F2006M235

MODELLING FSI PROBLEMS IN FLUENT: A GENERAL PURPOSE APPROACH BY MEANS OF UDF PROGRAMMING

Baudille, Riccardo, Biancolini, Marco Evangelos
Department of Mechanical Engineering of University of Rome "Tor Vergata", Italy

KEYWORDS - dedicated FEM solver, FSI, UDF programming, GUI development, code validation, reed valves

ABSTRACT - Fluid-structure interaction (FSI) is an important and interesting phenomenon, but it is a difficult challenge for numerical modelling. However there are several cases in which the interaction between the fluid and adjoining structure governs the physical behaviour of the system. Fluid-structure interaction plays an important role in general problems such as flow-induced vibration, blood flow in vessels and heart valves, wind instruments and sailing. More automotive specific examples are aerodynamic induced deformation, aero noise, tire hydroplaning, air bags deployment, pressure driven engine valves, ducts deformation in high pressure injection systems.
An FSI model has been developed for FLUENT 6.2 using a dedicated FEM solver, coded as a user-defined function (UDF). Arbitrary thin structures are handled by the model, and include non-linear effects, such as the contact between rigid and flexible walls. Special care was taken to manage boundary motion in FLUENT since the contact between moving surfaces is denied by FLUENT's remeshing algorithm. A special solution strategy that leaves the CFD simulation time virtually unaffected by the simultaneous working of the FEM code, is applied. The result is a general-purpose model, capable of solving a variety of problems without the need for significant modifications to the software each time. Some test cases and application examples are presented in order to validate the code and to demonstrate its functionality.

TECHNICAL PAPER - Fluid-structure interaction (FSI) is a very interesting topic and it is the key for resolving many physical problems that cannot be handled separately by a structural or a fluid point of view. However integrated solution of both problems is a difficult challenge especially for numerical modelling where advanced solution strategies and tools are already available. It even poses difficulties for numerical modellers. Structural behaviour is a troublesome boundary condition for the CFD analyst, who prefers to assume that boundaries are rigid. The structural analyst, on the other hand, would like to assume that fluid inside or outside a structure merely generates a constant pressure on the walls. Unfortunately, there are several cases in which neither analyst can make these simplifying assumptions. In such cases, it is the interaction between the fluid and adjoining structure that governs the physical behaviour of the system. In fact some coupled problems can converge in a stable configuration, others can generate an oscillating solution and some others can produce an unstable behaviour.

In order to focus the FSI solution strategies, a brief review of relevant scientific contributions was conducted. In [1] an FSI application is presented adopting a multibody technique for the structural solution, by subdividing a beam in rigid bodies connected by elastic joint, and a kinetic gas-lattice method for the solution of the flow field. Two cases were presented: fluid motion induced by the movement of the structure and structure deformation induced by the flow field. In [2] a deep insight about theoretical topics of FSI is given. 2D FEM solvers were here presented both for fluid and for structure, adopting a weak coupling, i.e. a time marching solution in which structure and fluid are solved separately and exchange information at the end of each time step. Two practical applications were proposed: the flow in a channel with a flexible boundary, suitable for blood motion analysis, and the vortex shedding produced by a cantilever beam in the stream of a bluff body. The last study was further investigated by [3, 4, 5]. In [3] a strong coupling is presented with a FEM monolithic solver developed by the authors. In this approach both problems are formulated and solved simultaneously at each time step. Non linear Lagrangian motion was considered for the structural part while a mixed Lagrangian Eulerian technique was implemented for the fluid. Several applications were presented including the vortex shedding of the cantilever beam. In [6] a strongly coupled numerical analysis of FSI with two immiscible fluid flows is presented using a discretization method based on stabilized space-time finite elements. Numerical results for the collapse of a water column onto an elastic dam are here compared with experimental data in order to validate the numerical method. In [7] both numerical and experimental results are presented for a wing interacting with a flow field. 3D motion of the wing was solved using a FEM approach by means of beam elements based on Timoshenko theory. The fluid motion was solved by 3D CFD solver adopting Euler or Navier-Stokes equations, depending on flux intensity. Strong coupling technique was adopted. In [4] the problem of vortex shedding was revisited with a strong coupled non linear solver, showing how different stable periodic solutions can arise if different initial conditions are imposed. A paper devoted to the general problem of shell structures design in Civil and Mechanical engineering was presented in [5]. Also FSI topics were here covered including vortex shedding of a cantilever, modelled by means of beam elements, and the study of a channel with an internal obstacle represented by a flexible cylindrical shell. In [8] a theoretical study of FSI is presented discussing the main advantages and disadvantages of weak and strong coupling FSI solvers. Furthermore a strong approach is proposed, based on a numerical evaluation of the Jacobian of the complete system of equations obtained by separate solvers. Such approach allows a great reduction in time step, but the numeric effort requested for differentiation leads to similar computation performance. An FSI capability was shown for multiphysics FEMLab solver by COMSOL [9] about a cantilever beam subjected to a transversal flow in which a progressive deflection is observed followed by a sudden release with a vortex generation. In [10] an experimental study on the vortex structures generated by reed petals movement of wind instruments is proposed. This work evidences that in this case the vortexes are produced mainly by the motion of the petal that is not particularly influenced by the fluid dynamics. A FLUENT simulation is also proposed in the same work, where the petal is modelled as a non flexible structure with a rigid motion, just for analysing the vortex structures produced.

In this paper an FSI approach is described. Two detail levels were covered. First a cantilever beam interacting with a surrounding two dimensional flow field is considered according to the theory developed in [11], and Fluent implementation presented in [12]. Then a general approach related to arbitrary thin walled structures interacting with three dimensional flow field is described.

