RELIABILITY ANALYSIS OF THE AIRCRAFT'S WING USING FINITE ELEMENT/FINITE ELEMENT APPROACHES

R. EL MAANI [1,2]

B. RADI[1]

A. EL HAMI[2]

ABSTRACT

Strong interactions occur between the flow about an aircraft and its structural components, which result in several important aerodynamic phenomena that can significantly influence the performance of aircraft due to the interaction of complex flows with aircraft components especially wings. In this paper, a numerical vibratory study is leaded on a three-dimensional wing of aircraft in air flow. The fluid and the structure are approximated using finite elements method. In this context, it was focused specifically on a deterministic, probabilistic and reliability analysis through numerical simulations.

Keywords: Fluid-Structure interaction, Arbitrary Lagrangian–Eulerian, Finite elements, Finite volume, Reliability.

[1] *LIMII, FST Settat, BP : 577, Route de Casablanca, Settat, Morocco*

[2] *LOFIMS, INSA de Rouen, BP: 8, Avenue de l'Université, 76801 St Etienne de Rouvray,France*

1 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, changesthe boundary condition of a fluid system. This can also happen the other way aroundwhere the structure makes the fluid flow properties to change. This kind of interactionoccurs in many natural phenomena and man-made engineering systems. It becomes acrucial consideration in the design and analysis of various engineering systems. Forinstance, FSI simulations are conducted to avoid flutter on aircraft and turbo-machines[1], to evaluate the environmental loads and dynamic response of offshore structures[3, 2] and in many bio medical applications [4].

The simulations of multi-physics problems have become more important in the fieldof numerical simulations and analyses. In order to solve such interaction problems,structure and fluid models i.e. equations which describe fluid dynamics and structuralmechanics have to be coupled. Although fluid and solid solvers can be used to solvethe respective domains, coupling i.e. interchange of results has been considered as oneof the challenging tasks due to nonlinear nature of the fluid solid interface. But, technicaladvancements in the fields of computational fluid dynamics (CFD), computationalstructural mechanics (CSM) and numerical algorithms have made the numerical FSIanalysis more realistic to be performed in a reasonable time frame.

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

This paper aims to analyze the fluid/structure interaction of a three-dimensionalwing of aircraft in air flow to determine it realistic behavior. The fluids and structureswere modeled independently and exchanged boundary information to obtain aeroelastic solutions. Finite elements discretization was used for modeling fluid and solidequations and the two disciplines were coupled to solve aeroelastic problems. Theloads obtained from the pressures are applied to the original finite element model toobtain the displacements. The fluid's flow is solved using ANSYS Flotran© and coupled with structural code.For the structural model,ANSYS Mechanical© is used in order to understand the dynamicof a structural member and to determine the natural frequencies and its respectivemode shapes.

Computer models are expressed and described with specific numerical and deterministicvalues, material properties are entered using certain values; the geometry of the component is assigned a certain length or width, etc. An analysis based on a given set of specific numbers and values is called a deterministic analysis. Naturally, the results of a deterministic analysis are only as good as the assumptions and input values used for the analysis. The validity of those results depends on how correct the values were for the component under real life conditions. But in reality, every aspect of an analysis model is subjected to scatter. Material's property values are different if one specimen is compared to the next. This kind of scatter is inherent for materials and varies among different material types and material properties.

In the case of this problem, better control of these parameters is thus based on the use of stochastic methods whose main objective is to improve the quality and the reinterpretation of results from simulations.

2 Fluid-structure interaction problem

A general fluid structure interaction problem consists of the description of the fluid and solid fields, appropriate fluid structure interface conditions at the conjoined interface and conditions for the remaining boundaries, respectively. For the presentation in this paper, we restrict ourselves to 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 About the fluid

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 fluidelement is equal to the net rate of flow of mass into a fluid element. Applying thisphysical principle to a fluid model results in a differential equation called continuityequation [5]. The continuity equation for a compressible fluid can be written asfollows:

