RESEARCH ARTICLE OPEN ACCESS

Comparison of the FE/FE and FV/FE treatment of fluid-structure interaction

R. El Maani[1*], B. Radi[1] and A. El Hami[2]
[1]LIMII, FST Settat, BP: 577, Route de Casablanca, Settat, Maroc
[2]LOFIMS, INSA de Rouen, BP: 8, Avenue de l'Université, 76801 St Etienne de Rouvray, France

I. Introduction

Fluid structure interaction (FSI) is a multi-physics phenomenon which occurs in a system where flow of a fluid causes a solid structure to deform which, in turn, changes the boundary condition of a fluid system. This can also happen the other way around where the structure makes the fluid flow properties to change. This kind of interaction occurs in many natural phenomena and man-made engineering systems. It becomes a crucial consideration in the design and analysis of various engineering systems. For instance, FSI simulations are conducted to avoid flutter on aircraft and turbo-machines [1], to evaluate the environmental loads and dynamic response of offshore structures [2] and in many bio medical applications.

In a typical single-field mechanics problem, such as a fluid-only or structure-only problem, one begins with a set of governing differential equations in the problem domain and a set of boundary conditions on the domain boundary. The domain may or may not be in motion. The situation is more complicated in an FSI problem. The sets of differential equations and boundary conditions associated with the fluid and structure domains must be satisfied simultaneously. The domains do not overlap, and the two systems are coupled at the fluid–structure interface, which requires a set of physically meaningful interface conditions. These coupling conditions are the compatibility of the kinematics and tractions at the fluid-structure interface.

This paper aims the analysis of the fluid-structure interaction of a three-dimensional wing of aircraft in airflow to determine its behavior. The fluid and structure were modeled independently and exchanged boundary information to obtain aero elastic solutions. The fluid was modeled using both two different discretization methods used as computational fluid dynamics (CFD) solvers based on the finite volume method and finite element method, the structure was modeled using finite element approximations and the two disciplines were coupled to solve aero elastic problems. The loads obtained from the pressure are applied to the original finite element model to obtain the displacements. The code ANSYS© is used as a pre-processing tool for creating the whole computational domain and volume mesh. The fluid's flow is solved separately using two solvers: Ansys/Fluent© and Ansys/Flotran© and coupled with structural code.

II. Fluid-structure interaction problem

A general FSI problem consists to descript the fluid and solid fields, appropriate fluid structure interface conditions at the conjoined interface and conditions for the remaining boundaries, respectively. In this paper, we restrict ourselves to the incompressible flows, which is a reasonable choice for many engineering applications.

In the following, the fields and interface conditions are introduced; furthermore, a brief sketch of the solution procedure for each of the fields is presented.

2.1 Fluid's equations

All kinds of fluid flow and transport phenomena are governed by basic conservation principles such as conservation of mass, momentum and energy. All these conservation principles are solved according to the fluid model which gives set of partial differential equations, called the governing equations of the fluid. The following part elaborates on the theoretical background of CFD and the way it is employed for this particular case.

The mass conservation principle states that the rate of increase of mass in a fluid element is equal to the net rate of flow of mass into a fluid element. Applying this physical principle to a fluid model results in a differential equation called continuity equation [4]. The continuity equation for a compressible fluid can be written as follows:

$$\frac{\partial \rho}{\partial t} + div(\rho u) = 0 \qquad (1)$$

where ρ represents the density and *u* represents the velocity of the fluid. The first term of the equation is the rate of change of density with respect to time and the next term is the flow of mass out of the element boundaries.

Newton's second law states that the rate of change of momentum of a fluid particle equals to the sum of the forces acting on a particle. The forces

acting on a body are a combination of both surface and body forces. When this law is applied for Newtonian fluid (viscous stress is proportional to the rates of deformation) resulting equations are called as Navier-Stokes equations. The equations written below explain the momentum conservation principle [4]:

$$\frac{\partial(\rho u_i)}{\partial t} + div(\rho u_i u) = -\frac{\partial p}{\partial x} + div(\mu grad\ u_i) + S_{Mx} \quad (2)$$

$$\frac{\partial(\rho u_j)}{\partial t} + div(\rho u_j u) = -\frac{\partial p}{\partial y} + div(\mu grad\ u_j) + S_{My} \quad (3)$$

$$\frac{\partial(\rho u_k)}{\partial t} + div(\rho u_k u) = -\frac{\partial p}{\partial z} + div(\mu grad\ u_k) + S_{Mz} \quad (4)$$

where ρ represents the density, u represents the velocity vector, u_i , u_j , u_k are the velocity components in Cartesian coordinates system, μ is the dynamic viscosity and S_M represents the momentum source term.

2.2 Structure's equations

In structural mechanics problems, in general, the task is to determine deformations of solid bodies, which arise because of the action of various kinds of forces. From this, for instance, stresses in the body can be determined, which are of great importance for many applications. For the different material properties there exist a large number of material laws, which together with the balance equations lead to diversified complex equation systems for the determination of deformations (or displacements). The basic governing equation of motion is given as follows [10]:

$$m\ddot{u} + c\dot{u} + ku = f(t) \quad (5)$$

where m is a structural mass matrix, $\ddot{u}$ is an acceleration vector, c is a structural damping matrix, $\dot{u}$ is a velocity vector, k is a structural stiffness matrix, u is a displacement vector, f is a force vector which is a function of time, the structural damping is not involved in the finite element model so the above governing equation is modified into following form

$$m\ddot{u} + ku = f(t) \quad (6)$$

It is normal practice to use a numerical technique called finite element method (FEM) to find the solution for the equation (6), because it is not feasible to use analytical methods to determine the solution for a system with infinite number of degrees of freedom (DOFs). The basic principle behind this method to find an approximate solution to the differential equations is to divide the volume of a structure or system in to smaller (finite) elements such that infinite number of DOFs is converted to a finite value.

