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

Large deformations Fluid-Structure interactions by means of Lagrangian finite element method

*** M. Cremonesi[1], A. Frangi[1] and U. Perego[1]**

[1] Dipartimento di Ingegneria Strutturale
Politecnico Milano
piazza Leonardo da Vinci 32 Milano, Italy
cremonesi@stru.polimi.it

Key Words: *fluid-structure interaction, Lagrangian approach, large deformation*

ABSTRACT

In problems with free-surface flows or flows with breaking waves, or when the fluid flow is coupled with a solid which undergoes large deformations, the treatment of free surfaces and solid-fluid interfaces is better dealt with in a Lagrangian framework where boundary and interface conditions can be explicitly enforced. On the contrary, in problems with a fixed domain or a control volume, the equations of motion for a Newtonian fluid can be conveniently formulated in an Eulerian framework . ALE methods (Arbitrarian Lagrangian Eulerian methods), often used for these type of problems, represent an intermediate approach which proves useful in many circumstances. In the Lagrangian framework, the Navier-Stokes equations are formulated using reference coordinates which are updated at each time-step of the time-integration procedure [3]. The main difficulty in the Lagrangian approach is represented by the distortion of the mesh. If a fixed finite element mesh is used and the position of element nodes is updated as a consequence of the fluid flow, very soon the element distortion becomes excessive. A remedy which allows to avoid these distortions consists of systematically remeshing the volume of the problem.

Using a Lagrangian formulation the nonlinear convective term, typical of Eulerian formulations, disappears. However, when remeshing is introduced within the time-step solution algorithm, a new source of nonlinearity appears due to the fact that the discretized equations are written using matrix operators based on an unknown geometry. In the present work, we develop a Lagrangian finite element method for the analysis of Newtonian fluid flows based on a continuous remeshing, in the spirit of the so called Particle finite element method [1]-[2]. The key feature of this method is that not only the fundamental physical properties, such as density and viscosity, but also velocity and pressure, are assigned to particles, which are identified with element nodes. In this way, the element nodes represent a real part of the physical domain. To update the position of the particles at every time-step, velocity and pressure are computed solving the Lagrangian version of Navier-Stokes equations, discretized by finite elements.

An additional difficulty connected to the particle finite element approach, is represented by the fact that the fluid domain where the Navier-Stokes equations are to be solved is defined by the positions of the particles. At each time step the particles change their positions, and consequently the domain changes. Particles belonging to the boundary at a time-step, may not be on the boundary at subsequent time-steps.

This implies that a method to identify the external boundary is required to define the current integration domain and to impose the boundary conditions of the differential problem. To this purpose, use is made here of the so called ”alpha-shape method” proposed in [1]-[2].

The Lagrangian finite element approach introduced before is particularly suitable for fluid-structure interaction problem. In fact, if for the discretization of the structural part a classical finite element approach is used, the interaction terms can be computed easily. The sequence of operation carried out to account for the fluid-structure interaction is described below in the case of a 2D interaction where the solid part is discretized by four node quadrilateral elements:

1. consider two different domains: the fluid domain (characterized by the particles) and the solid domain (characterized by the quadrilateral finite elements);
2. superpose to the nodes of the solid domain which can come in contact with the fluid domain a set of fictitious fluid particles;
3. perform the Delaunay triangulation;
4. apply the alpha-shape method to remove the unnecessary triangles;
5. check whether the fluid is in contact with the solid part:
    - if the two discretized domains are not in contact the fluid and the solid analyses are performed separately without any interaction.
    - if the two discretized domains are in contact, a coupled analysis is necessary.

The coupled analysis is carried out using a staggered scheme as follows. The displacements of the solid part are computed first with the finite element structural analysis and are assigned to the fluid particles superposed to the solid nodes on the fluid-solid interface. The fluid analysis is performed under these assigned velocities at the boundary, the stress in the fluid part is computed and applied at the interface of the structural domain. This procedure is iterated until the convergence is reached in a time step.

REFERENCES

[1] R. Aubry, S. R. Ideloshn and E. Oñate “Particle finite element method in fluid-mechanics including thermal convection-diffusion”. *Comput. Struct.*, Vol. **83**, 1459–1475, 2004.

[2] S. R. Ideloshn, E. Oñate and F. Del Pin “The particle finite element method: a powerful tool to solve incompressible flows with free-surface and breaking waves ”. *Int. J. Numer. Methods Engrg.*, Vol. **61**, 964–989, 2004.

[3] R. Radovitzky and M. Ortiz “Lagrangian finite element analysis of newtonian fluid flows ”. *Int. J. Numer. Methods Engrg.*, Vol. **43**, 607–619, 1998.