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

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

Newton's second law states that the rate of change of momentum of a fluid particleequals 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 [5]

$$\frac{\partial(\rho u_i)}{\partial t} + div(\rho u_i u) = -\frac{\partial p}{\partial x} + div(\mu \, grad \, u_i) + S_{Mx} \; (2)$$
$$\frac{\partial(\rho u_j)}{\partial t} + div(\rho u_j u) = -\frac{\partial p}{\partial y} + div(\mu \, grad \, u_j) + S_{My} \; (3)$$
$$\frac{\partial(\rho u_k)}{\partial t} + div(\rho u_k u) = -\frac{\partial p}{\partial z} + div(\mu \, grad \, u_k) + S_{Mz} \; (4)$$

where ρ represents the density , u represents velocity vector, u_i , u_j , u_k are the velocity components in Cartesian coordinate system, μ is the dynamic viscosity and S_M represents the momentum source term. Since the problem at hand does not involve the heat transfer, energy equation is not considered.

2.2 About the structure

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 [13]:

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

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 displacementvector, p is a force vector which is a function of time, the structural damping isnot involved in the finite element model so the above governing equation is modifiedinto 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 offreedom (DOFs). The basic principle behind this method of finding an approximatesolution to the differential equations is to divide the volume of a structure or system into smaller (finite) elements such that infinite number of DOFs is converted to a finitevalue. The sequence of steps involved in solving the equation of motion is as follows[13]:

- Conversion of a structure in to a system of finite elements which are interconnected at the nodes and defining the DOF at these nodes;
- Determination of element stiffness matrix, the element mass matrix, and the element force vector for each element in a mesh with reference to the DOF for the element. The force-displacement relation and inertia force-acceleration relation for each element can be written as:

  $$(fs)_e = k_e u_e \quad (f_I)_e = m_e \ddot{u}_e$$

  where k_e is the element stiffness matrix, m_e is the element mass matrix, u_e and $\ddot{u}_e$ are the displacement and acceleration vector for the element;
- Formation of transformation matrix (Boolean matrix contains zeros and ones) that connects the values of each element in to the global finite element assemblage. It simply locates the elements of k_e , m_e and u_e at the proper places of the global matrices. For

instance the elemental displacements u_e can be related to global matrix u through the following expression;

- Assembling of element matrices to evaluate the global stiffness, mass matrices and applied force vector for the final assemblage

$$k = A_{e=1}^{N} k_e \ (7)$$
$$m = A_{e=1}^{N} m_e \ (8)$$
$$f(t) = A_{e=1}^{N} f_e(t) \ (9)$$

where A is an operator responsible for assembly process. According to the transformation matrix a_e, the element mass matrix, element stiffness matrix and element force vector are placed in the respective global matrices and the arrangement is based on the number of an each element $e=1$ to N_e, where N_e is the number of elements:

$$u_e = a_e u \ (10)$$

- The final equation of motion with the global matrices is formulated as in the form of basic governing equation. This equation can be solved for u(t) using an appropriate iteration schemes which gives the response of system in term of nodal displacement values.

Apart from this sequence of steps, the values of element mass matrix k_e, element stiffness matrix m_e and element force vector $f_e(t)$ are determined by a function called element shape function or interpolation function.

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

$$\sigma^f.n = \sigma^s.n \quad \forall x \in \Gamma^{fsi} \ (11)$$

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

$$u.n = \frac{\partial d}{\partial t}.n \quad \forall x \in \Gamma^{fsi} \ (12)$$

3 Numerical discretization

The numerical computation is developed in two steps. In the first one, the conservationequations are formulated and an approach is adopted to evaluate all the terms. Inthe second one, a segregated, sequential solution algorithm is used to form the elementmatrices, to assemble them and to solve the resulting system for each variable separately ϕ [6]. 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 ofa grid, which consists of dividing the solution domain into a finite number of computationalelements [8]. In the second step, each term of the partial differential equationdescribing the flow is written in such a manner that the computer can be programmedto calculate it [9].

3.1 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 $N_j(x_j)$ (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 [16,17]:

$$\phi = \sum_{j=1}^{n} N_j \phi_j \tag{13}$$

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]:

$$\left[A_e^{transient}\right] + \left[A_e^{advection}\right] + \left[A_e^{diffusion}\right] \quad \phi_e \;=\; S_e^{\phi} \tag{14}$$

Galerkin's method of weighted residuals is used to form the element integrals [6]. 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 [6].

