NEW METHOD FOR COUPLED FLUID-STRUCTURE INTERACTION PROBLEMS IN BIOMECHANICS

[1]Damian J. J. Farnell, [2]Tim David and [3]David C. Barton
[1]Department of Medicine, University of Liverpool, Liverpool, UK [2]University of Canterbury, Christchurch, New Zealand
[3] School of Mechanical Engineering, University of Leeds, Leeds LS2 9JT; email: d.c.barton@leeds.ac.uk

INTRODUCTION

There are many examples in nature of situations in which a fluid interacts strongly with a flexible structure. Examples range from flapping flags to biological "systems" that are often critical to our survival, e.g. blood flow in the cardiovascular system. The role of numerical simulation of such complex systems is becoming increasingly important as they often yield insight into the fundamental behavior involved. Historically, numerical codes to analyse the dynamic behavior of structures (based on explicit finite element methods e.g. [1]) have evolved separately from those for fluids that are based on computational fluid dynamics (CFD) theory, with the resulting computational complexity and cost reducing the range of applications to which the coupled methodology has been applied. An alternative approach is described here, based on a Lagrangian dynamics approach, embedded in a commercial CFD package (FIDAP). This has the advantage of (relative) computational simplicity and flexibility. The basis of the approach is outlined below followed by a description of its application to two particular problems in biomechanics: blood flow through a heart valve and the swimming action of certain eel-like fish.

METHODS

The flexible structure is treated as an *N*-tuple pendulum with stiffness and damping terms at each "hinge." Each element of the pendulum is assumed to be rigid with uniformly distributed mass. There is only one degree of freedom associated with each element (its rotation), plus an additional freedom for the leading edge of the piecewise-linear structure in the case of the "swimming fish". A Lagrangian is formed from the potential, kinetic and dissipative energy terms of the structure and this is manipulated to form the Euler-Lagrange equations, solution of which yields the motion of the structure. The main external forcing terms in the equations are the pressure differential acting on each element but may also include other terms e.g. due to muscle forces in swimming fish. The structural dynamics solver is written as a subroutine to the commercial CFD package FIDAP that solves the Navier-Stokes equations for the fluid domain in question. The motion of the structure is imposed as a moving boundary condition on the fluid and, in a circular fashion, the pressures generated by the fluid are applied in the Euler-Lagrange equations for the structure. Hence, we solve the two physical problems using a weakly coupled solver, and this approach has been shown to be both highly efficient and accurate [2].

RESULTS AND DISCUSSION

Figure 1 shows a typical result from a 2D coupled analysis of an aortic heart valve. In this case, the two leaflets have been modeled independently, allowing flow asymmetries to develop. Animations of the complete cardiac cycle have been generated which show clearly the full opening and closing of the leaflets and the associated flow vortices.

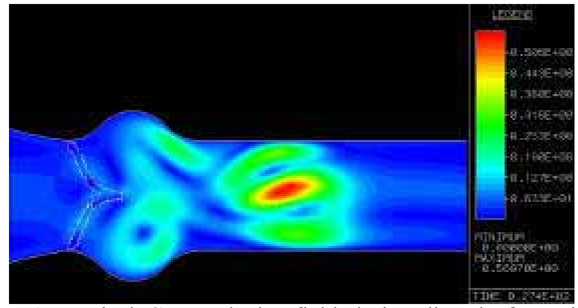

Figure 1: Typical CFD velocity field during diastole for artificial heart valve simulation. The "inlet" from the left ventricle is shown on the left and the two aortic leaflets are shown in outline in white.

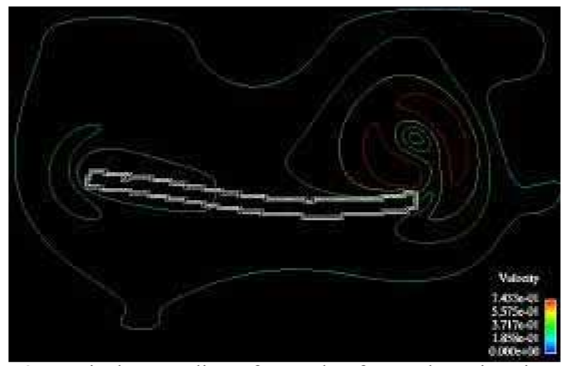

Figure 2: Typical streamlines from the forwards swimming "fish" simulation. The left hand end of the white outline represents the leading end ("head") of the fish.

Figure 2 shows a typical result from the swimming fish simulation. Unlike the heart valve case, the leading edge of the structure is not constrained in space and terms representing the fish's muscle forces are added to the Euler-Lagrange equations. By changing the parameters of the system, it was possible to reproduce both forwards and backwards swimming motions, with quite different vortex structures formed in the two cases. These results are in good agreement with experimental observations [3].

CONCLUSIONS

A novel method has been developed for numerically simulating coupled fluid-structure interaction problems in nature. The method has been applied to several problems, including heart valve dynamics and the swimming action of certain fish, and shown to be robust, efficient and reliable. In principle, it will be possible to extend the method to other problems (e.g. retinal tears in the posterior chamber of the eye) and to fully three-dimensional analysis.

REFERENCES

1. Thornton MA, et al. *Med Eng Phys,* **19**, 588-597, 1997.
2. Farnell DJJ, et al. *J Fluids & Struct,* (in press), 2005.
3. D'Aout K & Aerts P. *J Exp Bio*, **202,** 1511,1999.