A robust weighted-by-area interpolation method for 3D fluid-structure interaction

Tiago Etiene Queiroz
Universidade de São Paulo - Instituto de Ciências Matemáticas e de Computação. Av. Trabalhador São-carlense, 400 - São Carlos - SP.
etiene@grad.icmc.usp.br

Igor Prata Soares
Universidade de São Paulo - Instituto de Ciências Matemáticas e de Computação. Av. Trabalhador São-carlense, 400 - São Carlos - SP.
igor@lcad.icmc.usp.br

Luis Gustavo Nonato
Universidade de São Paulo - Instituto de Ciências Matemáticas e de Computação. Av. Trabalhador São-carlense, 400 - São Carlos - SP.
gnonato@icmc.usp.br

Antonio Castelo Filho
Universidade de São Paulo - Instituto de Ciências Matemáticas e de Computação. Av. Trabalhador São-carlense, 400 - São Carlos - SP.
castelo@icmc.usp.br

José Alberto Cuminato
Universidade de São Paulo - Instituto de Ciências Matemáticas e de Computação. Av. Trabalhador São-carlense, 400 - São Carlos - SP.
jacumina@icmc.usp.br

Marcello Mendes Hasegawa
Universidade Federal de São Carlos - Dep. de Física. Rodovia Washington Luís (SP-310), Km 235 - 13565-905 - São Carlos - SP.
hasegawa@df.ufscar.br

Gustavo Campos Padovese
Embraer - Aeroelastic Stability. São José dos Campos - SP.
gustavo.padovese@embraer.com.br

Guilherme Lara Oliveira
Embraer - Aerodynamics & Flight Mechanics. São José dos Campos - SP.
guilherme.oliveira@embraer.com.br

Abstract. *Coupled fluid-structure analysis is an important problem in aerospace industry. An alternative to a monolithic solution where both fluid and structure problems are solved simultaneously within a single code, is to link available fluid and structure analysis packages by interpolating properties. In this work, as preliminary results for the above approach, we show the first part of a two-way interpolation solution. In the interpolation from the fluid to the structure we employ a robust 3D method, for obtaining the force field over the structural mesh from the pressure field on the fluid mesh. The method relies on a weighted-by-area interpolation scheme which deals with complex geometry configurations. Many exception cases arise due to: inhomogeneous size of the elements, different scale of the structure mesh elements compared to the fluid mesh, and the possibility of having quadrilateral and triangular elements. These exception cases are all dealt with properly. Results for some aircraft components, employing different structural meshes, will be presented.*

Keywords: *aeroelasticity, fluid-structure, interpolation*

1. Introduction

The numerical simulation of fluid-structure interaction phenomena is an important approach used for solving aerospace industry problems in aircraft design [1]. In this work we present a method for coupling computational fluid dynamics (CFD) codes with computational structural dynamics (CSD) codes. Coupling of CFD/CSD codes is usually categorized in two ways: fully or loosely coupled. Fully coupled approaches require the solution of the CFD and CSD equations simultaneously. In this approach the coupling scheme is directly implemented into the code. Loosely coupled methods do not integrate the coupling schemes into either the CFD or CSD code. This allows for the use of a variety of CFD/CSD codes. The present coupling method employs the second approach. For this, it must accept input data from the CFD code generating data to the CSD code in one way, and accept data from the CSD code generating data to the CFD code in the other way. The coupling method is required to exchange information between CFD and CSD codes through the codes' native files, thus no code integration is required. In this work we shall only report results obtained from transferring CFD data to the CSD code. This is acomplished by a robust 3D interpolation method. The interpolation method can use structured and unstructured meshes, do not rely on any assumption concerning density of elements and allows quadrilateral and triangular elements for both fluid and structural meshes. This work will be organized as follows: first we will present an overview of the coupling method, followed by details of the 3D interpolation method, and finally, examples will be presented and analyzed.

2. Overview

The coupling method presented in this work is intended to convert a pressure field in a fluid mesh into a force field, and, subsequently transfer it to a structural mesh. A 3D interpolation scheme is used for transfering the force field. No physical conservation laws are taken into account for the interpolation. Conversely, the method weighs the contribution to the force, from each cell in the fluid mesh to a given cell in the structural mesh, based only on geometrical information. In the limit where both fluid and structural meshes are perfectly conformal, i.e. in a perfectly matching interface, the interpolation is exact. This is not true for other interpolation schema. The method permits the use of a non matching interface. The quality of the results will depend on how broad the gap between the fluid and structural meshes is. Through this, a certain control of interpolation quality is conferred to the user. The interpolation process can be summarized as follows:

1. Load the fluid and structure meshes.
2. Load the pressure field obtained from the CFD code.
3. Converts pressure into force for each fluid cell.
4. Determine the cells, $\{C_f\}$, from the fluid mesh that contribute to the force over a given structure cell, C_s.
5. Determine the projected area for the fluid cells, $\{C_f\}$, over the structural cell, C_s.
6. Interpolate the force using the weighted-by-area interpolation scheme.
7. Write out the result to be used by the CSD code.

