] \tag{4}$$

$$U_{ij}^*(\xi,\chi) = \frac{-1}{8\pi(1-\nu)G}\left[(3-4\nu)\ln(r).\delta_{ij} - r_i \cdot r_j\right] \tag{5}$$

Where ν is the Poisson Coefficient, *G* is the shear modulus, r_i e r_j are the unitary components of vector position ***r*** defined in Fig. 5, n_i and n_j are the normal components at the field point on the boundary Γ.

The numerical solution is obtained by subdividing the boundary Γ into N_e elements. The integrals are usually solved by the Gauss Quadrature procedure. When the collocation point is integrated within the element, it is necessary to use expressions developed to calculate the singular integral such as those given by Guiggiani and Lombardi (1992).

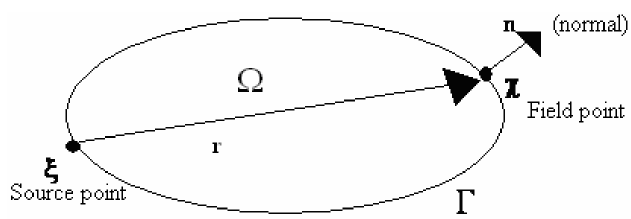

Figure 5. Basic definitions.

Applying the boundary conditions, $U_j = \bar{u}_j$ in Γ_u and $T_j = \bar{t}_j$ in Γ_t, and rearranging the equation, a linear system of equations with dimensions $2N_e$x$2N_e$ is set. The system is solved by Gauss elimination or by iterative solvers, especially when there is a larger number of degrees of freedom (Pessolani, 1999).

4 THE FLUID STRUCTURE INTERACTION SCHEME

The Fluid Structure Interaction Scheme is shown in Fig. 6.

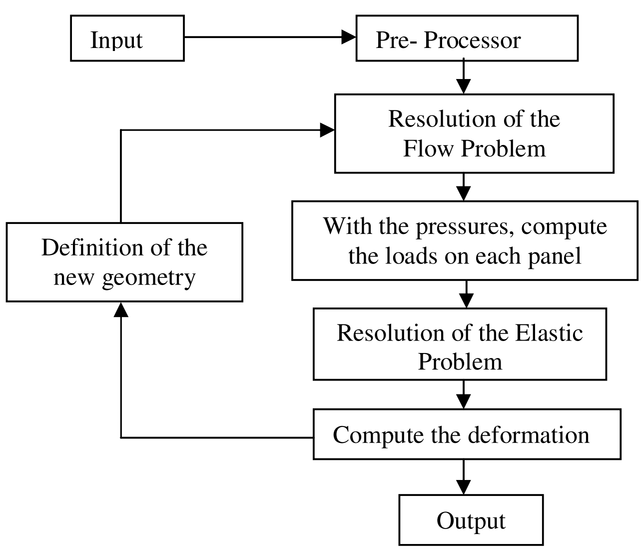

Figure 6. Diagram of blocks of the Fluid Structure Interaction Scheme

Initially the program reads the input data, which contains the body geometry and the flow and material properties. Then the program verifies the presence of sharp corners generating new points to smooth them, according to a given tolerance. After that, the Panel Method routine analyzes the flow and computes the velocities and pressures over each panel on the body surface. With the pressures, the elastic analysis is performed computing the displacement of each node, modifying the geometry and returning to the flow analysis. The process is repeated until the increments of force or displacements are nearly zero.

5. DYNAMIC FLUID-STRUCTURE INTERACTION

For the dynamic analysis, a separated model was developed. This model takes the airfoil as a rigid body supported by beams, which are treated like springs with spring constant components k_x and k_y, as shown in Fig, 7.

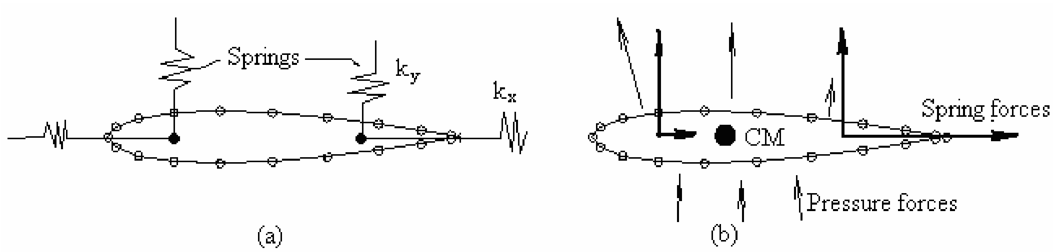

Figure 7. Model for the dynamic analysis. (a) Springs replacing the beams. (b) Spring and pressure forces.

