A General Partitioned Fluid-Structure Interaction Model for Non Matching Unstructured Mesh

J. HAMBURGER[a], A. KUMAR[b], S. KRISHNAMURTHY[b]

a. *Fluidyn France SARL, 7, Bd de la libération, 93200 PARIS, FRANCE*
b.Fluidyn India, 146 ,Ring Road, Sector 5, H.S.R. Layout, BANGALORE - 560 102, INDIA

Résumé :

Les solutions disponibles pour l'analyse et la simulation 3D de phénomènes d'interactions fluide-structure appliqués sont généralement : soit trop lourdes et complexes en termes de résolution soit plutôt difficile à gérer pour des solutions découplées utilisant deux codes de calcul séparés et communiquant entre eux. Dans le présent article, une solution faible-forte est présentée, dans laquelle la déformation dynamique de la structure est résolue par une méthode éléments finis, et la solution de l'écoulement fluide par une méthode volumes finis. Les solutions sont intégrées et optimisées à partir d'une interface unique. A partir de ce modèle éprouvé, des possibilités de calcul sur des cas industriels sont rendus plus aisés pour l'ingénieur en mécanique, tant pour des interactions fluides structures à interfaces séparées, que pour des cas d'interactions volumiques (milieux poreux). Deux applications typiques de validation sont présentées en interaction fluide structure (FSI) et en transferts thermique conjugués (CHT).

Abstract :

3D direct simulations in the field of applied industrial fluid-structure interaction problems are often either to heavy to solve (computational resources for e.g. fully coupled methods, matching single solution with different physical phenomena…), either difficult to manage when the coupling is performed externally between two independent codes. Present article exposes an integrated (interfaced) method which couples finite volume methods and finite element methods for respectively fluid flow analysis and structure (stress or thermal) analysis. Coupling operations are performed with inhouse specific methods, which allow flexibility in coupling and is kept open for advanced coupling methods through user-coding. With this practical and accurate tool, many industrial problems involving fluid-structure interaction can be solved by mechanical engineers. Three specific applications of such coupling are presented.

Mots clefs : Mechanical engineering, numerical simulation, Fluid-Structure interaction, Conjugate Heat Transfer, Finite volumes, Finite elements

1 Introduction

Fluid-structure interaction studies are now increasingly being used in many areas. Examples include response of aircraft structures to wind loads, failure analysis of walls of process equipments subjected to transient flow fields, interaction of flow and heat transfer through porous heat exchangers and filters, interaction of electric fields and multi-phase flow fields in electrolysis, coupling of magnetic fields and flow of conducting fluids through cooling channels. Many situations of fluid-structure interaction involve process with a wide range of length and time scales. Hence it is very useful to have a solution method, which is flexible enough to allow the treatment of individual processes according to their characteristic length and time scales and at the same time is accurate and sufficiently coupled. In general two approaches are used in fluid-structure interaction modelling: (a) monolithic and (b) partitioned. A review and comparison of different fluid-structure interaction methods are given by Loon et al [6]. In this paper we describe a general three dimensional flexible partitioned fluid-structure interaction solver, Fluidyn-MP, developed to study interaction of flow, stress, temperature, electric and magnetic fields. Section 2 describes different aspects of

the model and Section 3 describes three case studies to illustrate the use of this model.

2 Fluid-Structure Interaction Model

The computational model includes the equations for multiple fluid flows through porous media, heat and electric conduction and displacements in porous and solid structures, and the additional conditions at the solid-fluid interfaces.

2.1 Fluid Flow Solvers

Fluid flow solvers solve three dimensional unsteady or steady Navier-Stokes equations along with conservation equations for e.g. electric, magnetic and any additional user specified scalar fields and specific models for porous media, gaseous and surface reactions and two phase flows. They are based on finite volume methods and are implemented for both the unstructured and multi-block structured mesh. The equations are solved in both steady and transient mode, with the latter using both explicit and implicit methods. All the variables are computed at the element centres and are interpolated on to the nodes to exchange them with the structural solvers. The convective flux calculations are modified to include the mesh movement due to the domain deformation resulting from structural displacement.

2.2 Structural Solvers

The stress analysis solver uses convective coordinate approach. The use of convective coordinate approach and incremental strains enables handling problems involving large displacements. The boundary conditions include specified nodal forces, nodal displacements, pressures, body forces and nodal temperatures. The external loading may be constant or time dependent (specified as a piecewise linear function of time). Temperature distribution in the solid is obtained by solving the steady or unsteady heat conduction equation using finite element method. The thermal boundary conditions include specified boundary temperature, boundary heat flux, element/nodal heat generation, convective and radiation boundary conditions. If there is a specified convective surface present as a fluid boundary to the solid then the convective flux that enters the solid from the fluid according to Newton's law of cooling is calculated using the local heat transfer coefficient and the fluid temperature. Electric and magnetic fields in the structure are also calculated in similar way.