2.3 Interface conditions

The main conditions at the interface are the dynamic and kinematic coupling conditions. The force equilibrium requires the stress vectors to be equal as

$$\sigma^f.n = \sigma^s.n \quad \forall x \in \Gamma^{fsi} \quad (7)$$

We assume no mass flow across the consequently, also the normal velocities at interface the interface have to match as follows:

$$u.n = \frac{\partial d}{\partial t}.n \quad \forall x \in \Gamma^{fsi} \quad (8)$$

III. Numerical discretization

In both discretization methods, the numerical computation is developed in two steps. In the first one, the conservation equations are formulated and an approach is adopted to evaluate all the terms. In the second one, a segregated, sequential solution algorithm is used to form the element matrices, to assemble them and to solve the resulting system for each variable separately [5]. In order to solve the governing equations of the fluid motion (2) (3) (4), their discretized form must first be generated. Thus, the first step is the generation of a grid, which consists to divide the solution domain into a finite number of control volumes or computational elements [14]. In the second step, each term of the partial differential equation describing the flow is written in such a manner that the computer can be programmed to calculate it [4].

The dynamics conservation equations of the generic variable in three dimensions that describe the transport phenomena for flows in free convection are of the general form:

$$\frac{\partial\left(\rho C_\phi \phi\right)}{\partial t} + \frac{\partial\left(\rho v_i C_\phi \phi\right)}{\partial x_i} = \Gamma_\phi \frac{\partial^2 \phi}{\partial x_i^2} + S_\phi \quad (9)$$

where ϕ represents the common variable of interest as a concentration of the transported quantity in a non-dimensional form, ρ is the density of the air, C_ϕ is the advection coefficient (C_ϕ is the specific heat of air in the energy equation and the unit stands for the other conservation equations), v_i is the component of velocity vector in the direction i, Γ_ϕ is the

diffusion coefficient and S_ϕ is the source term. These variables with their different terms are shown in [13] for a steady state (first term in Eq (9) is equal to zero) and incompressible flow.

In order to solve the governing equations of the fluid motion [Eq (9)], their discretized form must first be generated. Thus, the first step is the generation of a grid, which consists todivide the solution domain into a finite number of control volumes or computational elements [14]. In the second step, each term of the partial differential equation [Eq (9)] describing the flow is written in such a manner that the computer can be programmed to calculate it [4].

3.1 Finite volume discretization

The finite volume method is one of the numerical techniques applied in well established commercial CFD codes to solve the governing equations of the fluid. The basic and foremost step of CFD is dividing the computational domain (geometry of the region of interest) in to number of smaller regions called control volumes or cells and the collection of these cells is called a grid or a mesh, also, the calculated scalar values are stored at the center of the control volumes. Fluent uses the finite volume technique to convert the general transport equation into a system of algebraic equations and it uses different iterative methods to solve the algebraic equations. The following are the key steps in order to find the solution for the transport equation of a physical quantity [4]. The steps are as follows:

- Division of geometry in to smaller regions (control volumes) using a computational mesh.
- Integration of the governing equations of fluid over all the control volumes of the domain.
- Discretization-conversion of the resulting integral equations in to a system of algebraic equations.
- Finding a solution to the system of algebraic equations by an iterative method.

The general form of transport equation in conservative form can be written as [4]:

$$\frac{\partial(\rho\phi)}{\partial t} + div(\rho\phi u) = div(\Gamma\, grad\,\phi) + S_\phi \quad (10)$$

where the variable ϕ can be replaced by any scalar quantity, Γ is the diffusion coefficient. The left hand side of the equation contains the rate of change term and convective term, whereas the diffusive term and source term lie on the right hand side of the equation. Integrating over the control volume and applying the Gauss's divergence theorem on the general transport equation gives [4].

$$\frac{\partial}{\partial t}\int_{CV}\rho\phi dV + \int_A n(\rho\phi u)dA = \int_A n\left(\Gamma\, grad\,\phi\right)dA + \int_{CV} S_\phi dV \quad (11)$$

3.2 Spatial and temporal discretization schemes

The above transport equation is subjected to the stated key steps of the finite volume technique and the discretized equation for each control volume is obtained through suitable discretization schemes. There are many spatial discretization schemes for formulating diffusive and convective terms of the transport equation.

In the case of diffusive term in the discretized equation, the gradients of a variable at the faces of the control volume are required. In order to find a value of this term, a central differencing scheme is used by considering linear approximation. On the other hand, the convective term at the faces of the control volume is evaluated by using an upwind scheme. The main idea of this scheme is that convective values at the face are calculated by using the values of upstream control volume or relative to the direction of the normal velocity (u). Fluent has a range of upwind schemes such as first order upwind, second order upwind, power law and QUICK (Quadratic Upstream Interpolation for Convective Kinetics scheme) [7]. All the above discretization schemes are mainly categorized by the order of solution accuracy. Apart from this, results produced from these schemes are physically realistic when it fulfills the following properties [4]: conservativeness, boundedness and transportiveness.