The meshes along with the pressure and the resulting force fields are read or written in one of the following industry standard formats: NASTRAN, Tecplot or Ensight. The data structure is flexible and can support singularities in the mesh topology [2]. Such singularities are supported by the interpolation method but can introduce errors if it implies lack of information. It is the user responsibility to ensure the quality of the mesh.

3. 3D Interpolation

The force in each fluid cell is calculated from its pressure, its area and normal vector. The area and the normal vector are obtained from a suitable cross product. The direction of the resulting force will depend on the cell's vertices orientation, following the right-hand rule. The resulting forces are interpolated on the structural cells. For this, the weighted-by-area interpolation scheme is used [3]. In this scheme the force is given by:

$$\vec{F}_s = \sum_i \frac{A_i}{A_s} \vec{f}_i \ , \tag{1}$$

where the index i runs over all the fluid cells that contributes to the force on the structural cell s, A_s is the area of the structural cell and $A_i = A_s \cap A^i_{fp}$, A^i_{fp} being the projected area of a given fluid cell i. In order to obtain the projected area the vertices of the fluid cell i are orthogonally projected over the structural mesh. The projected vertices defines a polygon whose area is the projected area A^i_{fp}. The intersection $A_s \cap A^i_{fp}$ is calculated by polygon intersection. Polygon intersection in the 3D space is not a simple problem. An algorithm that deals properly with the many exceptions cases that arises from the many possible geometry configurations was carefully designed. This algorithm is limited to work on convex polygons and have a computational efficiency of $O(n)$, where n is the total number of edges in both polygons. It is based on the convex hull test and also on an algorithm for obtaining the intersection points of polygons in 2D space [4]. In order to find the cells from the fluid mesh that contributes to the force interpolated on a given cell in the structural mesh, an algorithm was implemented. It avoids sweeping all the cells on the fluid mesh to verify if they might contribute to the interpolation. Figure 1 represents two cases that have to be dealt with. In Fig. 1 a) the structural mesh is almost conformal to the fluid mesh; the gap between them is exaggerated in the figure. In Fig. 1 b) the structural mesh is only a rough approximation to the structure itself. For the case depicted in Fig. 1 a) it is clear which regions of the fluid mesh are going to contribute to the interpolation on the structural mesh. Figure 1 b) shows a different situation; in this case forces from both sides of the structural mesh are going to contribute to the results of the interpolation. In order to treat these

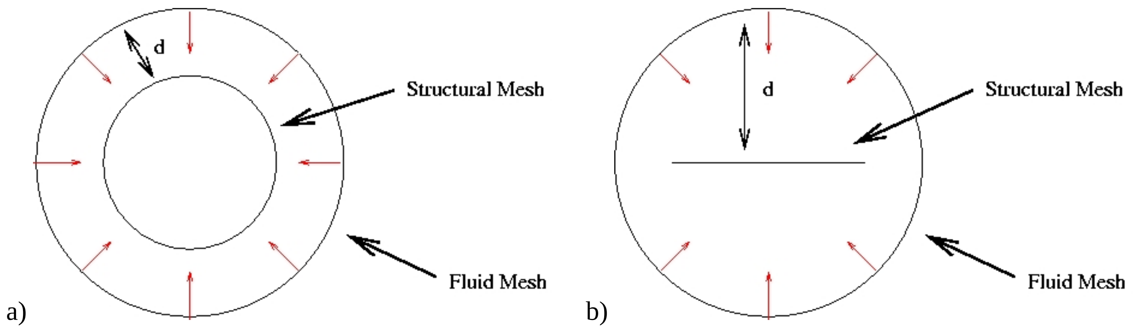


Figure 1. This figure shows two cases of structural mesh configurations defining different interfaces.

cases, a parameter d is required to be specified by the user. This parameter, as sketched in both Fig. 1 a) and b), is the maximum distance from which a cell in the fluid mesh can contribute to the force of a cell in the structure mesh. Thus, it is clear that if we make the parameter d large enough, the structural mesh in Fig. 1 b) can receive contributions from the opposite sides of the fluid mesh as required. In the same way the parameter d can be adjusted to restrict the contributions for the interpolation as required in the case shown in Fig. 1 a).

