A FLOW-SOLVER WITH FLEXIBLE BOUNDARIES

A.V. Smirnov, W. Huebsch, C. Menchini West Virginia University Mechanical & Aerospace Engineering Department Morgantown, USA email: asmirnov@wvu.edu

ABSTRACT

A method suitable for computing fluid flow in the presence of moving boundaries is presented. In particular, the interaction of fluid with elastic walls is considered. The flow solver was developed based on artificial compressibility concept. The discrretization is done on unstructured tetrahedral meshes. A model of fluid interaction with elastic boundaries including boundary and mesh deformations was incorporated. Several applications of fluid interaction with elastic membranes are presented.

KEY WORDS

Numerical Methods; 3-Dimensional Modelling; Dynamic Modelling; Physically-based Modelling; Biomedical Modelling; Air Modelling and Simulation

1 BACKGROUND

In recent years, the area of computational fluid dynamics (CFD) has seen many advances in flow solver and grid generation techniques, which include unstructured grids (meshes) and moving boundaries, among others. Simulation of complex flows involving interaction with the elastic structures and moving boundaries poses challenges in the development of flow solvers and mesh readjustment techniques.

The conventional flow solvers based on the Poisson equation for pressure [1] are not well designed to handle the problem of fluid interaction with elastic objects and boundary deformations. This is because the Poisson equation is a rough approximation, and masks many physical effects related to the thermodynamic relation between pressure and density, ranging from static pressure distributions to acoustic modes. In this study we present a direct approach to solve the flow problem based on conservation laws and thermodynamic relations. The simulation makes use of a finite volume flow solver with a three-dimensional unstructured tetrahedral grid and vertex-centered discretization.

The primaray advantage of the unstructured mesh approach over a structured grid generation is its applicability to a wider range of geometrical shapes and domain topologies, as well as the reduced effort in grid generation for complex geometries. Another advantage of unstructured meshes is the ability to incorporate adaptive grid methodologies based on local flow features [2]. For three-dimensional flows, the tetrahedral elements have been a popular choice for the mesh generation [3]. However, unstructured meshes involve considerably higher algorithmic complexity and additional computational overheads: increase in memory requirements and computer run time on a per grid point basis [4]. Another problem arises when trying to use tetrahedral elements in a highly stretrched grid area to resolve boundary layer flow. This technique is typically inefficient and difficult to implement. Work by Lai [5] has started to address this deficiency in unstructured grids.

There are two basic finite-volume techniques for the discretization on unstructured meshes: cell centered scheme and a cell vertex scheme [1]. The discretization scheme used in this study is based on a vertex-centered finite-volume approximation. This has advantages over the cell-centered scheme in handling moving domain boundaries. Vertexcentered finite volume schemes have been used on various types of CFD problems, which as the name implies, retains the flow variables at the cell verticies. Foy and Dawes [3] point out that the cell-centered version is approximately five times as expensive in memory requirements as compared to the vertex storage for an unstructured mesh. There is actually a limited amount of available literature for CFD work that incorporate the models used in the present study: finite volume flow solver for 3-D unstructured tetrahedral meshes with a vertex-cented discretization scheme. Some of the relevant literature include the work by Anderson, et al. [4], Foy and Dawes [3], and Watterson [2].

The idea of incorporating a moving boundary in the flow simulation has also started to gain increased interest in the CFD world as researchers look to investigate problems such as deformation of droplets, liquid free surfaces, aeroelasticity, and lung compliance, to name a few. Enhancing the capabilities of the flow solver with unstructured grid generation and moving boundaries allows CFD research to continually investigate more complex physical problems [6].

2 METHOD

Flow Solver

The method of this study is based on the coupled solution of equations of mass and momentum conservation, augmented with the thermodynamic relation of state¹

$$\dot{\rho} + (\rho u_i)_{,i} = 0 \tag{1}$$

$$\dot{u}_i + u_j u_{i,j} = v u_{i,jj} - \rho^{-1} p_{,i}$$
⁽²⁾

$$p = \rho RT \tag{3}$$

where ρ , u_i , p, v are density, velocity, pressure and kinematic viscosity. For simplicity we consider the ideal-gas law as an equation of state with R as the gas constant, and T as the absolute temperature.

The inclusion of the thermodynamic equation of state (3) instead of the more commonly used Poisson equation for pressure presents challenges for achieving a stable solution procedure. However, the incompressible formulation of the momentum equation (2), a special form of a pressure source term, and quasisteady approximation can provide a steady solution in a certain range of Reynolds numbers. Furthermore, the usage of high-precision arithmetics can open this approach to predicting a wider range of unsteady phenomena than is possible with the conventional incompressible formulations.