2.3 Fluid-Structure Coupling

In FSI applications, pressure load at fluid boundary faces at F-S interface is transferred to the corresponding structural boundary faces. The pressure load is converted to nodal forces at structural nodes. The nodal displacements calculated from these external nodal forces are transferred back to the fluid nodes. In CHT(Conjugate Heat Transfer) applications, the heat flux (in the form of the film coefficient and temperature) at fluid boundary faces is transferred to the corresponding structural faces. The face-centred flux value is transferred to structural nodes for thermal calculation. In the thermal solver, the face-centred flux value is used to compute the nodal heat values for thermal conduction calculation. The resulting nodal temperatures are passed back to the fluid nodes as boundary temperatures. The above coupling scheme is exact in case of matching mesh (node-node and face-face matching).

To facilitate the use of non-matching F-S interfaces another coupling method is used [2,3]. The evaluation of loads on the structure induced by the fluid and the displacement of the fluid mesh induced by the structural motion must satisfy the requirement of conservation of work and accuracy. The principle of virtual work is used to ensure conservation of energy. This coupling scheme involves converting face-based values on the fluid boundary face to values on the fluid nodes and using an interpolation matrix to transform fluid nodal values to structural nodal values. Considering a general non-matching interface, this matrix is a non-square matrix. The displacement of CFD grid x_f is expressed in terms of the structural displacements x_s using the transformation matrix $[G]$ as:

$$x_f = [G]\, x_s$$

The requirement of conservation leads to a corresponding matrix for the transformation of forces:

$$f_s^T\, dx_s = f_f^T\, dx_f = f_f^T\, [G]\, dx_s$$

$$f_s = [G]^T f_f$$

The coupling for conjugate heat transfer between solid and fluid is done through an exchange of boundary conditions at the F-S interfaces of interest. Like in the case of pressure, the flux and temperature exchange across possibly non-matching F-S interface is done using a suitable transformation matrix. Conservation of heat flux condition ensures accurate exchange of heat flux across the boundary.

2.4 Remeshing

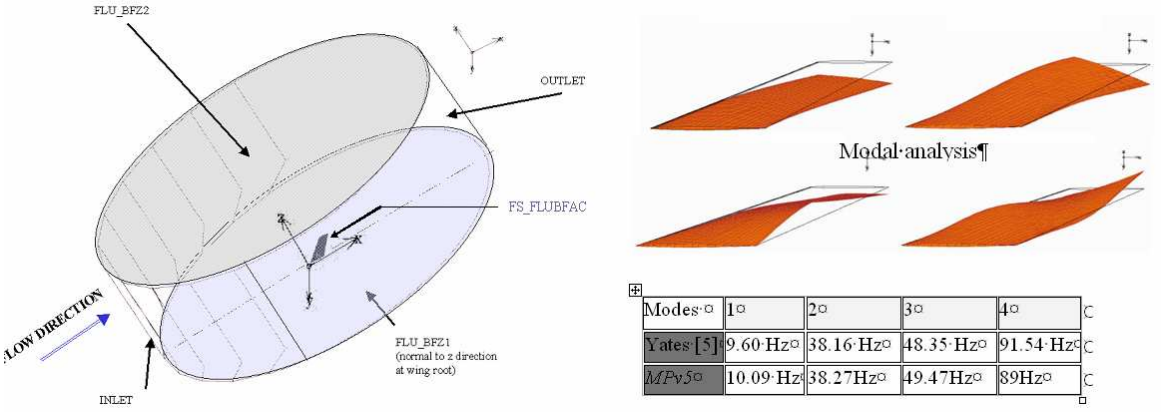
Fluid nodes may be classified as: (1) interior nodes (interior to fluid domain), (2) boundary nodes at the F-S interface, and (3) boundary nodes not at the F-S interface. During remeshing the interior nodes may be moved anywhere within the fluid domain, the boundary nodes at the F-S interface follow the corresponding structural nodes, and the boundary nodes, which are not at the F-S interface, may be modified, subject to the condition that the fluid domain boundary may not change.