4. Results

Some results are shown in this section. Figure 2 shows both fluid and geometry meshes for a hypothetical wing. We see in a) a fluid mesh composed of triangular elements, and, in b) a structural mesh composed of quadrilateral elements. The structural mesh, in this case, is simply a open flat surface not conformal to the fluid mesh. The fluid mesh is more faithful to the wing geometry. Figure 3 shows the pressure field obtained from a CFD simulation. The pressure field is

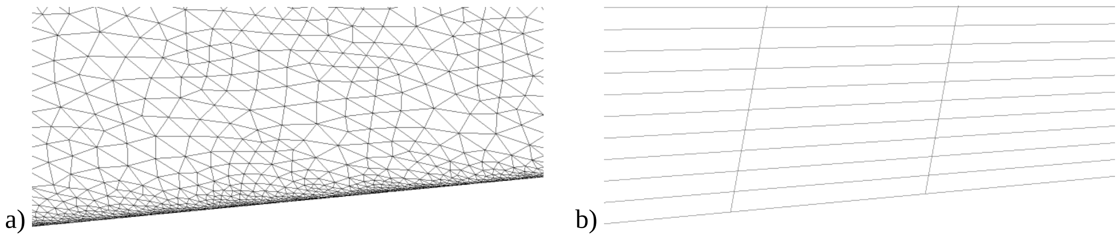


Figure 2. Meshes employed for the wing model. The fluid mesh is seen in a) and he structural mesh in b).

represented by colors. Clear blue to dark represents decreasing negative pressure relative to a given reference pressure, or outward pressure, and, clear red to dark represents decreasing positive pressure or inward pressure. In both cases dark represents the reference pressure.

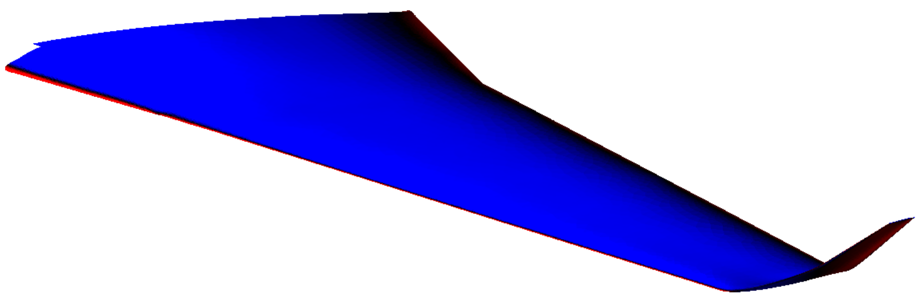


Figure 3. Pressure field over the wing (top view). The blue area shows a pressure field originating a force field pointing outward.

Figure 3, Fig. 4 and Fig. 5 shows different perspectives of the wing. We observe in Fig. 5 the presence of holes (four) in the geometry, these affect the results of the interpolation. Figure 6 shows the results of the interpolation. In this picture

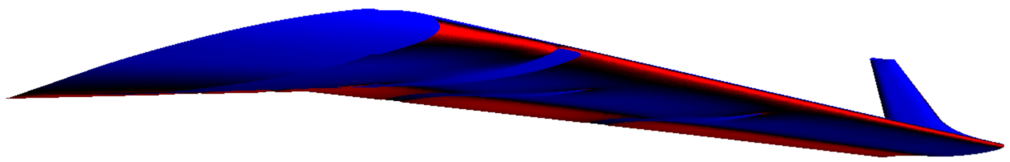


Figure 4. Pressure field over the wing (side view). We can see on the leading edge a red region where the wing feels a inward force.

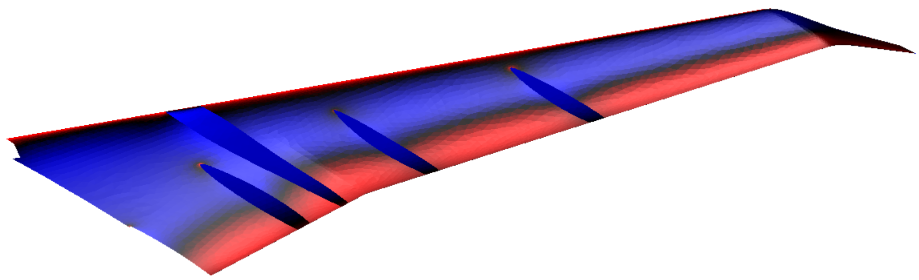


Figure 5. Pressure field over the wing (botton view). On this figure we can observe a more complex pattern on the pressure field. Red represents inward pressure, the dark region the zero pressure region and blue represents an outward pressure field.