A stable discretization of the equation system (1)-(3) in a control-volume formulation can be obtained by using the mass of the control volume, *m* as an independent variable

$$m = \rho c v \tag{4}$$

where cv is the control volume. The equations are discretized on an unstructured tetrahedral mesh, with vertex-centered locations of the variables (Sec.2). The volume of the polyhedron representing the control volume is computed as

$$cv = \sum_{k=1}^{N_f} x_i^k a_i^k \tag{5}$$

Here the summation is done over all the N_f faces of the polyhedron. Vector x_i holds the coordinates of the face center, and a_i is the face area vector.

With m, u_i, p as independent variables, the solution is accomplished through the following scheme.

$$m(t+dt) = m(t) + \dot{m}dt \tag{6}$$

$$u_i(t+dt) = u_i(t) + \dot{u}_i dt \tag{7}$$

$$\dot{m} = \frac{1}{cv} \sum_{k=1}^{N_f} m^k u_i^k a_i^k$$
(8)

$$\dot{u}_i = \frac{c_i + d_i}{cv} + \frac{f_i}{m} \tag{9}$$

These relations are given for a single control volume. Terms c_i, d_i, f_i represent convection, diffusion and pressure-force, each computed through by summation over the faces of a control volume.

$$c_{i} = -\frac{1}{cv} \sum_{k=1}^{N_{f}} u_{i}^{k} u_{j}^{k} a_{j}^{k}$$
(10)

$$d_i = \frac{\mathbf{v}}{cv} \sum_{k=1}^{N_f} u_{i,j}^k a_j^k \tag{11}$$

$$f_i = \sum_{k=1}^{N_f} \Delta p^k a_i^k \tag{12}$$

where Δp^k is the pressure drop across the face *k*. Equations (6)-(12) are closed by the equation of state, obtained from (3) and (4):

$$p = \frac{mRT}{cv} \tag{13}$$

The discretization scheme above is based on combined equations of fluid and thermo dynamics (1), (2), (3), and as such it is capable in principle of reproducing a wide variety of physical effects, including acoustical modes. At the same time the range of scales associated with the fluid flow and acoustical modes can be very different, and may cause convergence problems. For this reason the momentum equation (2) is formulated and discretised in an incompressible manner. It should be noted that the special form of the pressure source term (12) appeared to have better stabilizing properties compared to other commonly used control-volume or finite difference approximations [1] that were tried in the course of this study.

This approach provides for a greater simplicity of implementation and higher stability by avoiding the solution of the Poisson equation for pressure commonly used in Navier-Stokes solvers. The method is still accurate in conserving mass and momentum for steady-state and quasi-steady-state solutions, i.e. on the time-scales much longer than the acoustical time scale. In these situations the exact values for thermodynamic parameters, such as R and T in (3) can be replaced by pseudo-values to speed up the convergence. As such the approach is similar to the artificial compressibility method [7]. At the same time the method has a potential of producing fast non-steadystate solutions and acoustical modes, at the expense of smaller time steps and possibly higher precision arithmetics.

¹We use notation: $a \equiv da/dt, a_i \equiv \partial a/\partial x_i$

Another advantage of this scheme that was exploited in this study is its straightforward implementation in the problems with moving boundaries, such as biomedical flows. Since the pressure field is related locally to the density it is easy to account for the effects of changing domain geometry on the flow-field through the pressure-density relation. Several applications of the technique are considered in Sec.3.

Moving Boundaries

To enable a simple and stable implementation of moving boundaries a tetrahedral mesh with vertex-based location of all variables is used. Figure 1 shows a twodimensional version of the mesh. As it can be seen on the figure, the control volumes represent an overlapping set of polyhedrons, each consisting of several tetrahedral cells. This arrangement is commonly used in finite element method, and can also be found in finite-volume methods [1]. The advantage of this scheme as compared to the cell-centered scheme, commonly used in a control-volume method, is in simplification of mesh deformation routines, and account of boundary conditions. Since the deformations of the mesh are accomplished by moving the cellvertexes, the vertex-location of the variables would be preferred. The control-volumes are represented by polyhedrons as opposed to tetrahedrons in a cellcentered scheme, and they are generally 3-4 times fewer in number and have better convex properties. Still another advantage of vertex-centered discretization is a more accurate interpolation in the interior of the cell, which makes this scheme better in the problems of particle dynamics and the handling of overlapping domains. The disadvantage is in the necessity to perform a separate assembly step, similar to the one done in a finite-element scheme, which consists of a loop over all tetrahedral cells to assemble the source-terms and the contributions of convective and diffusive fluxes.