In the case of cantilever beam the structural dynamic problem is quite simple to make it a very good starting point to study FSI problems. Moreover several practical applications can be handled by this model. Among them there are reed valves opening, flag motion, paper sheet motion in paper making, horizontal motions of skyscrapers interacting with wind, interaction between dams and water waves. Cantilever dynamics was in this case handled by a dedicate FEM solver.

The general approach requires the use of an external FEM for the generation of mass and stiffness matrix and allows to handle arbitrary thin walled structures.

For both approaches, special care was taken to manage the interface between structural parts and CFD domain. At each time step, boundary motion is imposed in FLUENT on the basis of dynamic structural solver results. Fluid pressure are given back to the structural solver to drive its dynamic. A special solution strategy is applied that leaves the CFD simulation time almost unaffected by the simultaneous workings of the FEM code. The result is a general-purpose model, capable of solving a variety of problems without the need for significant modifications to the software each time.
In order to test the accuracy of the proposed method, a typical benchmark is first considered [2, 3]. Then more complex applications are studied solving the problem of a water wave interacting with a flexible dam [6]. Finally the practical application of a F1 front wing is proposed.

NUMERICAL MODEL

Structural model

In the case of the cantilever, Euler beam theory [14] has been implemented by means of UDF in order to couple the structural behaviour with the fluid flow. For the case of the thin walled structure a structural mesh conformal to the wetted boundaries is generated and transferred to the FEM solver MSC/Nastran for the computation of discrete mass and stiffness matrix.
In both cases numerical mass, stiffness and damping matrix are adopted to perform a direct numerical integration of the discretized equations of motion.
The dynamic equation of motion in matrix form is the following:

$$[M]\{\ddot{y}(t)\}+[B]\{\dot{y}(t)\}+[K]\{y(t)\}=\{F(t)\} \qquad (1)$$

where [M] is the mass matrix, [B] is the damping matrix, [K] is the stiffness matrix, {F} is the forces vector and {y} is the vertical displacement in the local coordinate system. The Newmark method [15] of central finite difference representation is then applied to displacement derivatives. In the presented model the time step size is fixed, in order to improve computational performances.

CFD Coupling strategy

The coupling technique used is based on a weak approach, that means that the structural problem is solved after the fluid time step is computed. The forces applied to the structure are passed by FLUENT at the end of each time step to the FEM solver. Then the structural problem is solved and the new geometry configuration is applied to the fluid domain by means of MDM capability of FLUENT. The flow chart of the solution process is represented in figure 1. This coupling strategy is very efficient and is able to handle a wide range of FSI problems. Convergence is ensured by choosing the correct time step size for the coupled problem.

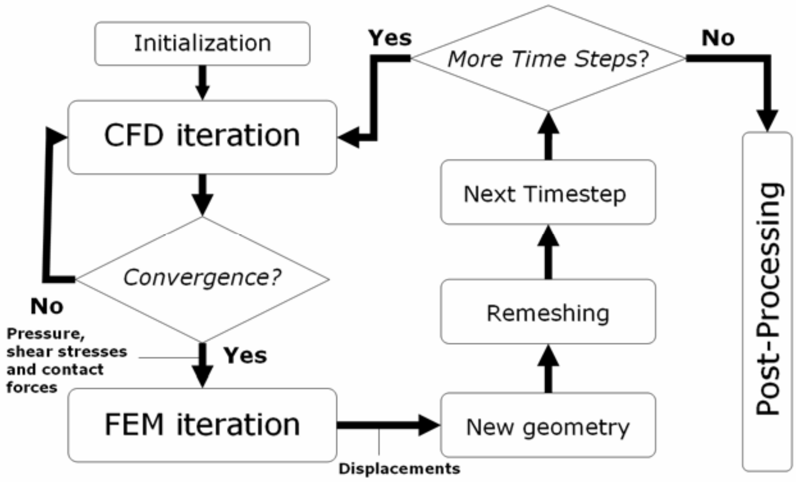

Figure 1: flow chart of the solution process

CFD/FEM interface management

The computational mesh has to be prepared as usual with an appropriate pre-processing tool. No different mesh has to be added to the fluid mesh because the FEM mesh is automatically built by the FSI code from the boundary mesh of the fluid domain. As far as the cantilever problem is concerned, both thin and thick beams are handled by the code. If a zero thickness structure is modelled in the fluid domain only two faces are passed to the FEM model to build the corresponding beam elements. Otherwise in the case of a non zero thickness structure, three faces are passed. A basic requirement for the mesh is that the opposite faces must be aligned and straight, in the initial configuration, and that their nodes must be aligned. This condition allows the code to create a line of nodes in the middle line of the structure and to generate the beam elements between them.

For general thin walled structures only one face is used to generate the structural model but pressure data are recovered summing the contributions of both wetted faces. In this case extra information are required for each face to build the FEM model with proper material and thickness.

In both approaches some pointers are introduced in order to correctly point from FLUENT data structures to the FEM ones and vice versa. These pointers are very important not only to create the FEM mesh but also to allow communication between CFD and FEM: the direct pointers are used to pass forces from CFD to FEM and inverse pointers are used when the FEM solution has to be applied to the CFD boundary nodes. In case of non zero thickness cantilever, not only vertical displacements are considered, but also rotations of the section are taken into account in order to correctly move the CFD boundary nodes.

INVESTIGATED CASES

In this section three applications of the proposed method are illustrated.

Vortex shedding

A well-known benchmark case has been used as an initial test of the model. This case describes the vortex shedding excitation of a cantilever beam positioned downstream of a square obstacle in a laminar flow. From a structural point of view, the deformed shape of the cantilever beam is determined by the pressure load and shear stresses. From a fluid dynamics point of view, the beam constitutes a time-varying boundary condition for the fluid flow. The reference values for the benchmark were found in the works [2, 3]. The geometry is reported in figure 2 where all quotes are expressed in metres.