The movement of the wing is then calculated in function of the acting forces F, the mass m, the moment of the forces around the center of mass M_{CM} and the moment of inertia at the center of mass J_{CM}.

5.1 Spring and pressure forces

The spring forces F_k depend on the airfoil position at each time step and are given by Eq. (6).

$$F_{kx} = k_x(x - x_0) \tag{6}$$

Where x is the horizontal component of the point respective to the spring and x_0 is the relaxed position of the spring. The same equation is valid for the vertical component y.

The pressure forces are calculated by the panel method and must consider the translation and rotation velocities of the airfoil at each time step. In a panel, the pressure force F_p is calculated as follows:

$$F_p = \int p \cdot dL \tag{7}$$

Where p is the pressure distribution over the panel and L its length.

5.2 Moments of the forces

The moments are calculated using all the forces in their respective locations.

$$M_{CM} = \sum F_y \cdot (x - x_{CM}) - \sum F_x (y - y_{CM}) \tag{8}$$

F_x and F_y are the horizontal and vertical components of each force, x and y are the coordinates of their respective points of application, x_{CM} and y_{CM} are the coordinates of the center of mass.

5.5 Accelerations

The center of mass has three acceleration components: the linear components $\ddot{x}_{CM}$, $\ddot{y}_{CM}$ and the angular component $\ddot{\theta}$. They are obtained using Newton's second law:

$$\ddot{x}_{CM} = \frac{\sum F_x}{m} \tag{9}$$

$$\ddot{\theta} = \frac{M_{CM}}{J_{CM}} \tag{10}$$

Equation (9) is also valid for the *y* component.

5.6 Movement and positioning

The airfoil's movement is obtained by integrating the accelerations using a step of time Δt. At each step the acceleration is constant, therefore the displacement Δx is:

$$\Delta x = x_{CM}(t) + \dot{x}_{CM}(t) \cdot \Delta t + \frac{1}{2}\ddot{x}_{CM}(t) \cdot \Delta t^2 \tag{11}$$

The velocities are incremented as follows:

$$\dot{x}_{CM}(t + \Delta t) = \dot{x}_{CM}(t) + \ddot{x}_{CM}(t) \cdot \Delta t \tag{12}$$

Equation (11) and Eq. (12) are the same for the vertical Δy and angular $\Delta\theta$ displacements.

With the displacements and increases or decreases of velocity, a new position is calculated for all points. This new position and velocity relative to the free stream will cause a change of the flow, so a new pressure distribution is calculated with the panel method and the process is repeated, showing a vibrating movement.

6. RESULTS

For test comparison, a Van de Vooren airfoil with 1m chord, 15% thickness, having 20 boundary nodes and an angle of attack equal to 20º was immersed in a fluid with velocity equal to 1m/s.

The original mesh refinement at the leading edge and the resulting one after applying the pre-processor are shown in Fig. 8. A tolerance of 30° was used for the angle between adjacent panels. It can be seen how the sharp corners are modified by adding a number of new nodes, converting them into smooth boundaries. Notice that the new discretization is closer to the real form.