3 Case Studies

3.1 Fluid-Structure Interaction: Deformation of Structure

3.1.1 Flutter Motion of Agard 445.6 Wing

The implemented method has been used to study the fluttering of a wing with symmetrical profile (see Kumar et al [4]). This is a problem where the flow and structure interacted at a surface and in which conformal mesh between fluid and structure is maintained (relatively small overall mesh sizes). Mahogany wood wing is considered, with orthotropic properties. Deformation modes are computed at first as described in figure 1.



Modes	1	2	3	4
Yates [5]	9.60 Hz	38.16 Hz	48.35 Hz	91.54 Hz
MPv5	10.09 Hz	38.27Hz	49.47Hz	89Hz

FIG. 1 – Computational domain and structural mode computation of the clamped Mahogany wood wing of AGARD445.6 Wing.

After that, fluid-structure computations are performed for different Mach numbers (M= 0.64, M=0.92, M=1.14). For each Mach conditions, parametric studies are performed, by ranging the flight conditions in order to capture transition from stable (damped) to unstable (flutter) motions. Transition is visible through variation of the dimensionless flutter velocity, defined as:

$$V_f = \frac{V}{b_S \omega_\alpha \sqrt{\mu}}$$

where V is the free-stream velocity, b_s wing chord at root, ω_a frequency of the first uncoupled torsion mode, $\mu = m/\rho v$ mass ratio, with m mass of the wing, ρ density and v volume related to the wing geometry, for each Mach number. The reduced frequency ω / ω_a is also reported (the frequency of the structural modes are affected by the presence of the fluid). Note that the dimensionless flutter frequency is proportional to the square root of the dynamic pressure. Physically, an increment of dimensionless velocity corresponds to an increment of dynamic pressure. When the dynamic pressure exceeds the flutter limit, at a given Mach

number, the aeroelastic system becomes unstable.

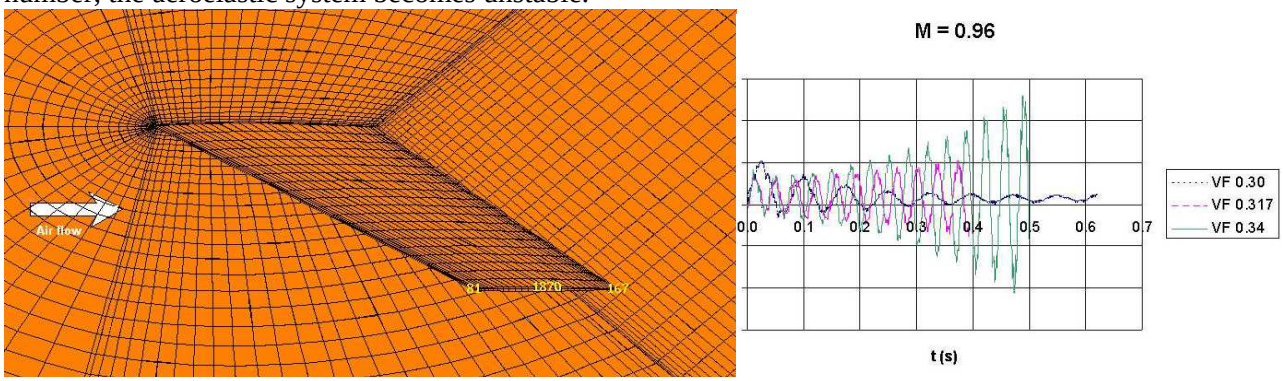


FIG. 2 – Trace point of the structure at tip of the wing, and displacement record for different V_f values