For unsteady calculations, the transport equation must be discretized in both space and time. Temporal discretization involves the integration of all terms of transport equation over a time step Δ*t*. The two main schemes available for temporal discretization in ANSYS/FLUENT are implicit time integration and explicit time integration [11].

3.3 Finite element discretization

The FEM divides the continuum region of interest into a number of simply shaped regions called elements. In this discretization method, the variables within each element are interpolated using a local polynomial $N_j(x_i)$ (shape or interpolation function) in terms of the values *j* at a set of nodal point *j* in a way that guarantees continuity of the solution across element sides [22,23]:

$$\phi = \sum_{j=1}^{n} N_j \phi_j \quad (12)$$

where N_j is a polynomial shape function at nodes j and *n* is the number of nodes on the element.

The discretization process, therefore, consists of deriving the element matrices to put together the matrix equation [5]:

$$\left(\left[A_e^{transient}\right]+\left[A_e^{advection}\right]+\left[A_e^{diffusion}\right]\right)\{\phi_e\}=\{S_e^{\phi}\} \quad (13)$$

Galerkin's method of weighted residuals is used to form the element integrals [5].

Each degree of freedom is solved in sequential fashion. The equations are coupled, so that each equation is solved with intermediate values of the other degrees of freedom. The process of solving all the equations in turn and then updating the properties is called a global iteration. Before showing the entire global iteration structure, it is necessary to see how each equation is formed [5].

IV. Fluid-structure treatment

4.1 Partitioned Analysis

In general, one can choose to describe the whole coupled system in a monolithic way and solve all fields together or separate the fields and couple them in the sense of a partitioned analysis. In the latter case either sequential (staggered) or iterative algorithms can be used. The monolithic approach is straightforward and allows to solve the resulting system of equations with a complete tangent stiffness matrix (if - in an ALE setting - fluid, structure and mesh degrees of freedom are included). However, such monolithic approaches have a number of obvious severe drawbacks like loss of software modularity, limitations with respect to the application of different sophisticated solvers in the different fields and challenges with respect to the problem size and conditioning of the overall system matrix. Hence they are generally considered not very well suited for application to real world problems, where often not only specific solution approaches but also specific codes should be used in the single fields, and for this and a number of additional reasons we prefer to use a partitioned approach [17].

For the fluid-structure coupling an implicit partitioned approach is employed [12]. After the initializations the flow field is determined in the actual flow geometry. From this the friction and pressure forces on the interacting walls are computed, which are passed to the structural solver as boundary conditions. The structural solver computes the deformations, with which then the fluid mesh is modified, before the flow solver is started again.

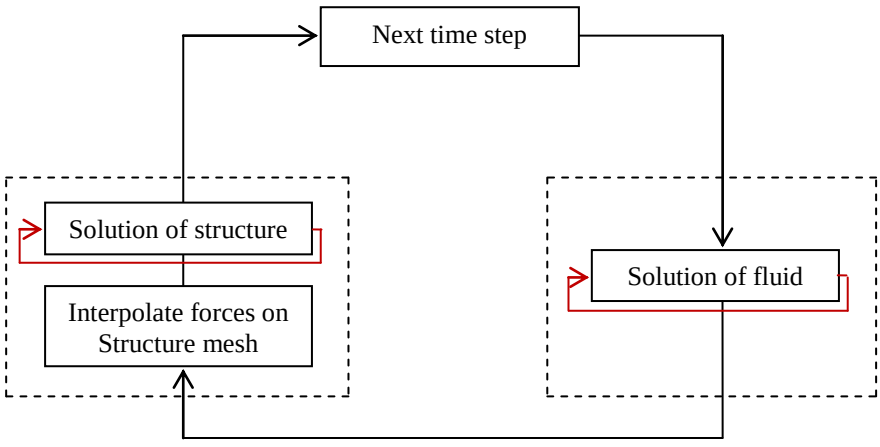
4.2 One-way coupling

The coupling is one-way if the motion of a fluid flow influences a solid structure but the reaction of a solid upon a fluid is negligible [11]. The other way around is also possible.