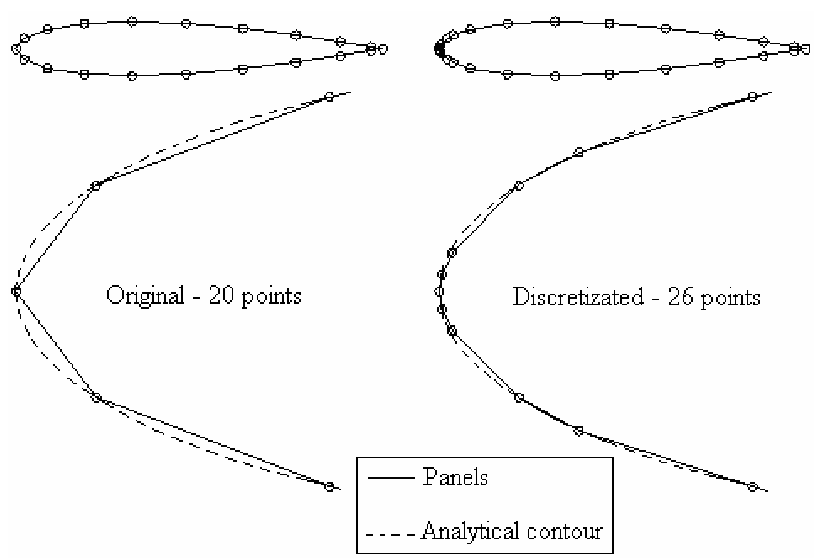

Figure 8. Discretization of a Van de Vooren airfoil at the leading edge.

Figure 9 presents a comparison between Cp coefficients of three solutions: analytical, panel method using 20 nodes (original panels) and 26 nodes (with smoothed leading edge). Notice the increment of accuracy for the smooth discretization.

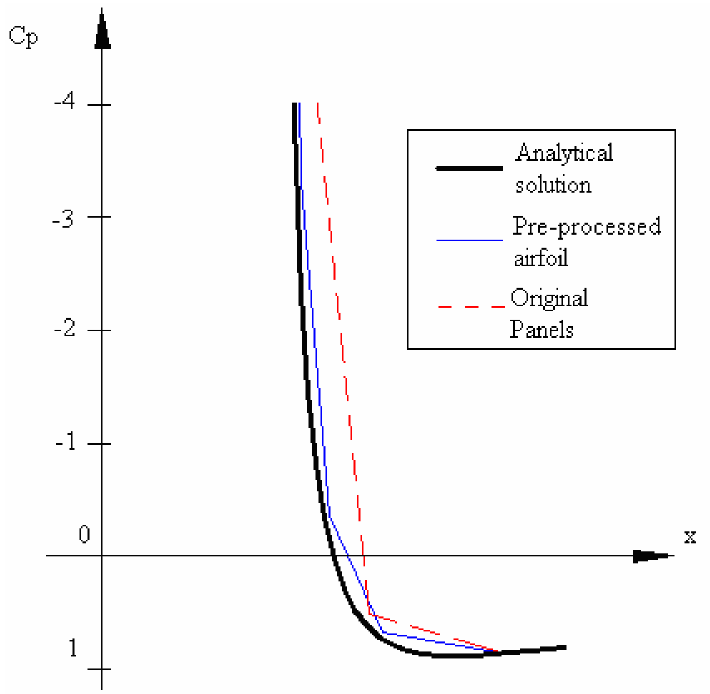

Figure 9. *Cp* graphics at the leading edge.

6.1 Example 1 – Airfoil supported by two rigid beams.

In this example, the airfoil is supported by two rigid beams, one at the leading edge and another at the trailing edge. The fluid-structure interaction scheme for static problems was applied. Figure 10 shows the deformed shape after 8 iterations, when convergence was reached. As expected, the airfoil deforms upwards and more intensely near the trailing edge, where it gets thinner.

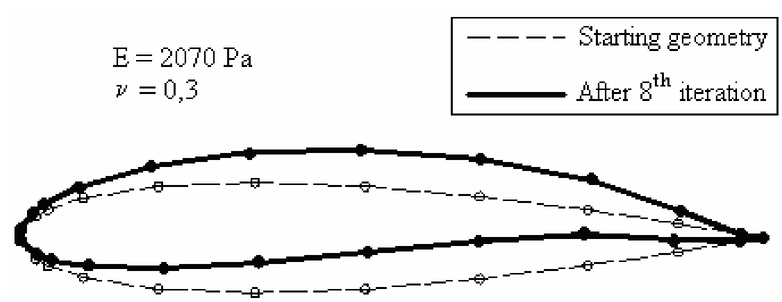

Figure 10. Comparison between the initial and final geometry of the airfoil.

Figure 11 shows the surface forces due to the pressure at the first and final iterations. It can be seen how the pressure is modified with airfoil deformation. At the 8th iteration, the pressure is lower at the front and higher at the back, compared to the first iteration.