4 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.

For this and a number of additional reasons we prefer to use a partitioned approach.The trade-off is an incomplete tangent stiffness for the overall problem. The consequencesare discussed in the following section.

For the fluid-structure coupling an implicit partitioned approach is employed [14], in Figure 1 a schematic view of the iteration process, which is performed for each timestep, is given. After the initializations the flow field is determined in the actual flowgeometry. From this the friction and pressure forces on the interacting walls are computed,which are passed to the structural solver as boundary conditions. The structuralsolver computes the deformations, with which then the fluid mesh is modified, beforethe flow solver is started again.

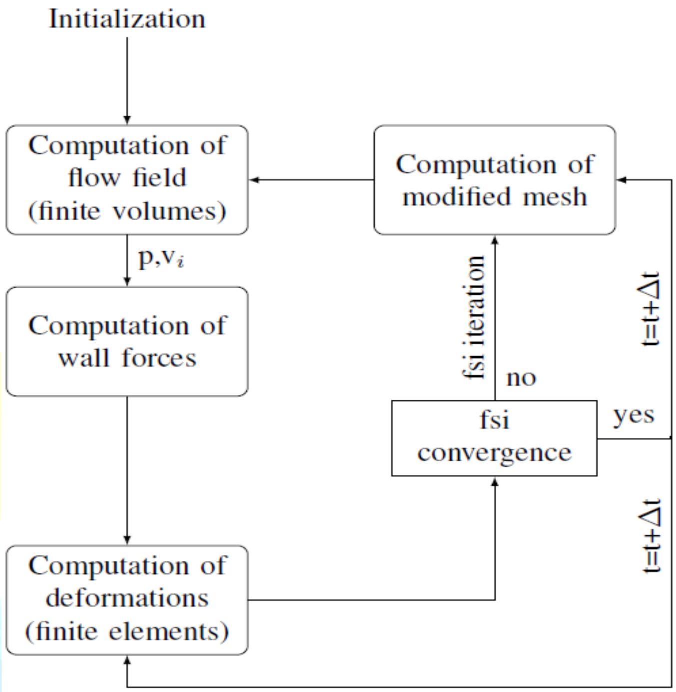

Figure 1: Coupled solution procedure

5 Numerical simulations

Initially, geometric models of both fluid and solid domains are created with appropriate dimensions. ANSYS© is used as a preprocessor for creating and meshing the geometries models. 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 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.

For the finite elements calculation: SOLID186 is used for the 3-D modeling 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 2