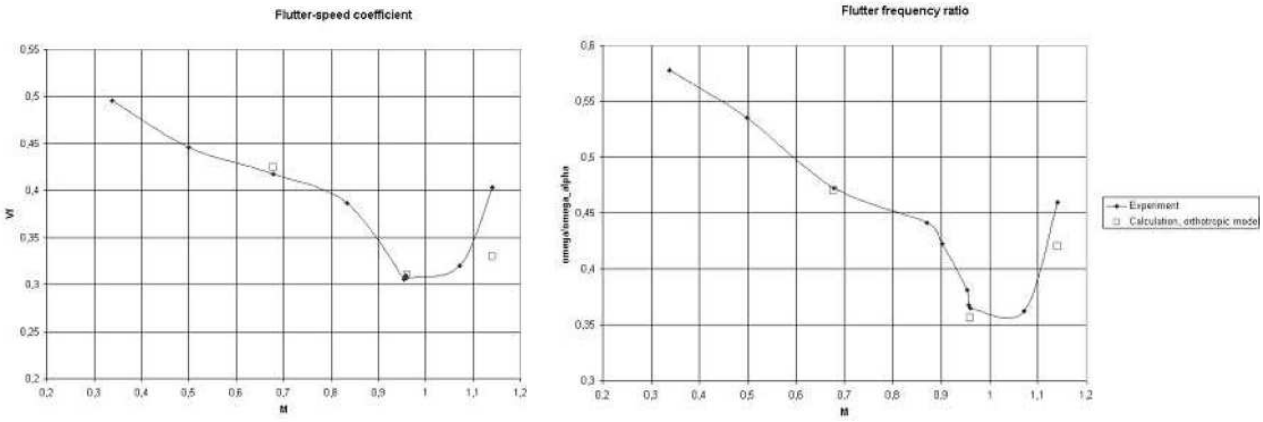


FIG. 3 – Fluid flow and stability transition limits obtained compared to [5]

The fluid solver used is NSNT, available in Fluidyn-MP solver library, which is a segregated iterative collocated pressure based finite volume method for solution of fluid flows at all speeds (including supersonic flows). A simple “Euler-type” flow is computed with optimized mesh refinement along and perpendicular to the wing.

For each simulation, the time-history of the displacement at the structural node located at wing-tip (see Figure 2), on the trailing edge, was recorded. This node was chosen because it is the locus of the major displacement for all of the first four modes, which ensures the possibility to detect the displacements associated to these modes. V_f is chosen first as to be in the stable domain (down wards), and progressively upgraded (parametrically), by increasing local ambient pressure value (flight static pressure). As seen in Figure 2 (right), limit is reached when the local maximum displacement of the wing tip results in a sustained signal (limit between damped signal and amplified signal).

The results obtained by the direct time-domain simulations are in good agreement with the experimental data, in terms of dimensionless velocity transition limits (figure 3) and validate FSI procedure for such flows. For values of Mach number below Mach=1 (subsonic and transonic flows), transition to flutter motion is well predicted while for M>1, a small discrepancy appears, which indicates that the compressible effects induce some different behaviour of the wing, due to predominant shocks-waves over the wing profile.

This module has been proven to work after a parametric study on mesh refinement around the wing, time-steps to be used in transient, and also on performance of CFD solvers. This module is being used for more

complex aeroelastic case studies around aerodynamic bodies.

3.2 Fluid-Structure Interaction: Flow and Heat Transfer in Porous Media

The objective of this study is to validate the volume based F-S interaction for conjugate heat transfer with local thermal non-equilibrium model implemented in Fluidyn-MP with experimental data of Calmidi and Mahajan [1]. In this model, the energy equations are solved separately for the solid and the fluid, and are coupled through the local heat transfer between the solid and the fluid. This model is found to be more appropriate, than the dispersion model (where an effective conductivity is used) when the conductivities of the solid and fluid are highly different or in the presence of large heat generation in the domain. Figure 5 shows the problem domain. It consists of airflow through aluminium metal foam. In the experiment the foam is heated from one end and the temperature variation at the top surface of the sample along the length is noted with the help of thermocouples. The average Nusselt number is calculated using the average of the measured temperatures along the wall. The Nusselt numbers obtained from the computation, for different flow conditions are compared with the above values.

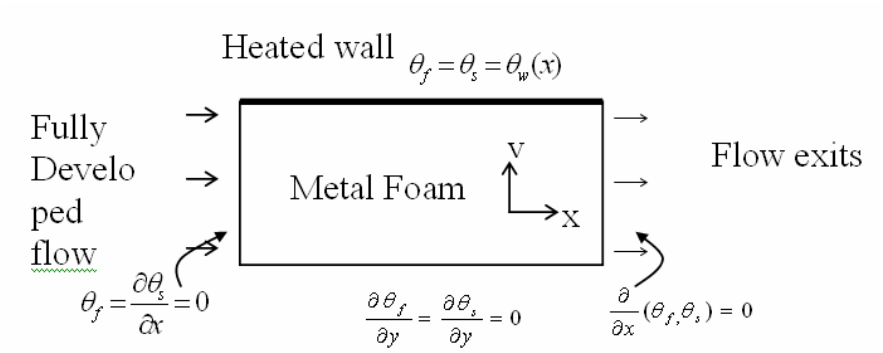


FIG. 5 – Fluid Schematic for the flow through and around a porous structure: Geometry and FSI mesh

In this study, the steady volume-averaged momentum equation that governs fluid flow in porous media is considered. The source term accounts for the pressure drop due to viscous friction at the walls of the metal foam and pressure drop due to the form drag. Former is calculated using Darcy's law and the latter is calculated using Forscheimer model.

