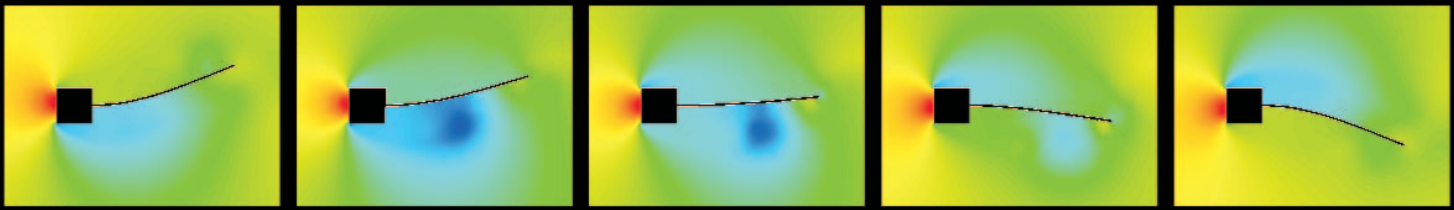
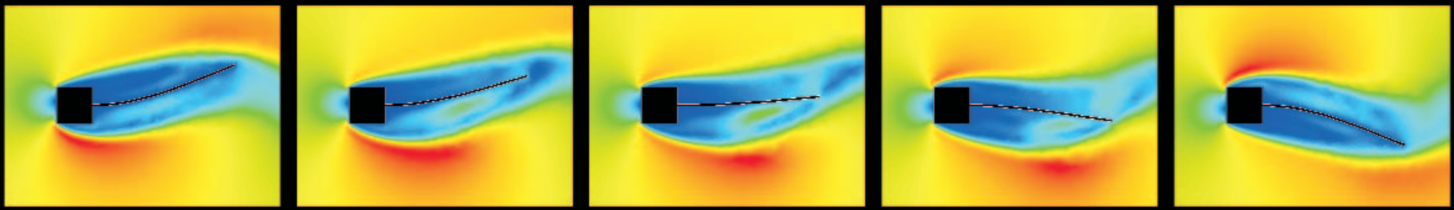


Static pressure

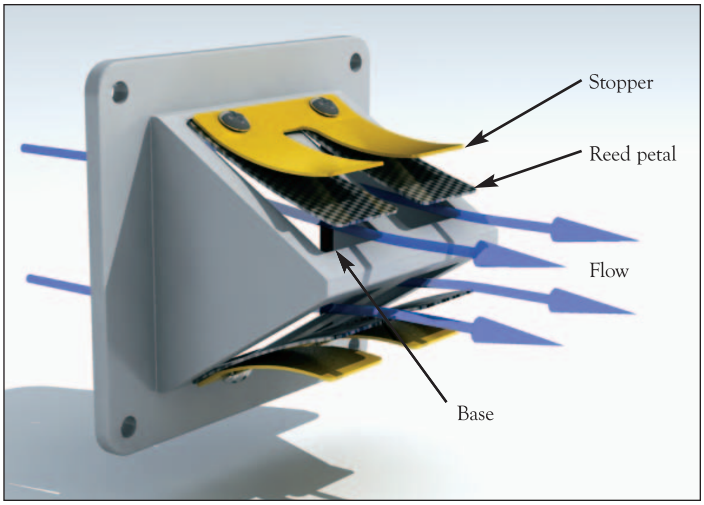


Velocity magnitude

The pressure and velocity fields around a square bluff body and slender flexible structure are shown for different times during an FSI simulation

FSI Makes FLUENT More Flexible

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



A diagram of a reed valve, showing the flexible reed petal, base, and stopper

FLUID-STRUCTURE INTERACTION (FSI) is an important and interesting phenomenon, but it is a difficult challenge for numerical modeling. It even poses difficulties for numerical modelers. Structural behavior 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 behavior of the system. Fluid-structure interaction plays a role in flow-induced vibration, blood flow in vessels and heart valves, tire hydroplaning, wind instruments, sailing, and the deployment of air bags, for example.

At the University of Rome "Tor Vergata", a 2D FSI model has been developed for FLUENT using a dedicated FEM solver, coded as a user-defined function (UDF). Arbitrary Euler beams are handled by the model, and include non-linear effects, such as the contact between rigid and flexible walls. Special care is taken to manage boundary motion in FLUENT since contact between moving surfaces is denied by FLUENT's remeshing algorithm. Given that some applications require taking surface contact into account, a dedicated algorithm was specifically developed to manage the contact problem, not only from a structural point of view, but from a fluids perspective as well. Moreover, a special solution strategy is applied that leaves the CFD simulation time virtually 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.

A well-known benchmark case was used as an initial test of the model. This case describes the vortex shedding excitation of a cantilevered beam positioned downstream of a square obstacle in a laminar flow. From a structural point of view, the deformed shape of the cantilevered 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 results obtained were compared with the reference values given by Ramm and Wall (1) and Walhorn, Hubner, and Dinkler (2). Animations of the CFD predictions for the pressure and velocity fields around the bluff body and the slender flexible structure are consistent with expectations, and a plot of the vertical displacement of the beam free end with respect to time is in excellent agreement with the literature reference data cited above.

One practical application that needs an FSI approach is the working cycle of a reed valve (3). Reed valves are pressure-driven flow stoppers, used in systems such as two-stroke engines, compressors, and shock absorbers. The reed is comprised of a petal made of flexible material and mounted on a base. The base serves as a barrier for valve closure, while an upper stop, which can be curved, has a double function: it serves as a guide to prevent excessive opening of the valve, and it optimizes the shape of the open channel from a fluid dynamics point of view. The overall gas dynamics and local fluid motion depend on the open area of the valve. The open area depends on the reed motion, which is driven by pressure loads produced by the fluid.