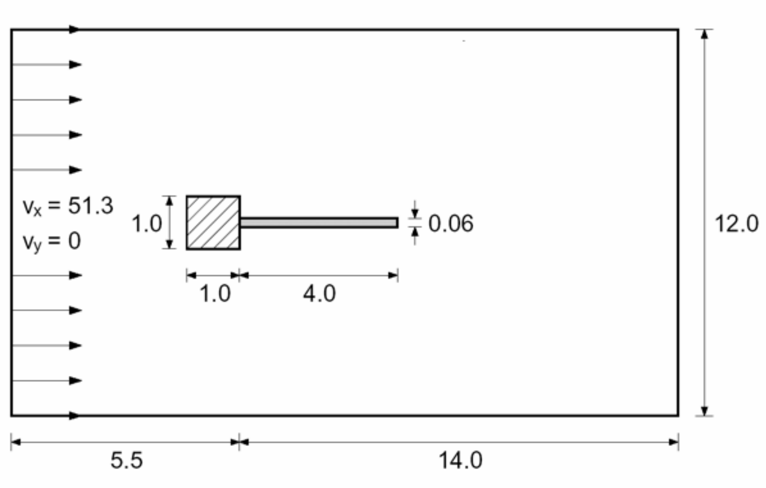

Figure 2: geometry of the test case

The fluid has a density of 1.18 10^{-3} kg/m³ and a viscosity of 1.82 10^{-4} Pa·s. The structure is characterized by a density of 0.1 kg/m³ and a Young modulus of 2.5 10^{6} Pa. The boundary conditions of the test case are a velocity inlet of 51.3 m/s, a constant outlet static pressure, a free sleep condition on lateral walls and a no slip condition on body and beam walls.

<u>Breaking dam</u>

A two-phase problem is investigated in order to test the FSI model in combination with other FLUENT sub-models. A water column is initially placed at the bottom left corner of a tank surrounded by atmospheric air. In the middle of the bottom of the tank there is the flexible dam. Geometry configuration is illustrated in figure 3 where all quotes are expressed in metres. Under the effect of the gravitational acceleration, the water falls down and impinges on the dam, producing its deformation. Free slip boundary condition is applied on all the walls and the top of the tank is considered as an open boundary with a prescribed static pressure condition. The dam is made of rubber with a Young modulus of 1 MPa and density of 1000 kg/m^3. This case has been studied in [6] where water shapes and dam tip displacements are reported for various times, for both numerical and experimental investigations.