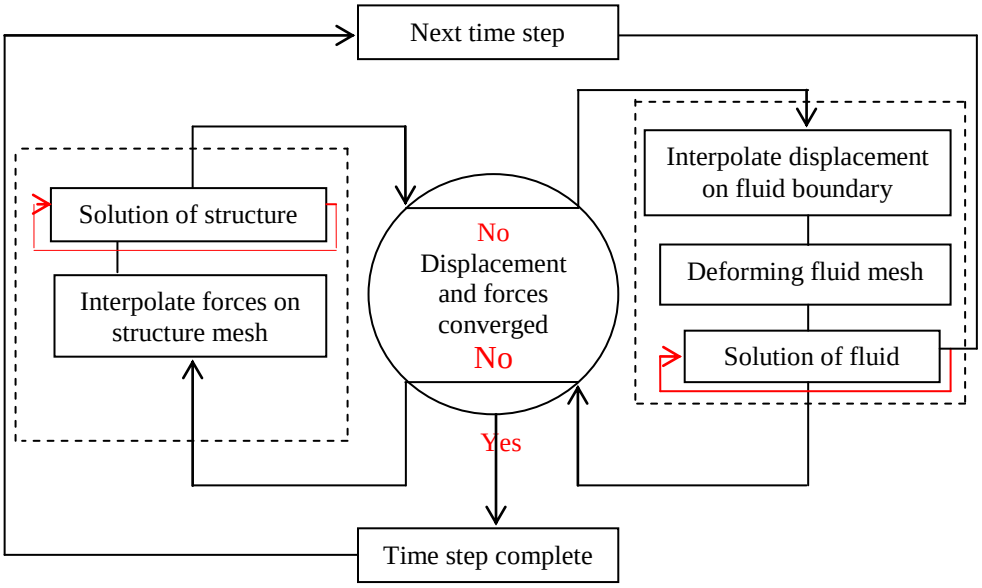
Initially, the fluid flow calculation is performed until convergence is reached. Then the resulting forces at the interface from fluid calculation are interpolated to the structural mesh. Next, the structural dynamic calculations are performed until the convergence criterion is met. This is repeated until the end time is reached.

4.3 Two-way coupling

This type of coupling is applied to the problem where the motion of a fluid influences a solid structure and at the same time the flow of fluid is influenced by reaction of a solid structure.During the first time step, converged solutions of the fluid calculation provide the forces acting on the solid body. Then the forces are interpolated to the structural mesh like in one-way coupling and the solution from the structural solver is obtained with those fluid forces as boundary conditions. As a consequence the mesh is deformed according to the response of structure. These displacement values are interpolated to the fluid mesh which results in deformation of the fluid domain. This process is repeated until both force and displacement values are converged below the pre-determined limit [11].



One-way coupling



Two-way coupling

Figure 1: Solution algorithm for one-way and two-way coupling

V. Numerical simulations

5.1 Fluid-interaction modelling

In ANSYS Workbench, the FSI (two-way coupling) analysis can be performed by connecting the coupling participants to a component system called System Coupling. A participant system is a system which either feeds or receives data in a coupled analysis. Here, Fluent (participant 1) and ANSYS Mechanical (participant 2) are acting as coupling participants [7]. Initially, system coupling collects information from the participants to synchronize the whole set up of simulation and then the information to be exchanged are given to the respective participant. The next step of the work process is organizing the sequence of exchange of information. The solution part of the chart varies for different ways of coupling. Finally, the convergence of coupling step is evaluated at end of the every coupling iteration.

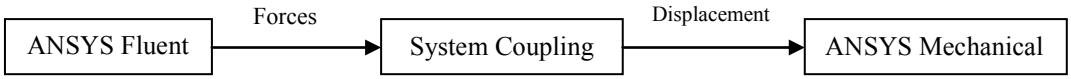


Figure 2: System coupling

As it launches the first time step, Fluent iterates and transfers the pre-requested information (fluid forces) to ANSYS Mechanical, so that this solver begins the iteration process to get nodal displacements. Now, the coupling service (System Coupling) collects the convergence status from both the participants and launches next time step. The calculated solution of ANSYS Mechanical is given back to the Fluent to determine a new set of fluid forces according to nodal displacements of previous step. This is said to be a coupling iteration and continues until the convergence criterion of data transfer is reached.

For the one-way coupling the result (forces or temperature or convection load) from a CFD analysis at the fluid-structure interface is applied as a load to the Mechanical application analysis. The boundary displacement from the Mechanical application is not passed back to the CFD analysis, that is, the result from the Mechanical application is not considered to have significant impact on the fluid analysis. In this case the called system coupling is not needed and the loads can be linked directly between the two systems.

Initially, geometric models of both fluid and solid domains are created with appropriate dimensions. ANSYS © is used as a preprocessor for creating the geometries models. The surface and volume mesh of fluent fluid domain are formed using GAMBIT and the finite element mesh is created by ANSYS Meshing. The two computational meshes differ with parameters such as cell type, cell size and mesh resolution. The completed meshes are imported to the respective numerical solvers where the simulation setup of a model is implemented. The simulation setup includes essential steps such as assigning the material properties, boundary conditions and numerical schemes for the two different models. In this paper, we propose a pre-stressed modal analysis of a wing of 3D model. The wing has uniform configuration along its length, and

its cross-sectional area is defined to be a straight line and a spline. It is held fixed to the body on one end and hangs freely at the other. The objective of the problem is the determination of the natural frequencies of the wing with the fluid flow in order to illustrate the effect of the fluid-structure interaction.