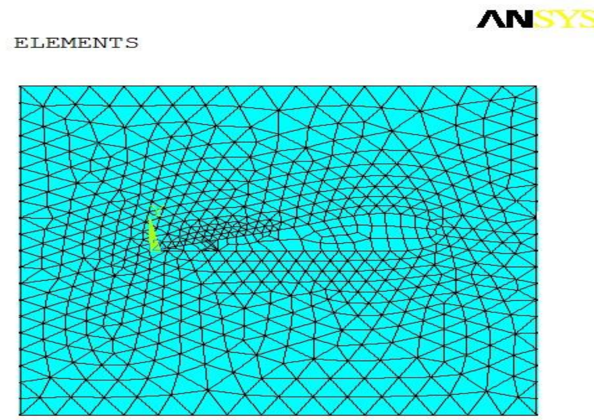

Figure 2: Computational mesh of the two domains

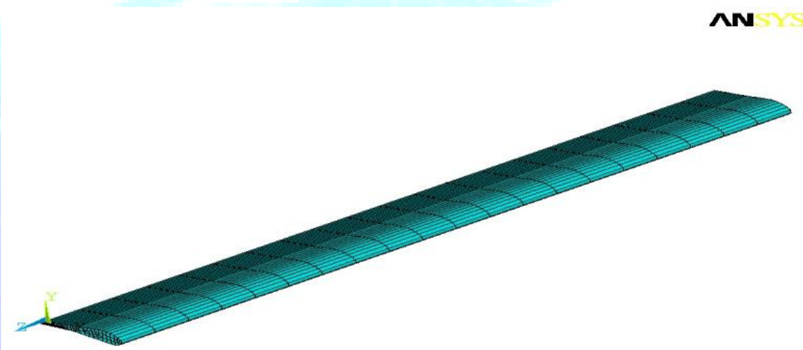

Figure 3: Computational mesh of the wing

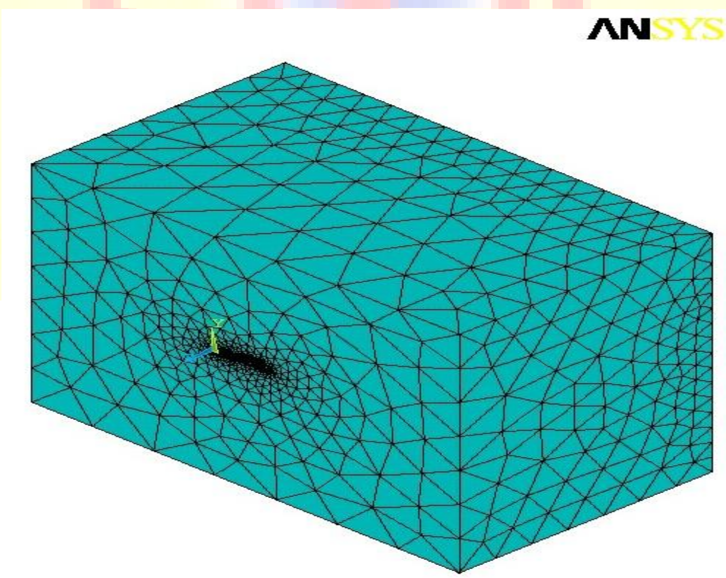

Figure 4: Computational mesh of the fluid domain

5.1 Results

We begin by the validation of our fluid-structure interaction in the deterministic case. The numerical study considered in this section an example which consists of a wing 3D coupled with air flow. 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 = 2770 Kg/m^3; Young's modulus = 7.1 e10 Pa; Poisson's ratio = 0.3; Length = 10 m
- For the fluid: density = 1.225 Kg/m^3; viscosity = 1.6e-5 Kg/(m.s); Length = 10m

Total displacement of the wing with fluid's velocity inlet VX = 200 m/s:

ANSYS/FLOTRAN	Total displacement(m)
Load transfer	0.662633

Table 1: Total displacement

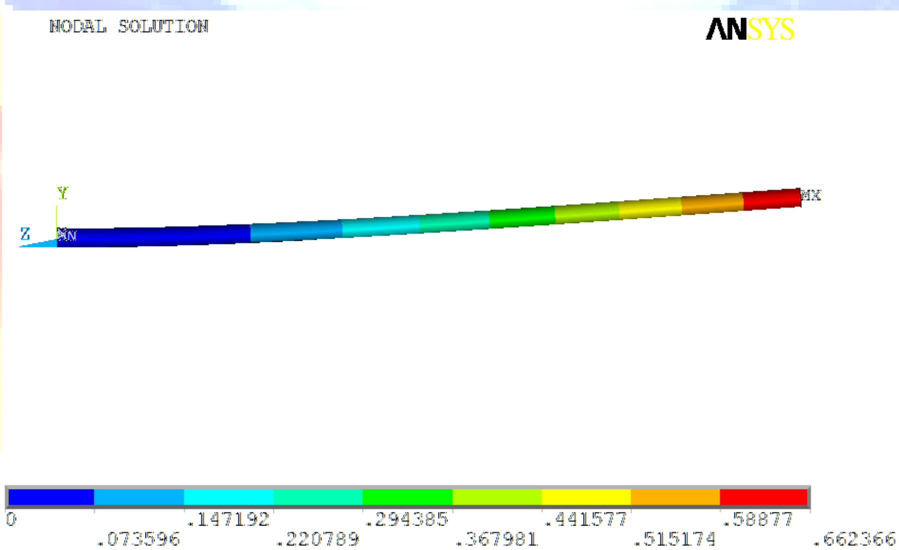

Figure 5: Total displacement of the structure

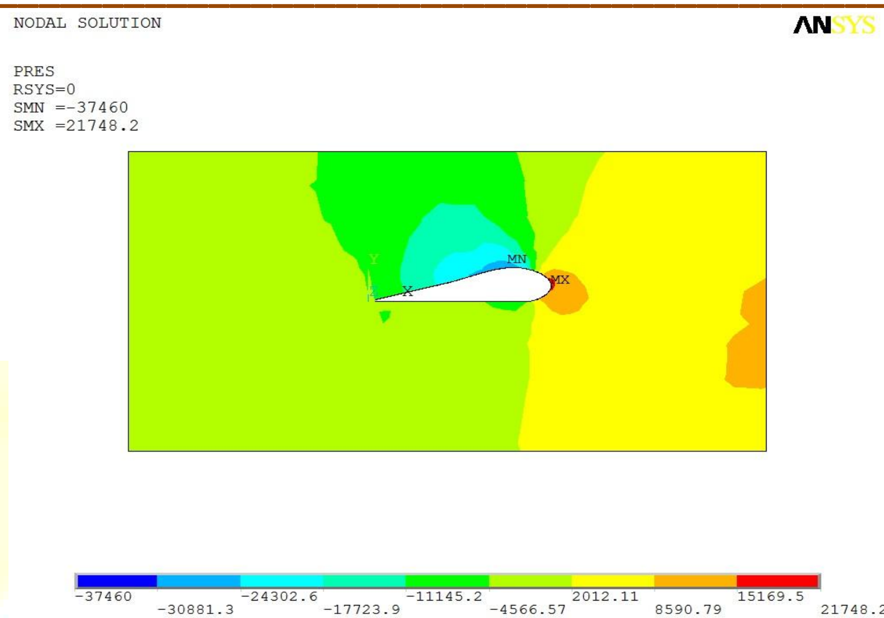

Figure 6: Fluid pressure

5.2 Modal analysis

The results of modal analyses are shown in Table 2. The result includes the first six mode shapes with its respective natural frequency values.

Mode shapes	Natural frequencies(ω_n)
F1	12.630
F2	19.182
F3	51.426
F4	72.938
F5	89.873
F6	126.02

Table 2: The natural frequencies for the aircraft's wing

5.2.1 Probabilistic study

Naturally, the results of a deterministic analysis are only as good as the assumptions and input values used for the analysis. The validity of those results depends on how correct the values were for the component under real life conditions. In reality, every aspect of an analysis model is subjected to scatter (in other words, is uncertain in some way). Material property values are different if one specimen is compared to the next.This kind of scatter is inherent for materials and varies among different material typesand material properties.

It is neither physically possible nor financially feasible to eliminate the scatter of input parameters completely. The reduction of scatter is typically associated withhigher costs either through better and more precise manufacturing methods and processesor increased efforts in quality control; hence, accepting the existence of scatterand dealing with it rather than trying to eliminate it makes products more affordableand production of those products more cost-effective.

A probabilistic study is used to determine the effect of one or more variables on theoutcome of the analysis and Front of the complexity of the problem, we have chosento consider in this work only the sources of uncertainties related to the material properties,but the uncertainties regarding the other elements of the structure (geometry,boundary conditions and mechanical behavior) have not been taken into account in aperspective of simplification.

Table 3 contains the means of random variables and their standard deviations usedin this study and the distributions laws chosen.

Parameters	Means	Standard deviation(SD)	Distribution
Young's Modulus(Pa)	7.1e10	0.355e10	Gaussian(μ,σ)
Density(Kg.m^{-3})	2770	83.1	Gaussian(μ,σ)

Table 3: Moments of the parameters of the problem and Distribution laws

In this context the stochastic calculation was carried out using probabilistic design system of the ANSYS code. This tool is based on a calculation with Monte Carlo simulation (MC) for 100 samples and the response surface method (RSM) for 9 samples.

Table 4 shows the means and standard deviations of the natural frequencies of the wing with air flow.