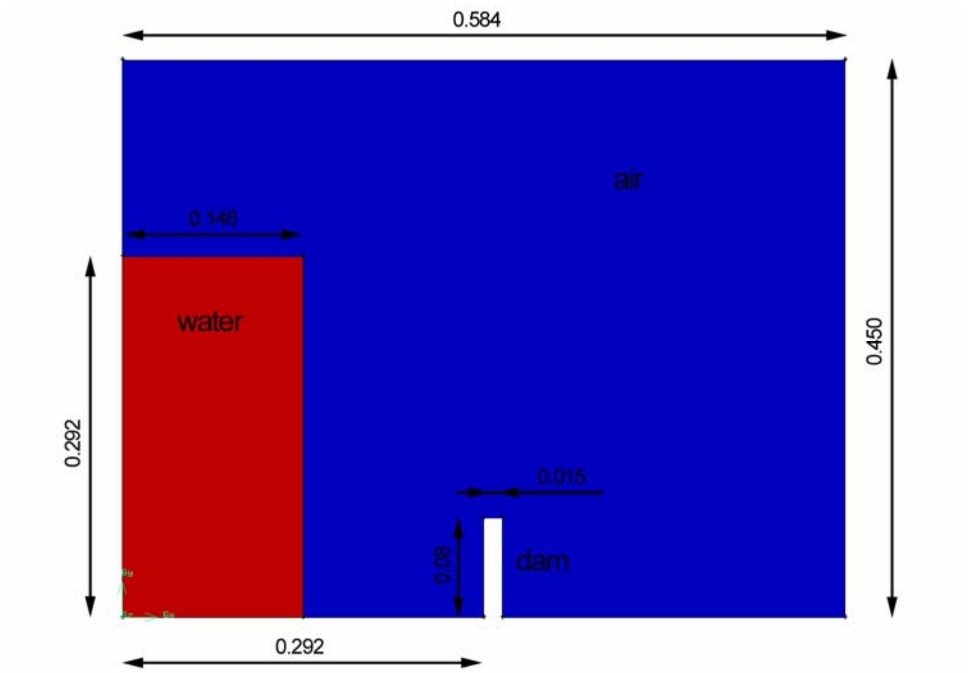

Figure 3: the initial configuration of the water column and the flexible dam

<u>F1 front wing</u>

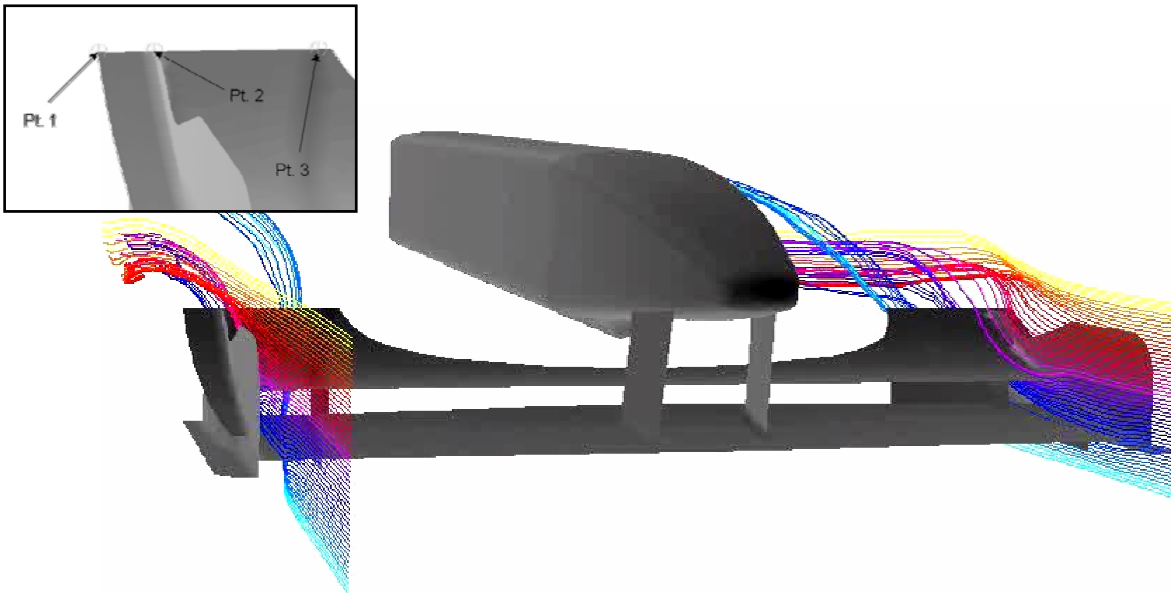

Figure 4: F1 wing model (with stream lines of Case 1).

An academic model of F1 front wing (Figure 4) was adopted to analyse the effect of structure dynamic on aerodynamic performances.
Symmetric boundaries condition was imposed both for CFD and FEM considering only half domain. A standard aerodynamics settings was considered and unsteady solution was considered for two cases. In the first case a constant inlet velocity of 300 kph (starting from a fully developed flow calculated for rigid condition) was prescribed, in the second case a variable inlet velocity from 0 to 300 kph was prescribed according to the vehicle acceleration law reported in Figure 5.

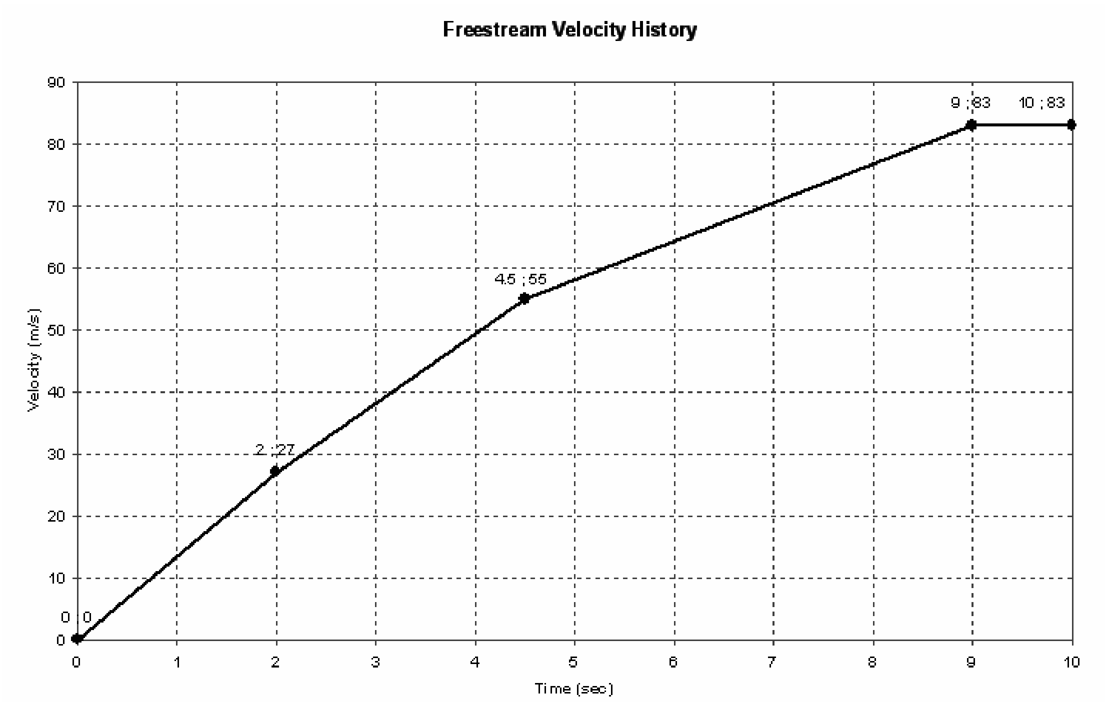

Figure 5: Vehicle speed history.

RESULTS

Vortex shedding

In order to quickly achieve the expected solution of stationary vortex shedding, a steady calculation of five hundred iterations was first performed without the FSI model. The flow field obtained is then applied as initial condition for the unsteady FSI simulation. This procedure shorten the total calculation time, but does not change the final result.

The fluid dynamics problem is laminar, given a Reynolds number of 333. The vortex shedding frequency of the isolated square cylinder is 6.25 Hz, that corresponds to a Strouhal number of 0.12, while the first natural frequency of the cantilever beam is 3.03 Hz. Comparing these frequencies to the resulting frequency of the coupled problem of 3.18 Hz, it is possible to conclude that the coupled behaviour is dominated by the first mode of the cantilever beam.

Animations of the CFD predictions for the pressure and velocity fields around the bluff body and the slender flexible structure are consistent with expectations, as shown in figure 6. A plot of the vertical displacement of the beam free end with respect to time is in good agreement with the literature reference data cited above, as illustrated in figure 7.