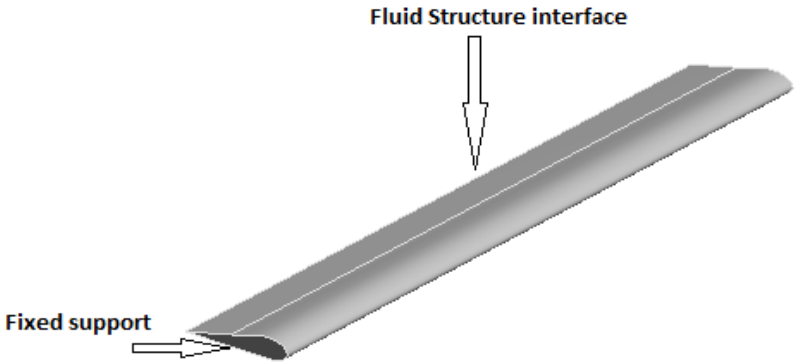


Figure 3: Boundary conditions of the structure model

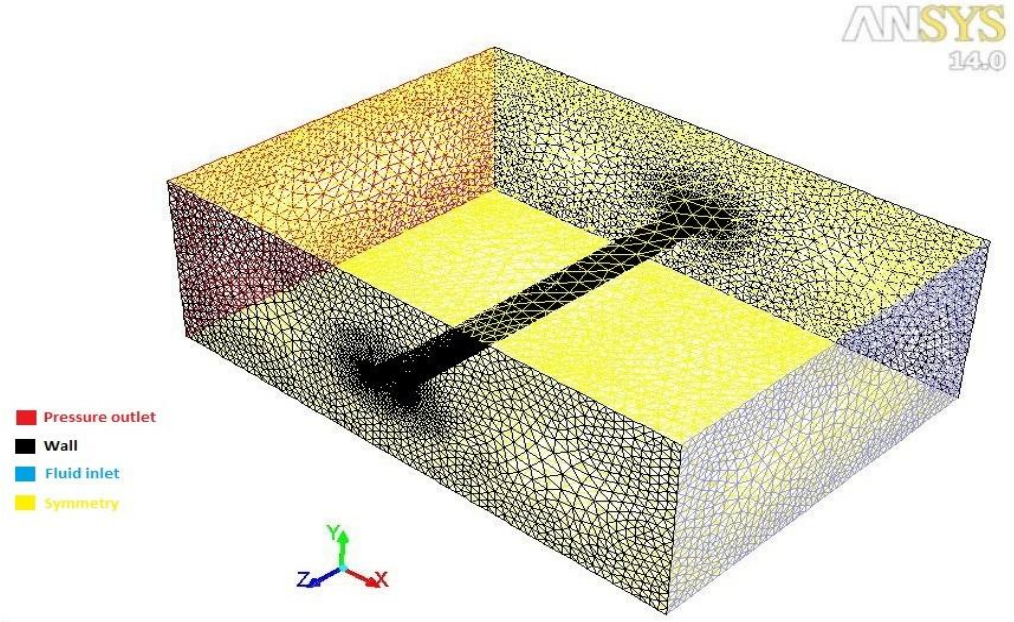


Figure 4: Boundary conditions of the fluid model

In the case of the CFD mesh, the surface mesh is first created using triangular elements, which is then used to create a volume mesh. The volume mesh is made up of tetrahedral cells, belonging to the category of unstructured mesh. The reason for not using hexahedral cells is that it is not compatible with the use of dynamic mesh in the current version of ANSYS/FLUENT. The mesh of the entire computational fluid domain is shown below in Figure 5.

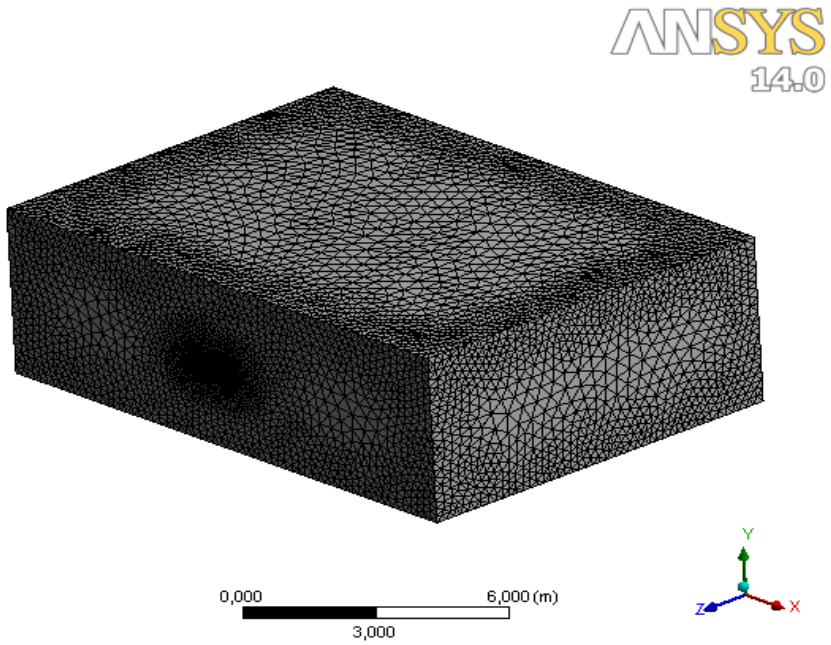


Figure 5: Computational mesh of the fluid domain (ANSYS/FLUENT)

The computational mesh of the structural member is created with the help of ANSYS Meshing tool. Figure 6 shows the mesh of structural member.