Modes	Deterministic case	MC	RSM
F_1	12.630	12.540	12.540
F_2	19.182	19.023	19.022
F_3	51.426	50.812	50.821
F_4	72.938	72.290	72.312
F_5	89.837	88.882	88.900
F_6	126.02	125.351	124.28

Table 4: Natural frequencies for the wing

5.2.2 Reliability study

Reliability is usually a big concern because product or component failures have significant financial consequences (costs of repair, replacement, warranty,or penalties), worse, a failure can result in injury or loss of life. Although perfection is neither physically possible nor financially feasible, probabilistic analysis helps you to design safe and reliable products while avoiding costly over-design and conserve manufacturing resources (machining accuracy, efforts in quality control, and so on). In a quality product, the customer rarely receives unexpected and unpleasant events where the product or one of its components fails to perform as expected. By nature, those rare "failure" events are driven by uncertainties in the design.

In this numerical study, The proposed reliability analysis methods employed to compute the probabilities of failure are the first and second-order reliability methods (FORM and SORM) [11, 12]. The analysis was based on an implicit limit state function G based on the first natural frequency F_1 of the coupled system limited by F_0:

$$G(E, \rho) = F_1 - F_0 \text{ with } F_0 = 11.8\text{Hz} \qquad (15)$$

Table 5 summarizes the design parameters and their statistical moments considered in the coupled wing for this example, and it illustrates the results obtained from FORM and SORM approaches.

Parameters	FORM	SORM
Young'smodulus(Pa)	7.793e10	7.7927e10
Density(Kg/m^3)	3007.1	3007.2
Reliabilityindexβ	3.4575	3.5126
ProbabilityP_f	0.00027258	0.0002219
Reliability	99.973	99.978

Table5:Random parameters and their statistical moments

6 Conclusion

In this paper a numerical vibratory study is leaded on a three-dimensional wing of aircraft in air flow and a deterministic, probabilistic and reliability analysis are proposed with fluid-structure interaction problems. Reliability analysis was based on FORM and SORM approaches that take into account uncertainties related to parameters such as properties of the structure. The numerical study is performed using a code developed which couples MATLAB© and ANSYS© to evaluate the reliability of the structure.

References

[1] Yun Z., Hui Y., 2011, Coupled fluid structure flutter analysis of a transonic fa, ChineseJournal of Aeronautics, 24,pp 258-264.
[2] Iwanowski B., Grigorian H., and Scherf I., 2002, Subsidence of the Ekofisk Platforms: Wave in Deck Impact Study. Various Wave Models and Computational Methods.In Proceedings, 21-st International OMAE Conference, Oslo, Norway.Paper OMAE2002-28063.
[3] Anagnostopoulos S.A, 1982, Dynamic response of offshore platforms to extremewaves including fluid structure interaction, Eng.Struct.,4, pp 179-185.
[4] Dubini G., Pietrabissa R., Montevecchi F., 1985, Fluid structure interaction problemsin bio-fluid mechanics: a numerical study of the motion of an isolated particlefreely suspended in channel flow, Med.Eng.Phys, 17, pp 609-617.
[5] Versteeg H., Malalasekra W., (2007), An introduction to computational fluid dynamics,Pearson Prentice Hall, 2nd ed.
[6] ANSYS, 2004, Complete Users Manual Set: ANSYS Theory Reference 9.0, ANSYS,INC., Canonsburg, USA.
[7] El Maani R., Radi B., El Hami A., 2015, Reliability study of a coupled three dimensional system with uncertain parameters, Journal of Advanced Materials Research, 1099, pp 87-93.
[8] Ranade V.V, (2002), Computational Flow Modelling for Chemical Reactor Engineering.Process Systems Engineering.Academic Press, New York, USA.
[9] Ashgriz N., Mostaghimi J., 2002, An introduction to computational fluid dynamics.
In: Saleh, J. (Ed.), Fluid Flow Handbook. McGraw-Hill, New York, USA, 24, pp 1-52.
[10] ANSYS Workbench 14.0- user guide.
[11] El Hami A., Radi B., (2011), Fiabilité et optimisation des systèmes, Technosup, Ellipses.
[12] El Hami A., Radi B.,(2013), Incertitudes, optimisation et fiabilité des structures, Hermès.
[13] Chopra A.K.,(2001), Dynamics of structures, Pearson Prentice Hall, 2nd ed.
[14] Schafer M., Lange H. and Heck M., 2005, Implicit Partitioned Fluid-Structure Interaction Coupling. In Proceedings of 1er Colloque du GDR Interactions Fluide-Structure, pages 3138, Sophia Antipolis (France).