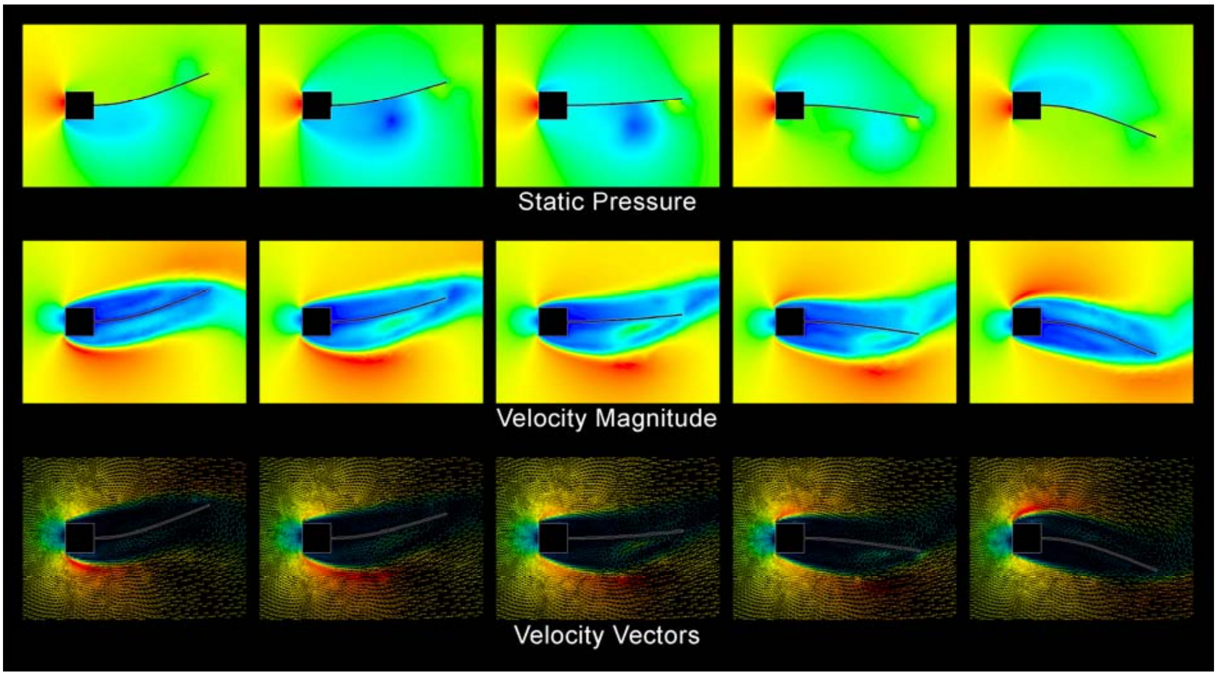

Figure 6: simulated pressure and velocity fields around the square bluff body and the slender flexible structure for different times

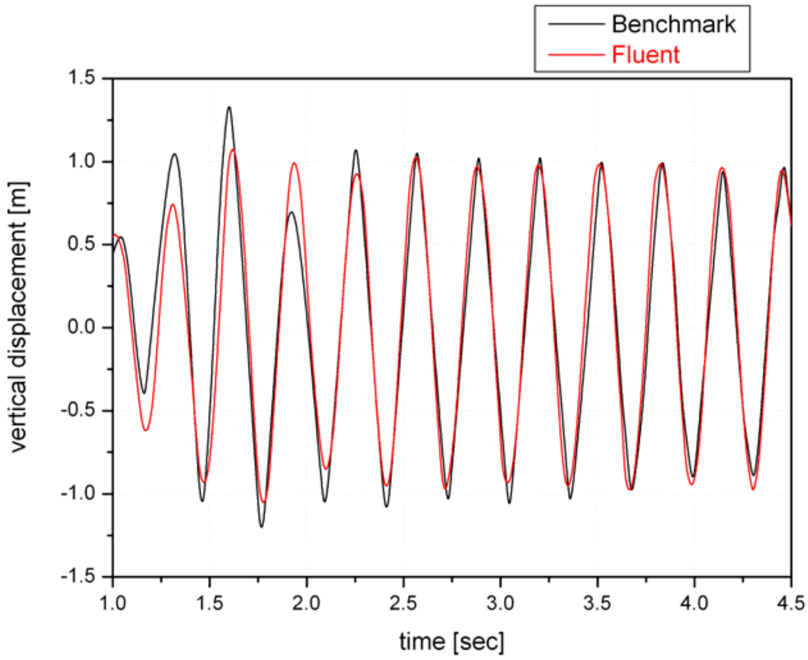

Figure 7: vertical displacement of the beam free end predicted by the FSI model compared to literature reference data [2], [3]

<u>Breaking dam</u>

The simulation of the two-phase problem of the braking dam was performed by means of the FLUENT multiphase model Volume Of Fluid (VOF), with the Geo-Reconstruct scheme, in conjunction with the proposed FSI model. The air fluid was considered as a compressible ideal gas because its compressibility plays a crucial role when the air is trapped by water in a closed volume, as shown in the simulation results. Moreover surface tension and wall adhesion of water are neglected. The FLUENT standard k-ε turbulence model has been used for the fluid dynamics simulation.

Initially the water column begins to fall down and then it reaches the obstacle which starts to be deformed by the water mass impact. During the raise of the water on the left side of the dam the deformation of the structure increases until it reaches its maximum when the water has flown more than one dam length over the tip. Then the tip displacement starts to decrease and it shows a sudden reduction when the water impinges on the right wall of the tank. Subsequently some tip oscillation are observed, when the water starts to cover all the free end of the structure.

Figure 8 illustrates a comparison of the water shape between experimental images and simulation contours. In figure 9 the horizontal displacement of the free end of the dam is plotted versus time, both for numerical and experimental data. All experimental reference data are taken from literature [6].

Comparisons confirm a good agreement between the proposed FSI model results and experimentation. A little lack of accuracy can be explained because of some simplification that have been made: firstly the simulated case has two dimensions and what happens in the third dimension is not taken into account; then the dam is modelled as a cantilever beam and its elasticity behaviour is taken as linear, even if rubber actual behaviour could be far from it; finally the small displacements approach is used even if the deformation of the structure is not small.