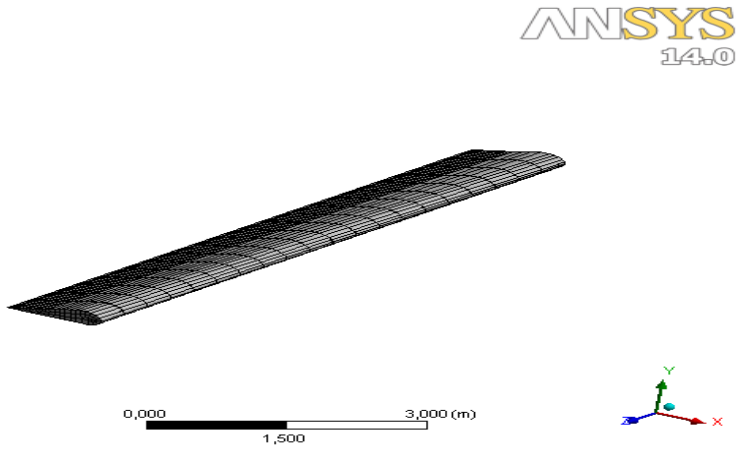


Figure 6: Structural mesh

For the finite elements calculation: SOLID186 is used for the 3-D modelling of solid, the element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. For the FLOTRAN CFD elements, FLUID142 is used for modelling the fluid flow and the interface in fluid-structure interaction problems as shown on Figure 7.

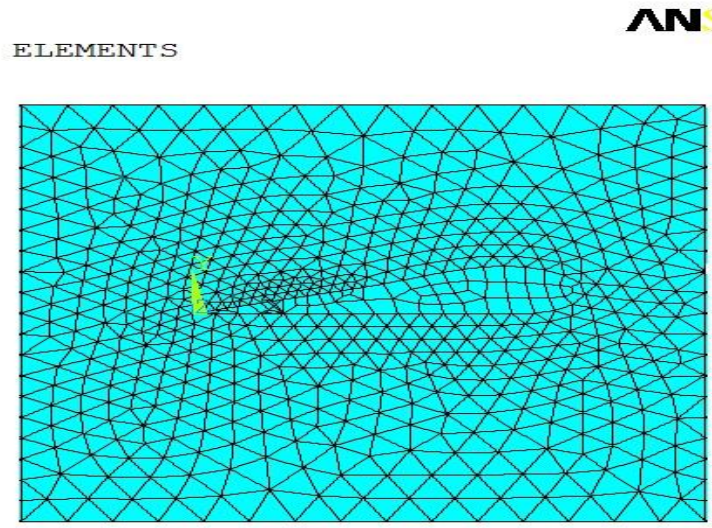


Figure 7: Computational mesh of the fluid domain (ANSYS/FLOTRAN)

5.2 Numerical results

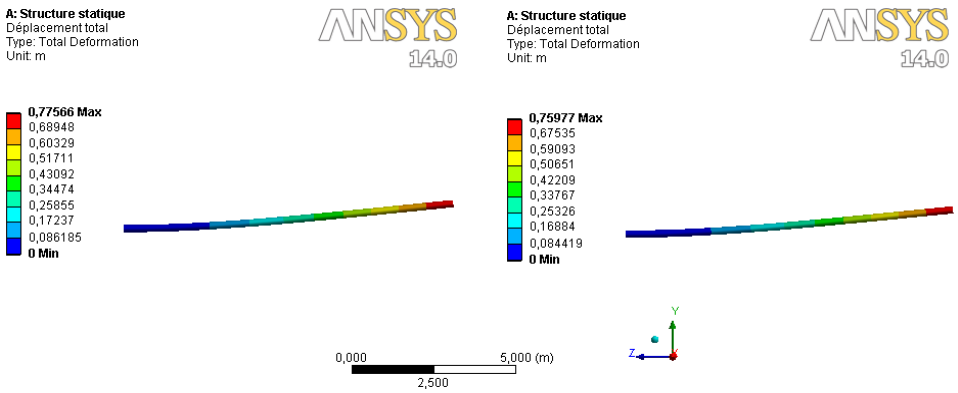
The numerical study considered in this section an example which consists of a wing 3D coupled with airflow. This application aims to illustrate the methodology proposed in a deterministic analysis. Geometrical and material properties of the coupled system are:

- For the structure: density = 2770Kg/m^3, Young's modulus = 7.1e10Pa, Poisson's ratio = 0.3, Length = 10m
- For the fluid: density = 1.225Kg/m^3, viscosity = 1.6e-5$Kg/(m.s)$, Length = 10m

Total displacements of the wing after airflow with velocity inlet VX = 200 m/s found by ANSYS/FLUENT are:

ANSYS/FLUENT	Total displacement (m)
One-way	0.759767
Two-way	0.77566

Table 1: Total displacement



(a) (b)

Figure 8: Total displacement of the wing (two-way (a) and one-way (b))