the interpolated forces where converted back again to a pressure field, now, over the structural mesh. This was done by simply dividing the interpolated force projected along the normal direction for each structural cell by its area. The forces contributing to these results come from both sides of the fluid mesh. We can observe the dark pattern in the interpolation

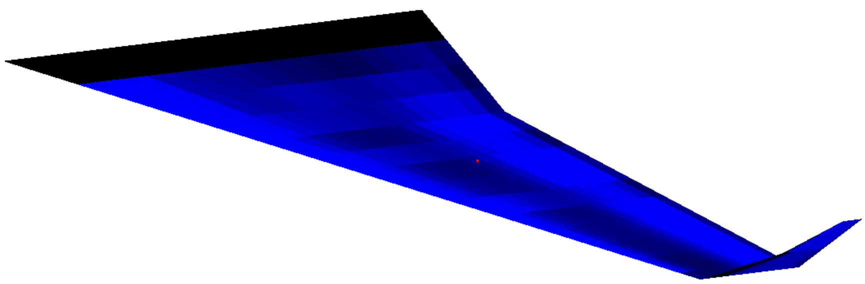
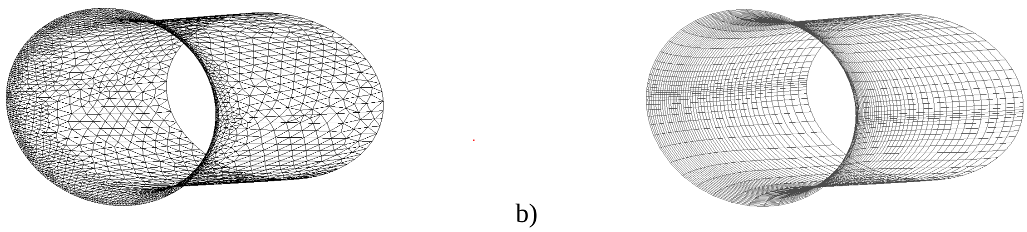


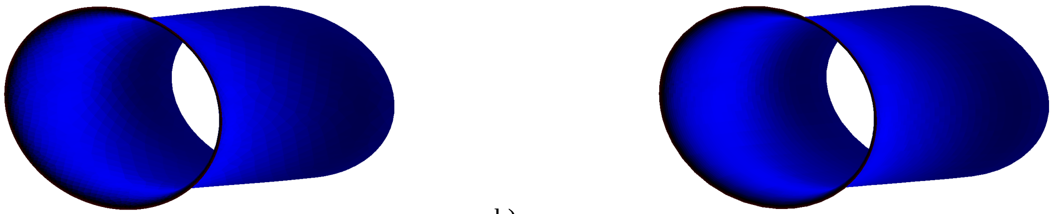
Figure 6. This figure shows the results for the interpolation on the structural mesh (top view). Dark patterns arise from the holes on the fluid mesh and the zero pressure region on the pressure field.

result. This comes from the presence of holes in the fluid mesh and the zero pressure region shown in Fig. 5. Figure 7, a) and b) shows respectively the fluid and the structural meshes for a nacelle. Distinctly from the case of the wing, the meshes are geometrically very close to each other. Figure 8 a) shows the pressure field and Fig. 8 b) shows the results of the interpolation. We observe that the pressure field resulting from the interpolated forces is very close to the original



a) b)

Figure 7. Meshes for the nacelle model. The fluid mesh is seen in a) and he structural mesh in b).



a) b)

Figure 8. This figure shows the pressure field in a) and the pressure field obtained from the interpolated forces on the structural mesh in b).

pressure field.

5. Conclusion

The interpolation scheme developed produces good results on transferring a pressure field from a fluid mesh to a force field on a structural mesh. We have obtained satisfactory results on different geometries and different fluid-structure interface configurations. Although these results are preliminary they encourage the continuity of this work in developing an efficient scheme for the interpolation of displacements from the structural mesh to the fluid mesh. Together these can be used to complete a fluid-structure interaction loop between CFD and CSD codes.

6. References

Bhardwaj, M.K., 1997, "A CFD/CSD Methodology for Aircraft Wings", PhD Thesis, Faculty of Virginia Polytechnic Institute and State University, USA.

de Berg, M., van Kreveld, M., Overmars, M., Schwarzkopf, O., 1997, "Computational Geometry Algorithms and Applications - Second Edition", Springer-Verlag, Berlin, Germany.

de Boer, A., Bijl, H., van Zuijlen, A., Procedings 17th AIAA CFD Conference, 2005, "Comparing Different Methods for the Coupling of Non-Matching Meshes in Fluid-Structure Interaction Computations", 2005-4620.

O'Rourke, J., 1998, "Computational Geometry in C - Second Edition", Cambridge University Press, UK, pp. 252.