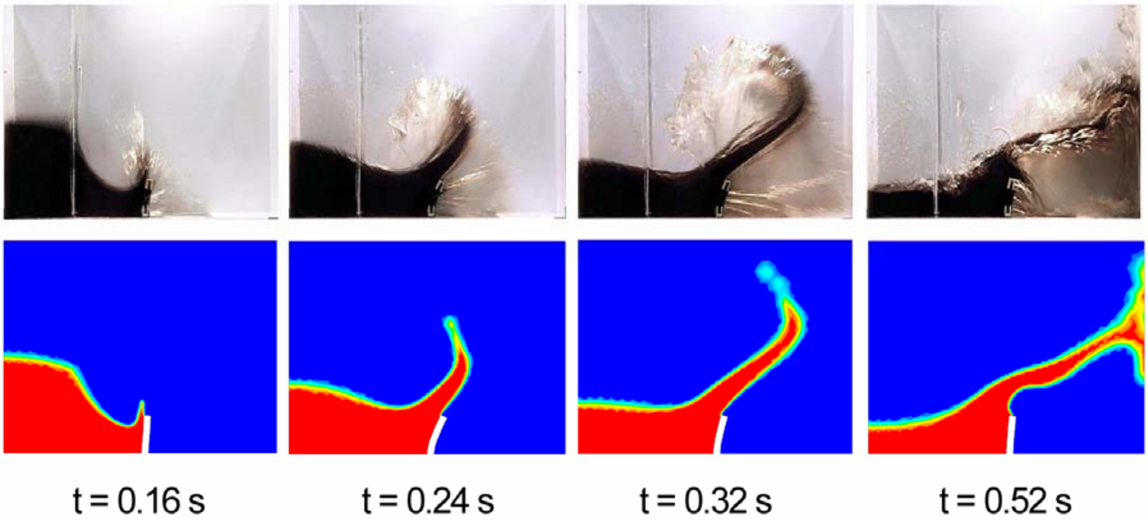

Figure 8: comparison of the water shape between experimentation and simulation

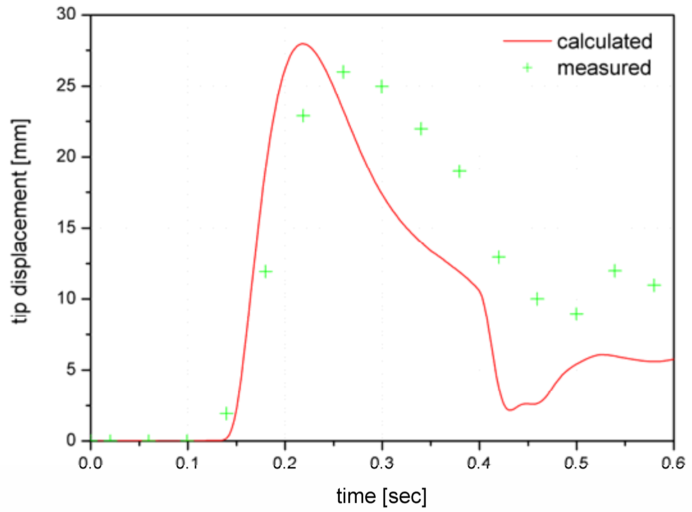

Figure 9: horizontal tip displacement of the dam predicted by the FSI model and measured experimentally by [6]

<u>F1 front wing</u>

For the Case 1, that represent the response of the wing for a prescribed inlet speed of 300 kph, a stable deformation is observed (Figure 10) after a transient of about 0.3 s. A 15 mm deflection is observed in the stabilized configuration, streamlines obtained for the stable configuration are represented in Figure 4. Such transient correspond to an evolution of aerodynamic coefficients (Figure 11) related to drag and lift that in the deflected configuration computed by proposed method results quite different from the rigid wall results.