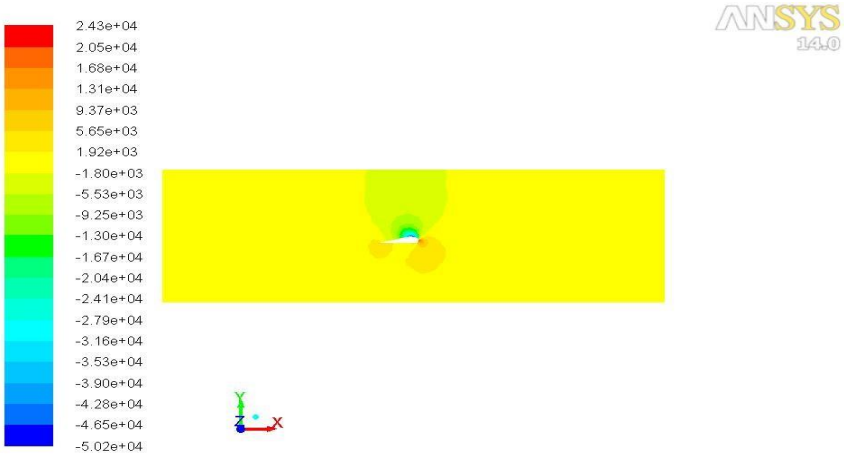


Figure 9: Fluid pressure

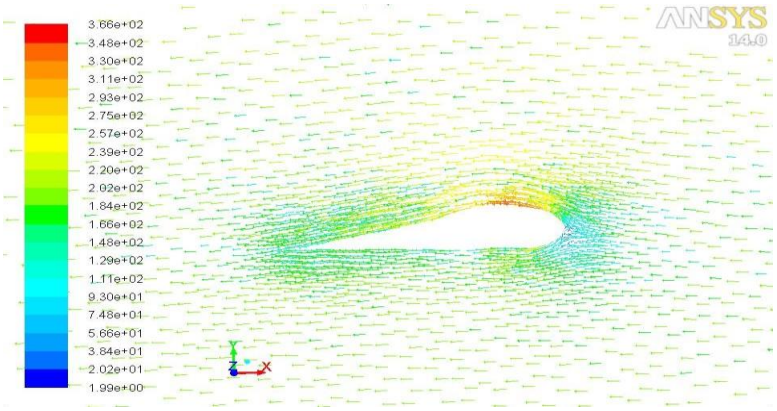


Figure 10: Fluid velocity

Total displacement of the wing after airflow with velocity inlet VX = 200 m/s found by ANSYS/FLOTRAN is:

ANSYS/FLOTRAN	Total displacement (m)
Load transfer	0.662633

Table 2: Total displacement

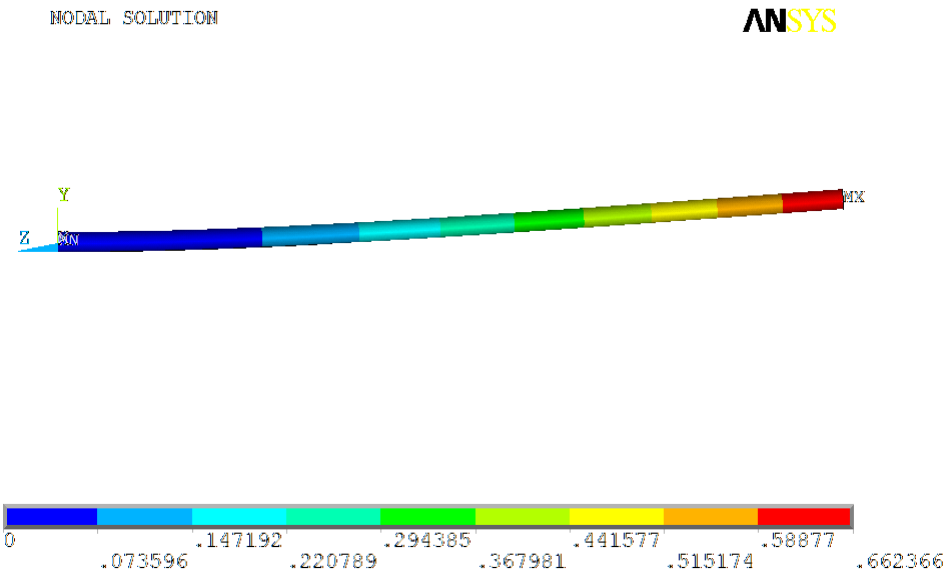


Figure 11: Total displacement of the wing

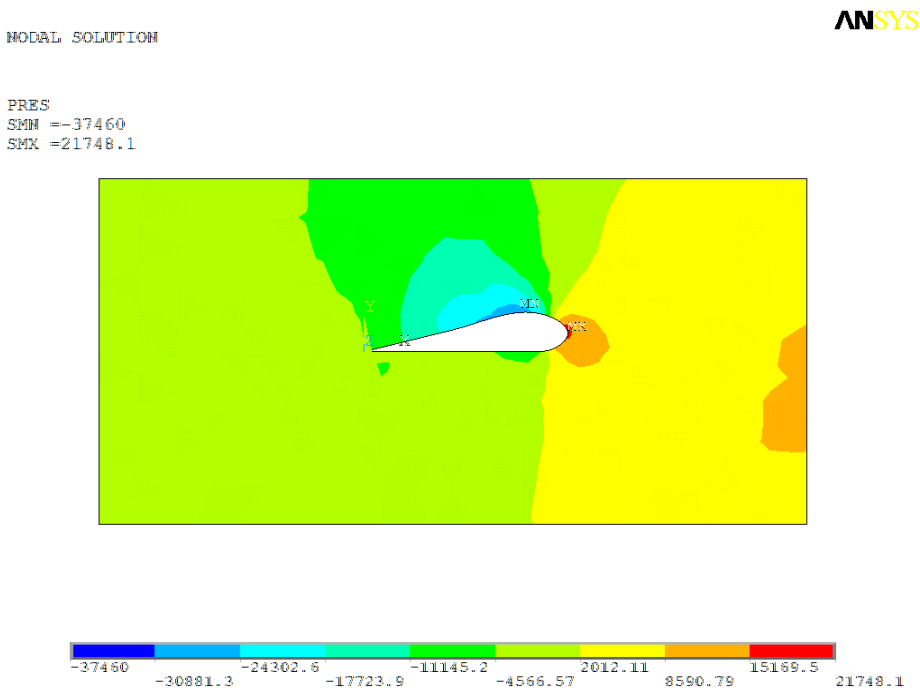


Figure 12: Fluid pressure

5.3 PrestressedModal analysis

The eigenvalue and eigenvector problem needs to be solved for mode-frequency analyses. It has the form of:

$$[K]\{\phi_i\} = \lambda_i[M]\{\phi_i\} \qquad (14)$$

where [K] is structure stiffness matrix, $\{\phi_i\}$ is eigenvector λ_i is eigenvalue and [M] is the structure mass matrix.