Boundary conditions. In the vertex-centered scheme the boundary control volumes represent a special case, since their centers are located exactly at the boundary, and not in the center of the control volume (Nodes C,D,E,F in Fig.1). Realization of wall boundary conditions can not be done by simply setting the velocities at the boundary nodes to zero since it will lead to the loss of conservation of mass and momentum which will still be transported from the neighboring cells to the boundary cells by means of convection and diffusion. In this case the boundary conditions are realized by introducing the appropriate forcing functions, i.e. pressure forces, at these nodes. This creates an artificial drag at the boundary, which represents the effect of the wall friction. The wall-drag



Figure 1. Overlapping control volumes in a vertex-centered mesh. Shaded areas represent overlapping control volumes around nodes A, and B, with dark shaded area being the region of overlap.

coefficient can be selected arbitrarily large, effectively setting the velocity at the boundary to zero, i.e. nonslip condition, which will still comply with the mass and momentum conservation. In a turbulent flow situation the drag coefficient can be selected according to the turbulent wall shear stresses (wall-functions).

Mesh adjustment. Each deformation of the boundary necessitates readjustment of the internal nodes of the mesh, so as to preserve the convexity of the control volumes. This is done by placing each node of the mesh in the center of mass of its respective control volume. This displacement of the nodes may cause an unphysical transport of variables. In order to avoid this error the motion of each node of the mesh is considered in a non-inertial frame of reference when the velocity and acceleration of the node are appropriately added to the velocities and source-terms in the transport equations for the variables in a laboratory frame of reference.

Elastic walls. When the boundary represents an elastic membrane, the elasticity model is used where each triangular face of the boundary can experience normal and shear stresses in response to deformations. These stresses are combined with the pressure forces acting from the fluid normal to the face, and applied to the boundary nodes. Each boundary node is considered to be a material points with the mass equal to the mass of the surrounding control volume. If the boundary shell itself has a non-negligible surface density, the combined weight of the surrounding boundary elements should be added to the mass of the boundary node.

Let's consider the computation of the elastic forces on a boundary element. Figure 2 shows a triangular boundary element ABC, which after the deformation changes shape to AB'C. The total deformation at node B' can be decomposed into the normal



Figure 2. Deformations of a triangular face.

and tangential components with respect to the segment AC as shown in the figure as dh and db respectively. Then the normal (\mathbf{F}_{\perp}) and tangential (\mathbf{F}_{\parallel}) stress forces acting at node B' will be proportional to the corresponding deformations: dh/h and db/h. Since in a general case all the edges can change their lengths and orientations, we have to store the original shape of each triangular face in separate arrays. Then the forces representing elasticity stresses are computed as

$$\mathbf{F}_{\perp} = \mathbf{n}_{\perp} C_{\perp} \frac{dh}{h_0} S_0$$
$$\mathbf{F}_{\parallel} = \mathbf{n}_{\parallel} C_{\parallel} \frac{db}{h_0} S_0$$
$$\mathbf{n}_{\parallel} = \frac{\vec{AC}}{|AC|}$$
(14)

where C_{\perp} , C_{\parallel} are the normal and tangential elasticity coefficients, h_0 , S_0 are the original height and area of the triangle, and vectors \mathbf{n}_{\parallel} and \mathbf{n}_{\perp} are the normal unit vectors in the directions tangential and normal to the current orientation of segment AC. These forces will act in the plane of the triangle. The vector sum of these forces is computed for each node of the triangle and accumulated at the nodes during the separate loop over all the boundary triangles. In addition to this the pressure force of the fluid will act normally to the plane of the triangle. The net fluid pressure force is computed in the main assembly loop over all the tetrahedrons of the mesh. After that another loop over the boundary nodes performs the adjustment of boundary node positions according to the net forces acting on them. Then the relaxation of internal mesh nodes follows. This coupling of elasticity and fluid pressure forces realizes the interaction mechanism between the fluid and elastic boundary.

The numerical implementation of this elasticity model involves allocation of extra variables that are placed at the boundary nodes, and represent the deformations and elastic forces, and introducing an additional assembly loop over the boundary face ele-



Figure 3. Matching the parabolic profile in a laminar pipeflow case.

ments as described above. The overall computational scheme of the solver consists of repeated execution of the following four steps:

- 1. Iterate the flow solver.
- Assemble body forces in a loop over all mesh cells (tetrahedrons).
- 3. Assemble surface forces in a loop over all the boundary elements (triangles).
- Move the boundary nodes under the action of body and surface forces.
- 5. Readjust the internal mesh nodes.