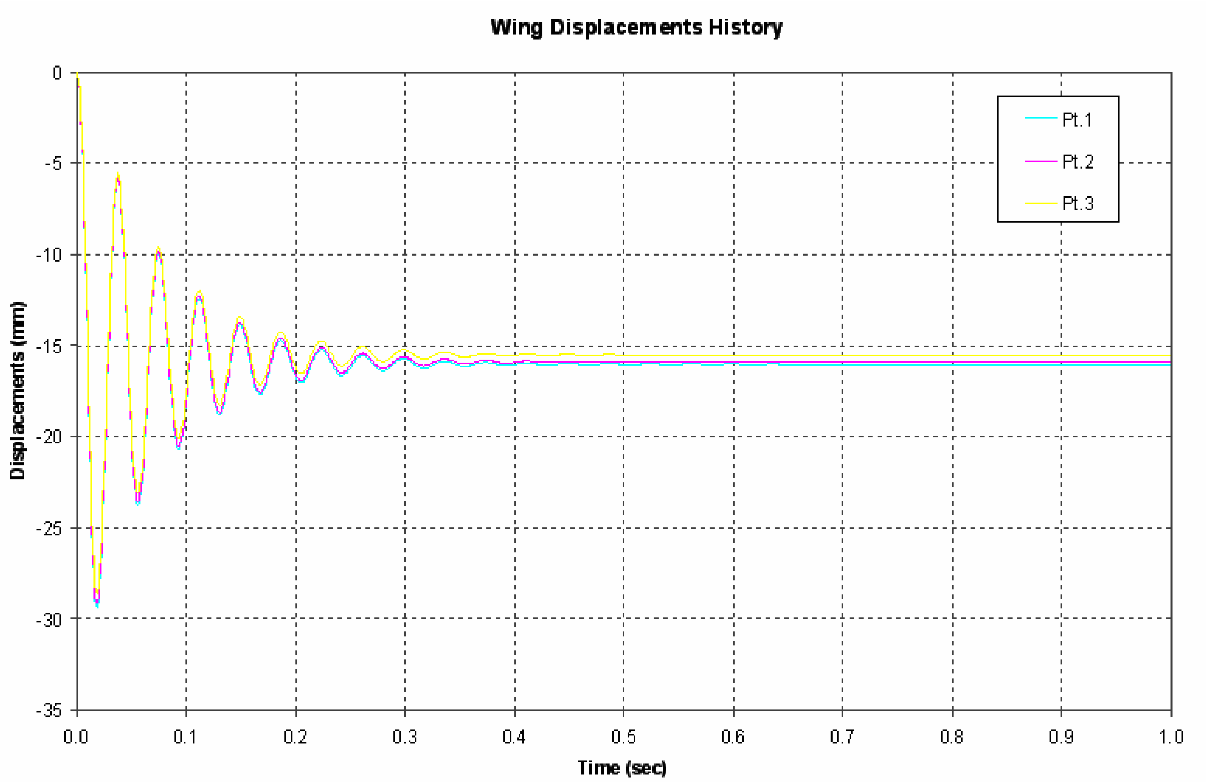

Figure 10: Wing displacement history obtained for Case 1.

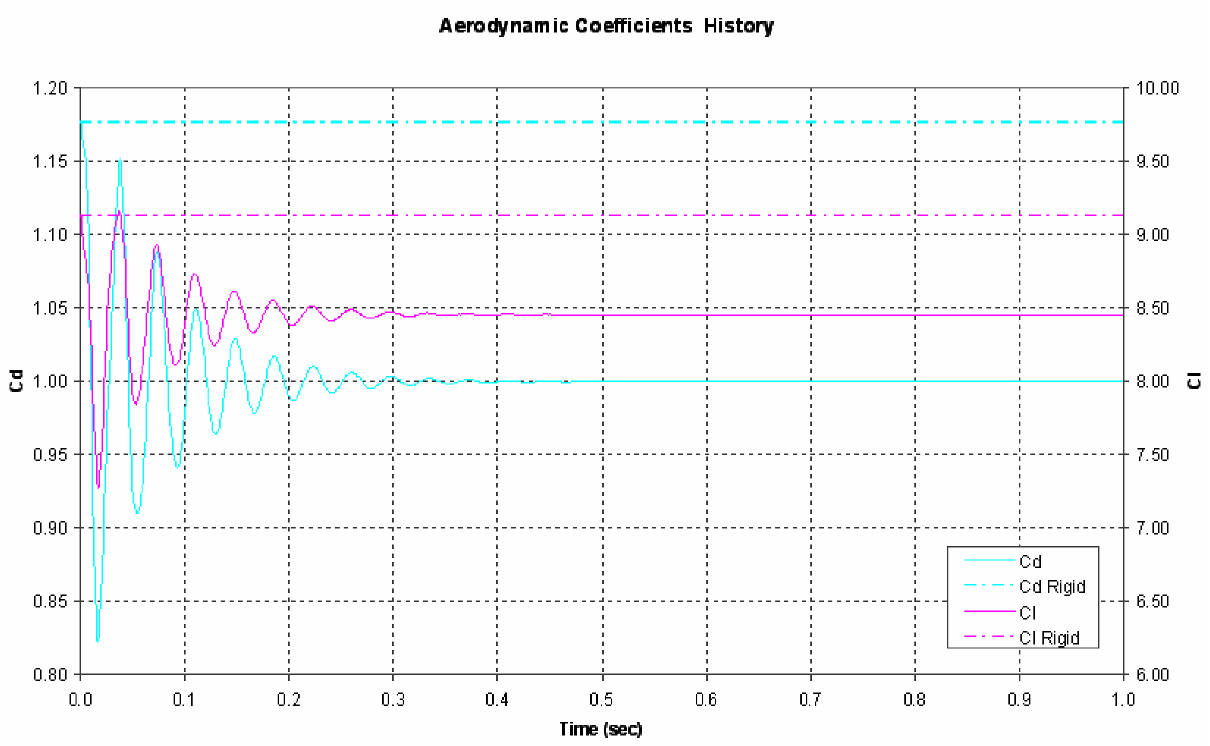

Figure 11: Aerodynamic coefficients history obtained for Case 1.

The evolution of wing deflection (Figure 12) and of aerodynamic coefficients (Figure 13) during an acceleration show that for investigated configuration the progressive deflection of the wing produces a reduction in drag an lift coefficients with the speed.

Figure 12: Wing displacement history obtained for Case 2.