For prestressed modal analyses, the [K] matrix includes the stress stiffness matrix [S]. The results of modal analyses with two different cases (ANSYS/FLOTRAN, ANSYS/FLUENT(one-way and two way)) are shown in Table 3. The result includes the first six mode shapes with its respective natural frequency values.

Modes	without airflow	with airflow		
		ANSYS/FLOTRAN	ANSYS/FLUENT	
			One-way	Two-way
F_1	1.4830	12.630	14.640	14.853
F_2	7.3146	19.182	22.016	22.290
F_3	9.2564	51.426	61.008	62.014
F_4	25.738	72.938	82.772	83.766
F_5	29.314	89.873	105.69	107.36
F_6	44.242	126.02	126.34	126.36

Table 3: Natural frequencies of the aircraft's wing

VI. Conclusion

Problems concerning the interaction between fluid and structure were solved on a three-dimensional wing of aircraft in airflow. The fluid was modeled using both two different approximation methods used as computational fluid dynamics (CFD) solvers based on the finite volume methodand finite element method, the structure was modeled using finite element equations and the two approaches were coupled in order to understand the dynamics of the structure, a comparison of the solution between the two methods, one-way and two-way coupling simulation is shown and then a pre-stressed modal analysis has been conducted to determine the natural frequencies and its respective mode shapes.

Our numerical results show little difference between the one-way and the two-way treatment and the FV/FE gives better results than the FE/FE approximations.

References

[1] Z.Yun, Y. Hui, "*Coupled fluid structure flutter analysis of a transonic fan*", Chinese Journal of Aeronautics, vol.24, pages 258-264, 2011.

[2] B.Iwanowski, R.Gladso, M.Lefranc, "*Wave-in-deck load on a jacket platform, CFD derived pressures and non-linear structural response*", 28th OMAE International conference, Hawaii, USA.May 31- June 5, 2009.

[3] Fluent 14.0, ANSYS Inc., South pointe. 275 Technology Drive, Canonsburg, PA15317, USA.
[4] H.Versteeg, W. Malalasekera, "*An introduction to computational fluid dynamics*", Pearson Prentice Hall, 2nd ed., 2007.
[5] ANSYS, "*Complete User's Manual Set: ANSYS Theory Reference 14.0",* ANSYS, INC., Canonsburg, USA, 2012.
[6] R. El Maani, B. Radi, A. El Hami, "*Reliability study of a coupled three dimensional system with uncertain parameters*", Journal of Advanced Materials Research, vol. 1099, pages. 87-93, 2015.
[7] ANSYS Workbench 14.0- User Guide.
[8] A. El Hami, B. Radi, "*Uncertainty and Optimization in Structural Mechanics*", ISTE-Wiley, London, 2013.
[9] A. El Hami, B. Radi, "*Incertitudes, optimisation et fiabilité des structures*", Hermès Lavoisier, Paris, 2013.
[10] A. K. Chopra, "*Dynamics of structures*", Pearson Prentice Hall, 2nd ed., 2001.
[11] F.K. Benra, H.J. Dohmen, J. Pei, S. Schuster, B. Wan, "*A comparison of one-way and two-way coupling methods for numerical analysis of fluid structure interactions*", Journal of Applied Mathematics, vol. 2011, pages 40-56, 2011.
[12] M. Schafer, H. Lange, and M. Heck, "*Implicit Partitioned Fluid-Structure Interaction Coupling*". In Proceedings of 1er Colloque du GDR Interactions Fluide-Structure, pages 31–38, Sophia Antipolis (France), 2005.
[13] F. D. Molina-Aiz, D. L. Valera, A. J. Álvarez, "*Measurement and simulation of climate inside Almería-type greenhouses using Computational Fluid Dynamics*". Agric. Forest Meteorol. vol. 125, pages 33-51, 2004.
[14] V.V. Ranade, "*Computational Flow Modelling for Chemical Reactor Engineering*", Process Systems Engineering. Academic Press, New York, USA, 2002.
[15] O. C. Zienkiewicz, R. L. Taylor, J. Z. Zhu, "*The Finite Element Method: Its Basis and Fundamentals",* Butterworth-Heinemann, Elsevier Ltd, 2013.
[16] C. T Shaw, "*Using a segregated finite element scheme to solve the incompressible Navier-Stokes equations*", Int. J. Numer. Method Fluid, vol. 12, pages 81-92, 1991.
[17] H.-J. Bungartz, M. Schäfer, "*Fluid-Structure Interaction: Modelling, Simulation, Optimisation*", Lecture Notes in Computational Science and Engineering, Springer-Verlag, Berlin, 2006.