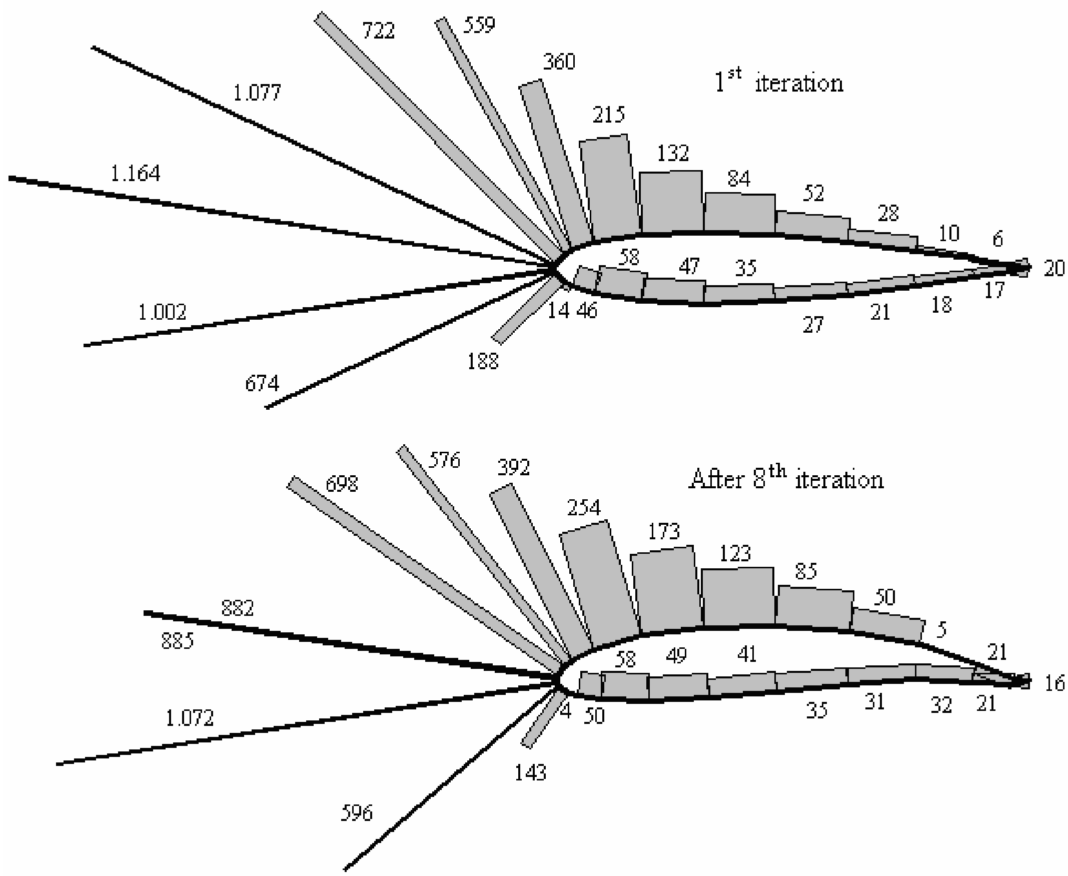

Figure 11. Evolution of the load applied on surface panels at the 1st and 8th iterations (10^{-2} N/m).

6.2 Example 2 – Airfoil supported by elastic beams.

To see how the airfoil can vibrate under pressure, this second example solves the dynamic model showed on Fig. 12, with two elastic beams supporting it.