For the case of a reed valve used in an engine, an FSI simulation was performed that made use of actual pressure histories at the inlet and outlet boundaries (4). Air suction in the crankcase is produced when the piston moves toward top dead center (TDC). This gives rise to a pressure difference relative to the inlet manifold, which causes the reed valve to open. The actual behavior is complicated by the valve dynamics, the inertial effects of the entering air, and a local vortex that forms. Even so, the contact algorithm predicts the anticipated bouncing of the petal at closure and at full opening. The reed tip vertical displacement agrees very well with experimental data given by Fleck *et al.* (4). The strong non-linear behavior of the reed petal suggests that it is not useful to study its free-vibration characteristics. Because the reed interacts with constraints that change its free length, its unconstrained behavior is totally different.

These and other examples have shown that fast, accurate FSI simulations are feasible in FLUENT with some programming effort to properly represent the dynamic behavior of coupled structures. In the next phase of the work, a UDF will be developed for the interaction of thin, three dimensional, non-isotropic structures based on the shell finite element method. ■

References

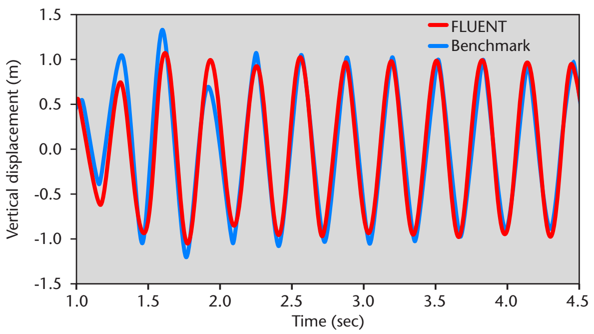
1 W.A. Wall, E. Ramm, Fluid-structure interaction based upon a stabilized (ALE) finite element method, 4th World Congress on Computational Mechanics – New Trends and Applications. S. Idelsohn, E. Onate and E. Dvorkin (eds.), CIMNE, Barcelona, 1998.

2 E. Walhorn, B. Hübner, D. Dinkler, PAMM, 2002, 1, 81.

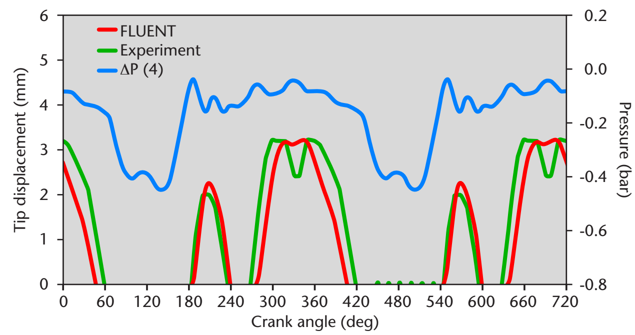
3 R. Baudille, M.E. Biancolini, Dynamic analysis of two stroke engine reed valve. XXXI Convegno Nazionale dell’Associazione Italiana per l’Analisi delle Sollecitazioni (AIAS), Parma, Italy, 2002.

4 R. Fleck, G.P. Blair, Houston R.A.R. SAE Paper 871654, 1987.

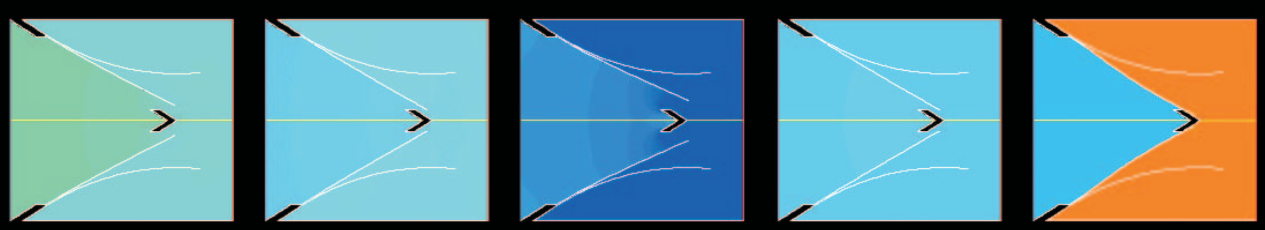
More.info@
rbaudille@torvergata-karting.it



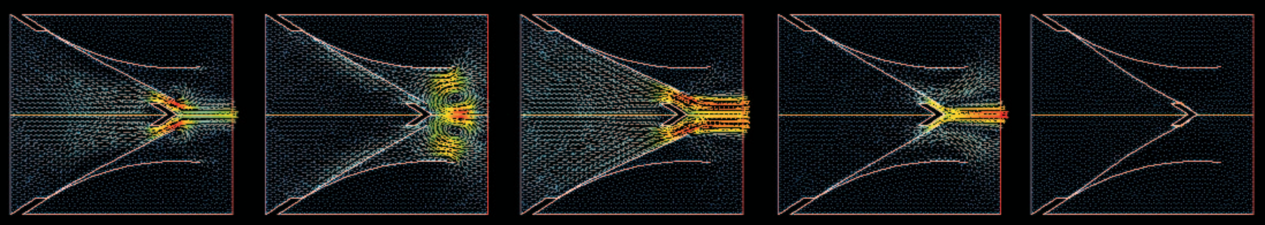
The vertical displacement of the beam free end predicted by the FSI model is compared to reference data (1, 2)



Tip displacement of an engine reed valve predicted by the FSI model and measured experimentally (4); the measured pressure differential is also shown



Static pressure



Velocity vectors

The flow field in a reed valve at various times for an engine running at 4770 RPM