The iteration time steps of the flow and elasticity solvers can differ depending on the physical response times of the respective forces. In most cases the flow solver is sub-cycled with respect to the elasticity solver.

3 APPLICATIONS

We considered several test cases representative of static, steady and unsteady flows, and flow interactions with elastic walls. The computational mesh for all the test cases was constructed using TAM mesh generation method [8]. Pre-post processing and the scenarios of hard body motions and body-shell-fluid interactions used in some cases were realized using the MulPhys simulation environment [9] (see also www.mulphys.org).

Figure 3 shows the predictions for a laminar flow in a cylindrical pipe compared to the analycal solution. The solution reproduces the parabolic shape as well as satisfies the mass and momentum conservation along the pipe.



Figure 4. Surface deformation of an elastic pipe with a passage of a compressed gas.

A more complex example includes the passage of pulse of a compressed gas through an elastic pipe. Figure 4 shows deformations of the surface of the pipe². The pipe wall was considered as an elastic weightless membrane. The interaction between the membrane and the fluid was modeled through pressure-induced normal surface forces.

Figure 5 shows a quasi-steady-state simulation of the pulsating flow passing through a bifurcating duct with elastic walls. With a passage of a pulsating flow the walls of the duct bulge and then return to the original shape.

An example in Fig.6 involves a quasi-static solution of a shape of an elastic membrane with a compressed fluid inside. Fig.7 shows a complex interaction of a hard ball penetrating into the fluid-filled elastic spherical shell.

4 CONCLUSIONS

The presented approach provides the solution of fluidelastic interactions, and can be used in problems involving static interactions, steady-state and quasisteady state fluid motion in the presence of elastic walls and moving boundaries.

The artificial compressibility method used in this study provides a simple implementation of fluidstructure coupling and opens perspectives of predicting vibrations, acoustical modes and other phenomena associated with thermodynamic equations of state. On the other hand, in a quasi steady state limit and incompressible solution is obtained.

The elasticity model of the wall is based on direct calculations of shear and normal deformations of boundary elements. The combination of these forces with fluid pressure forces determine the dynamics of the boundary. Extension of this approach to the 3D elasticity model is straightforward.

In addition to the flow-solver the effectiveness of



(a) Flow field



(b) Surface deformations

Figure 5. Unsteady flow in a bifurcating elastic duct.

²Animations are also available at www.mulphys.com/elastic



Figure 6. Bulging box with a compressed gas inside.



Figure 7. Elastic bag with a fluid inside hit by a hard ball.

The animation of this event can be found at mulphys.com/elastic.

the approach depends on the elasticity solver and the coupling between the two. Another important part of a moving boundary problem is the technique used for the motion and re-adjustment of the computational mesh, the detailed description of which is beyond the scope of this paper.

References

- [1] J.H. Ferziger and M. Peric. Computational Methods for Fluid Dynamics. Springer Verlag, 1997.
- [2] J.K. Watterson. A pressure-based flow solver for the three-dimensional navier-stokes equations on unstructured and adaptive meshes. In 25th AIAA Fluid Dynamics Conference, 1994.
- [3] B. de Foy and W. Dawes. Unstructured pressurecorrection solver based on a consistent discretization of the poisson equation. *International Journal for Numerical Methods in Fluids*, 34(6):463–478, 2000.
- [4] W.K. Anderson, R.D. Rausch, and D.L. Bonhaus. Implicit/multigrid algorithms for incompressible turbulent flows on unstructured grids. *Journal* of Computational Physics, 128:391–408, 1996.
- [5] Y.G. Lai. Unstructured grid arbitrarily shaped element method for fluid flow simulation. AIAA Journal, 38(12):2246–2252, 2000.
- [6] W. Shyy, M. Francois, and H.S. Udaykumar. Cartesian and curvilinear grid methods for multidomain moving boundary problems. In *Thirteenth International Conference on Domain Decomposition Methods*, 2001.
- [7] K.A. Hoffman and S.T. Chiang. Computational Fluid Dynamics for Engineers, volume 1. Engineering Education System, Wichita, Kansas, 1993.
- [8] A.V. Smirnov. Tool assisted mesh generation based on a tissue-growth model. *Medical and Biological Engineering and Computing*, (MS no 02/198), 2003. Accepted for publication.
- [9] A.V. Smirnov. Multi-physics modeling environment for continuum and discrete dynamics. In *IASTED International Conference*, number 380-174 in Modelling and Simulation, Palm Springs, CA, 2003.