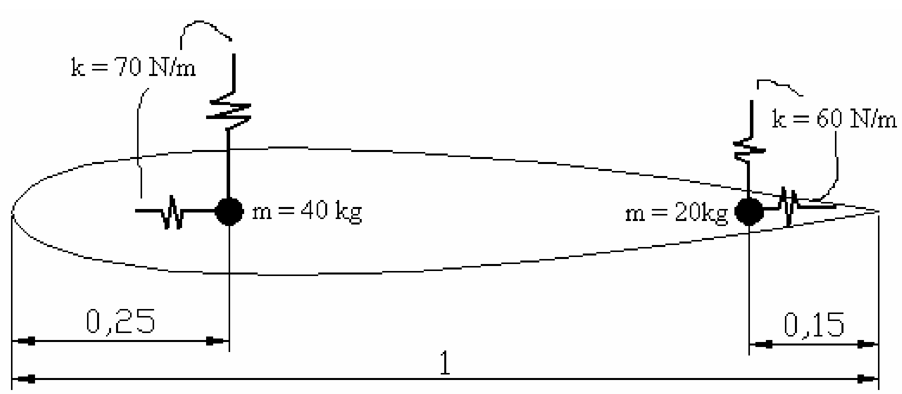

Figure 12. Vibration model with two elastic beams represented by springs.

A time step of 0,001s was taken and the final results are shown in Fig. 13. The vibration amplitude of the wing can be seen, of nearly 7 cm. The frequency is about 0,25 Hz.

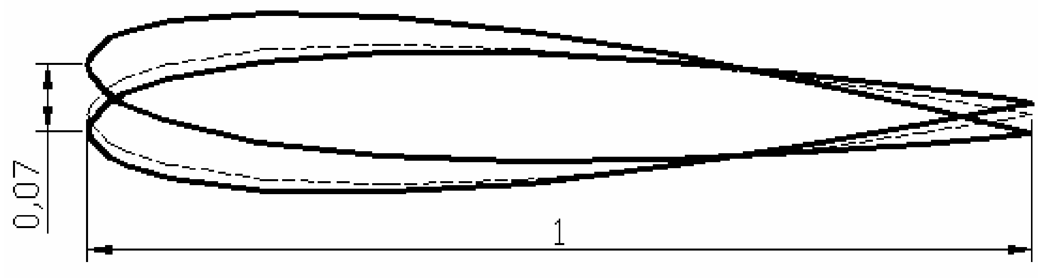

Figure 13. Wing vibration under wind pressure.

7. CONCLUSIONS

A scheme for a fluid-structure interaction using boundary formulation has been proposed. The fluid problem was solved using Panel Methods while the structure was solved with the Boundary Element Method for Elasticity. Both problems have the same mesh. This simplifies and avoids coupling problems on the interface when using domain methods.

The formulation was applied to two examples using the same airfoil with different supports: a rigid support for static analysis and an elastic one for dynamic analysis. The results show the applicability and the precision that can be obtained with few elements. Also, for the dynamic problem, the vibration frequency was obtained with a simple method.

As a conclusion, Boundary formulations can be further explored in order to solve fluid structure interaction problems on account of data entry simplicity, considerable reduction of the system dimension, and the easy way to treat the two problems using the same mesh. This procedure can also be applied to maritime structures, such as oil ducts and platforms. The dynamic analysis performed has to be improved and the formulation needs to be tested for more bodies and extended to 3D problems.

8. ACKNOWLEDGEMENTS

The authors wish to thank the CNPq for its support in developing this work.

9. REFERENCES

Anderson, J., 2001, "Fundamentals of Aerodynamics", Ed. McGraw-Hill, 3rd Edition.

Brebbia, C.A., 1978, "The Boundary Elements Method for Engineers", Ed. Pentech Press.

Katz, J., and Plotkin, A., 2002, "Low-Speed Aerodynamics", Cambridge University Press, 2nd ed.

Guiggiani, M. and Lombardi, F., 1992, "Self-Adaptive Boundary Elements with h-Hierarchical Shape Functions", Advances in Engng. Software, pp. 269-278.

Lachat, J.C. and Watson, J.O., 1975, "Effective Numerical Treatment of Boundary Integral Equation Method", Int. J. Num Meth. Engng, 10, 991-1005.

Liu W. K., 1980, "Development of Finite Element Procedures for Fluid Structure Interaction", A Report on Research Conducted Under Grants from the National Science Foundation and the Electric Power Research Institute, Pasadena, California.

Pessolani, R. B. V., 1999, "Computational Efficiency In Problems with a larger number of degrees of freedom for adaptive H-P Hierarchical Formulation Applied to the Boundary Element Method" In: XV COBEM - Congresso Brasileiro de Engenharia Mecânica, Aguas de Lindoia.

10. RESPONSIBILITY NOTICE