Simulations are done for different inlet velocities. Figure 6 shows the experimental and computed Nusselt numbers for different inlet velocities.

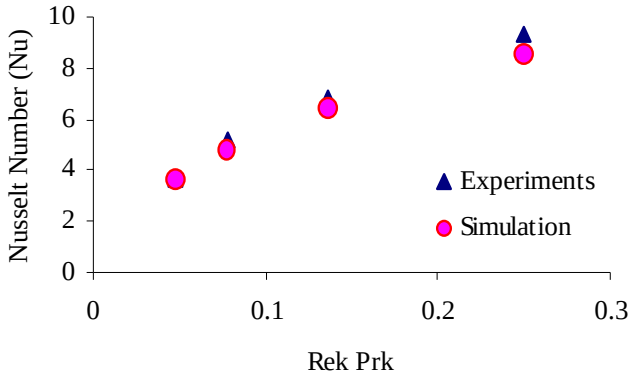


FIG. 6 – Variation of Nusselt number with product of Reynolds and Prandtl numbers (Re Pr).

Comparison is very good at lower velocities. At higher Reynolds numbers (higher velocities) there is a difference of about 8%. This error can be attributed to due to the average boundary wall temperature specified at the heated wall instead of measured variable temperature. Figures 7 show the longitudinal variation of temperature in the fluid and the solid, respectively. It is expected that near the inlet the fluid temperature would be much less than the solid temperature, which is correctly captured by the present model.

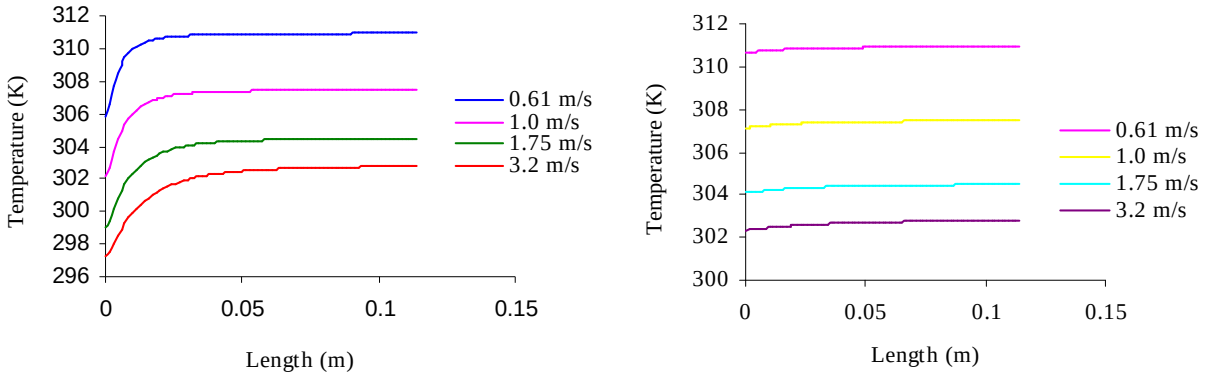


FIG. 7 – Longitudinal variation of temperature in the fluid (left) and solid (right)

References

[1] Calmidi, V.V. & Mahajan, R.L., Forced Convection in High Porosity Metal Foams, *Transactions of ASME*, **122**, pp. 557-565, 2000.

[2] Farhat, C. Lesoinne.M. & LeTallec P, Load and motion transfer algorithms for fluid/structure interaction problems with nonmatching discrete interfaces: momentum and energy conservation, optimal discretization and application to aeroelasticity, Comput. Methods Appl. Mech. Engrg, 157, 95-114, 1998.

[3] Jaiman.R.K., Jiao.X., Geubelle.P.H. & Loth.E, Conservative load transfer along curved fluid-solid interface with non-matching meshes, J.Comput.Physics 218, 372-397, 2006.

[4] Kumar R., Jain P., Ramesh, S. & Hamburger, J., Transonic Flutter Analysis using a Coupled Fluid-Structure Interaction Method, Seventh Asian Computational Fluid Dynamics Conference, 26-30 November, 2007, Bangalore, India.

[5] E. C. Yates, Jr., N. S. Land. , J.T. Foughner, Jr. Measured and calculated subsonic and transonic flutter characteristics of a 45° sweptback wing planform in air and in Freon-12 in the Langley transonic dynamics tunnel. Technical note NASA D-1616, 1963.

[6] Van Loon R, Anderson P. D., van de Vosse F. N. & Sherwin S.J., Comparison of various fluid–structure interaction methods for deformable bodies, Computers and Structures, 85, 833-843, 2007.