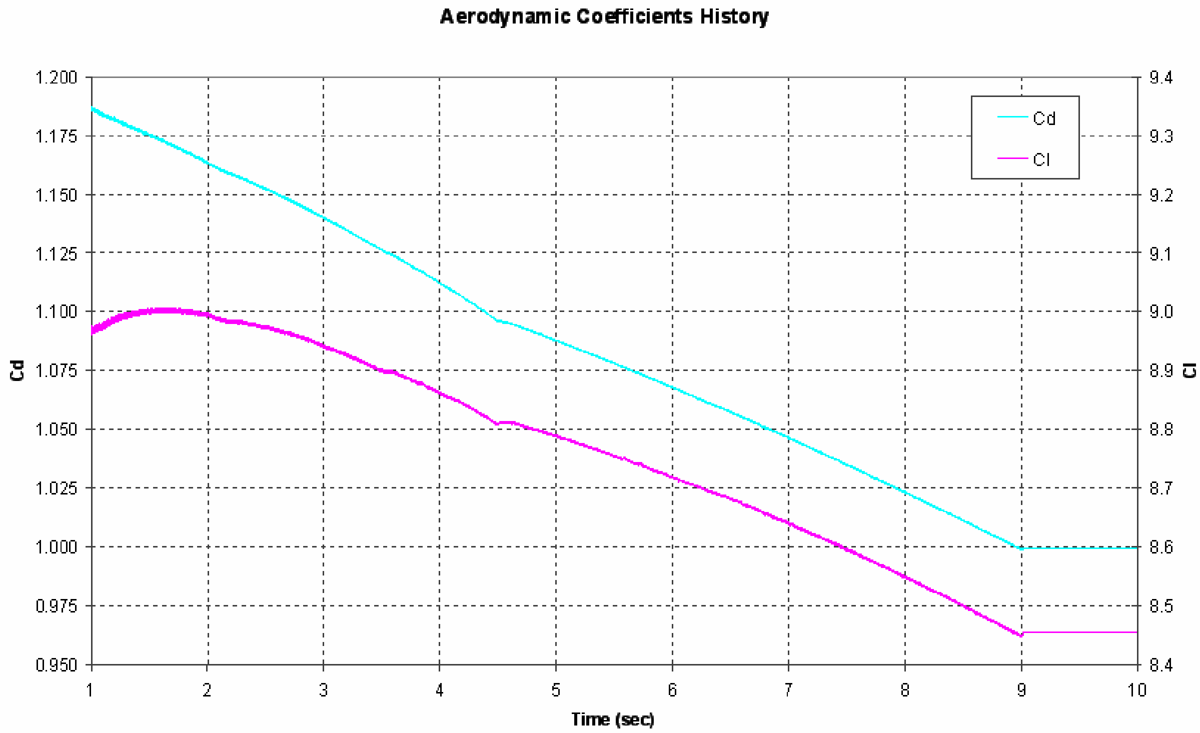

Figure 13: Aerodynamic coefficients history obtained for Case 2.

CONCLUSIONS

In this paper a dedicated approach for FSI modelling is presented. UDFs have been successfully used as a suitable programming tool for introducing FEM capability within FLUENT CFD solver. Three different applications are presented, in order to validate the code and illustrate some application examples.

In the case of vortex shadings and of the breaking dams a good correlation was found between calculated results and reference data.

The case of the F1 wing has sown the capability of proposed method to predict the actual aerodynamic coefficients during an acceleration manoeuvre.

These examples have shown that fast, accurate FSI simulations are feasible in FLUENT with some programming effort to properly represent the dynamics behaviour of coupled structures.

General thin walled structures are easily handled generating structural matrix by means of an external FEM solver. A dedicated solver for thin walled structures is now under development.

REFERENCES

(1) J.M. Genevaux, D. Bernardin, “The lattice gas method and interaction with an elastic solid”, Journal of Fluids and Structures 10, 873 – 892, 1996

(2) W. A. Wall, E. Ramm, “Fluid–structure interaction based upon a stabilized (ALE) finite element method in Computational Mechanics - New Trends and Applications”, Proceedings of the 4th World Congress on Computational Mechanics (WCCM IV), Idelsohn SR, Oñate E, Dvorkin EN (eds) CIMNE, Barcelona, 1998

(3) B. Hübner, E. Walhorn, D. Dinkler, “Space-Time Finite Elements for Fluid-Structure Interaction”, PAMM, Proc. Appl. Math. Mech. 1, 2002

(4) B. Hübner, E. Walhorn, D. Dinkler, “A monolithic approach to fluid–structure interaction using space–time finite elements”, Comput. Methods Appl. Mech. Engrg. 193, 2087–2104, 2004

(5) W. A. Wall, E. Ramm, “Shell structures - a sensitive interrelation between physics and numerics”, Int. J. Numer. Meth. Engng 60, 381–427 (DOI: 10.1002/nme.967), 2004

(6) Kölke, B. Hübner, E. Walhorn, D. Dinkler, “Strongly coupled analysis of fluid-structure interaction with two-fluid flows”, European Congress on Computational Methods in Applied Sciences and Engineering, ECCOMAS 2004, Jyväskylä, Finland, 2004

(7) M. Kämpchen, A. Dafnis, H.G. Reimerdes, G. Britten, J. Ballmann, “Dynamic aero-structural response of an elastic wing model”, Journal of Fluids and Structures 18, 63–77, 2004

(8) H.G. Matthies, J. Steindorf, “Partitioned Strong Coupling Algorithms for Fluid-Structure-Interaction”, Computers and Structures, 2002

(9) Comsol Inc., “Fluid-structure interaction example of multiphisics application”, Femlab internet site, 2004

(10) A.Z. Tarnopolsky, N.H. Fletcher, J.C.S. Lai, “Oscillating reed valves—An experimental study”, J. Acoust. Soc. Am. 108 (1), July 2000

(11) R. Baudille, M.E. Biancolini, “A general approach for the study of the motion of a cantilever beam interacting with a 2D fluid flow”, submitted for publication to the Journal of Fluids and Structures

(12) R. Baudille, M.E. Biancolini, “Modelling FSI problems in FLUENT: a dedicated approach by means of UDF programming”, Proceedings of the European Automotive CFD Conference, Frankfurt, Germany 2005.

(13) R. Fleck, G.P. Blair, R.A.R. Houston, “An improved model for predicting reed valve behaviour in two-stroke cycle engines”, SAE Paper no. 871654, 1987

(14) L. Meirovitch, “Elements of vibration analysis”, McGraw Hill, New York, 1986

(15) K. Blakely, “MSC/Nastran v68 Basic Dynamic Analysis guide”, MSC Corp, 1993

(16) R. Kelsey, W. Clinger, J. Rees, “Revised(5) Report on the Algorithmic Language Scheme”, Higher-Order and Symbolic Computation 11, 7-105, Kluwer Academic Publishers, August 1998

(17) R. Baudille, M.E. Biancolini, “Dynamic analysis of a two stroke engine reed valve”, XXXI AIAS Conference, Parma, Italy, 2002.

(18) R. Baudille, M. E. Biancolini, E. Mottola, “Nonlinear modelling of reed valve dynamics”, submitted for publication to the International Journal of Computer Applications in Technology

(19) R. Baudille, M.E. Biancolini, “Dynamic simulation of reed valves with fluid-structure interaction modelling”, submitted for publication to the Journal of Sound and Vibration