8th. World Congress on Computational Mechanics (WCCM8)
5th. European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 – July 5, 2008
Venice, Italy

A 3D Unsteady Incompressible Flow Solver and Its Application in Fluid Structure Interaction Problems

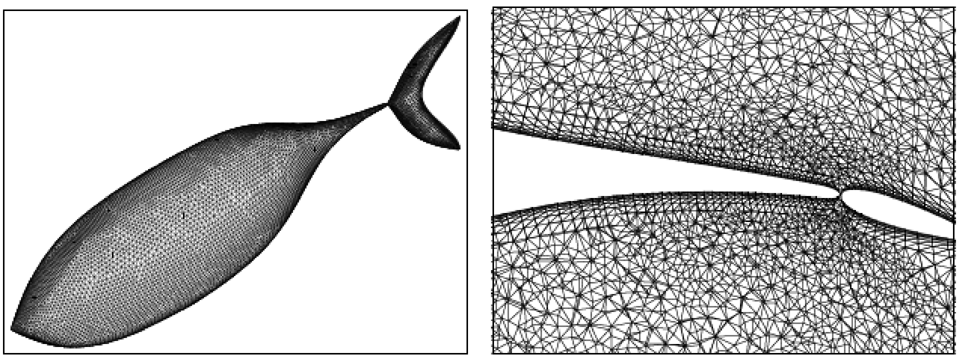
Oubay Hassan, C. A. C.Wood, Z. Zhang, A. J. Gil and K. Morgan

Civil and Computational Engineering Centre
School of Engineering
Swansea University
Swansea SA2 8PP
Wales, UK
o.Hassan@swansea.ac.uk

KEY WORDS: *incompressible flow, artificial compressibility, moving boundary, unstructured mesh, implicit time stepping, multidisciplinary, fluid/structure interaction.*

ABSTRACT

The incompressible three dimensional Navier–Stokes equations are solved using an implicit artificial compressibility approach. To enable simulations of practical interest, which often involve moving boundary components, an unstructured mesh ALE formulation is adopted. With this approach, the size of the allowable physical time step is independent of the spatial discretisation, so that the time step adopted can be selected by considering accuracy criteria only. To avoid working with large matrices, equation solution is achieved by explicit time stepping and the convergence is accelerated by employing an agglomerated multigrid procedure. A single consistent mesh is employed and any geometry changes are accommodated by mesh adaptation. For many problems, the geometry changes are such that it is sufficient to employ mesh deformation approaches, in which the connectivity remains unchanged as the mesh cells are deformed to conform to the new geometrical shape. This is accomplished by moving the mesh using a Delaunay graph mapping or a wall–distance based algorithm. The numerical performance of the procedure is demonstrated by considering the simulation of flow over a flexible fish–like body. Typical results are shown in Figures 1 and 2.



(a) (b)

Figure 1: Simulation of flow over a swimming fish: (a) view of the surface mesh; (b) detail of a cut through the mesh.

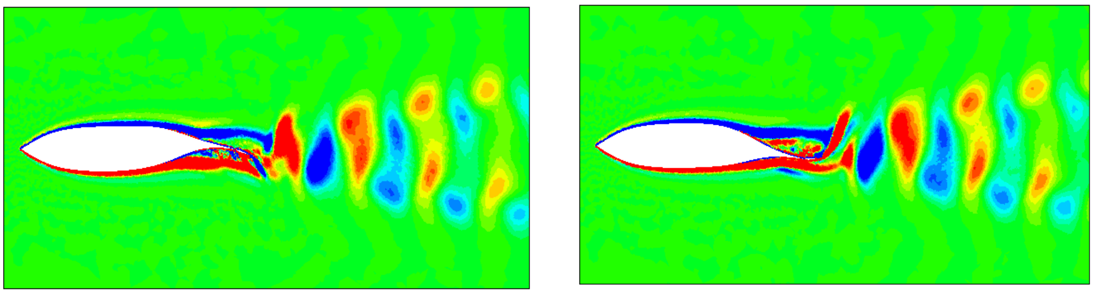


Figure 2: Simulation of flow over a swimming fish, showing a view of a vorticity component at two different times.

In the multi–disciplinary spirit of the Symposium, it will be demonstrated how this approach can be employed as part of a fully coupled, partitioned, time marching solver for the analysis of transient fluid/struture interaction problems. With this method, although the interacting fields are not solved simultaneously, very few resolutions of the coupled problem are required at each time step to achieve sufficient coupling. The approach is attractive, as it allows the modularity of the fluid and structure solvers to be retained. Coupling of the fields is achieved through a conservative transfer of information at the fluid/structure interface. An implicit coupling is accomplished when the solutions of the fluid and structure sub–problems are cycled at each time step until convergence is reached. The finite element analysis of highly deformable structures is performed by combining an implicit Newmark time integration algorithm with a Newton–Raphson second order method. The numerical performance of the coupled procedure will be demonstrated by simulating the flow induced oscillations of a flexible cantilever. Typical results are shown in Figure 3.

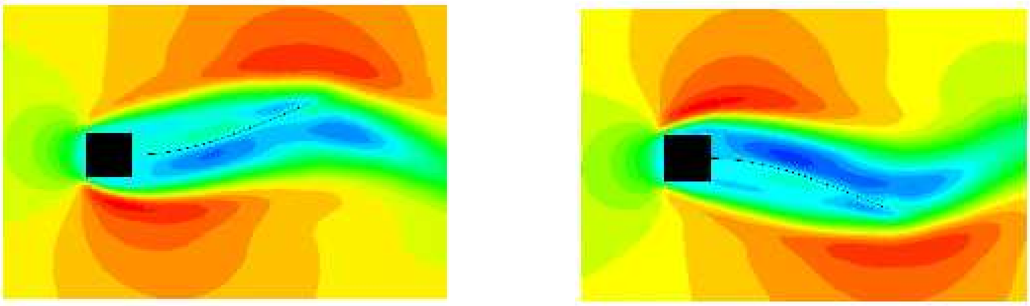


Figure 3: Flow induced oscillations of a flexible cantilever showing the x–direction velocity

REFERENCES

[1] R. Lohner, K. Morgan and O. C. Zienkiewicz, Adaptive grid refinements for the compressible Euler equations, *Accuracy Estimates and Adaptive Refinements in Finite Element Computations*, John Wiley & Sons, 1986, 281–297.

[2] X. Liu, N. Qin and H. Xia, Fast dynamic grid deformation based upon Delaunay graph mapping, *Journal of Computational Physics*, 211 (2006), 405–423.

[3] A. J. Chorin, A numerical method for solving incompressible viscous flow problems, *Journal of Computational Physics*, 2 (1967), 12–26.

[4] H. Steve, M. H. Lallemand and A. Dervieux, Unstructured multigridding by volume agglomeration: current status, *Computers and Fluids*, 21 (1992), 397–433.