Fluid Structure Interaction in LS-DYNA Using Lagrangian Interfaces, Automatic Re-meshing and Adaptivity

Facundo Del Pin and Grant O. Cook, Jr.
Livermore Software Technology Corporation
fdelpin@lstc.com

Abstract

The present work discusses a new Fluid Mechanics approach that will be introduced in future versions of LS-DYNA to solve incompressible flows. The objective of this new formulation will be to solve fluid-structure interaction problems using Lagrangian interfaces. In this way large deformations of structures are treated in a more natural fashion making it simpler to define the physical domain. Furthermore the proposed approximation will also deal with free surfaces and breaking waves. In a Lagrangian approach the mesh of the discrete problem moves together with the material particles. Thus for large deformations a very robust and fast re-meshing tool is being created. This tool will be incorporated in the software and all the re-meshing operations will be done automatically. Another key feature of this solver is that given an error estimator the re-meshing steps will also adapt the mesh to provide error control within the fluid solver.

Introduction

Incompressible Newtonian fluid is one of the most common types of fluids encountered in industrial problems. Examples of applications range from car hydrodynamics to bio-mechanical applications for the solution of blood flows in the human body. For these reason extensive studies have been carried out to find an accurate and fast way to deal with this kind of problems. At the present time a large number of commercial software has been developed that provide an extensive spectrum of solutions using different approaches and arising to acceptable results. Nevertheless new sophisticated problems are found that narrow the options and applications of the existing commercial technology. Such is the case of incompressible flows interacting with flexible structure under large deformations, multi-face problems, reactive flows, thermal coupling, noise problems and any combination of all this options.

The present approach will focus on the fluid-structure interaction of large deformations. All discrete interfaces will be treated as Lagrangian meaning that they will move following the mechanics of the physical problem. All the rest of the elements will move in such a way to minimize the perturbation of the element quality. The basics of this numerical technique [1,2,3,4] is a very fast and robust mesh generation tool that provides good quality tetrahedral elements and an accurate incompressible flow solver.

Automatic Mesh Generation

The mesh generation part of the solver is based on a Delaunay Algorithm. The main idea is to minimize the user input that is required to obtain an acceptable mesh for the FEM approximation. Provided with a surface mesh from LS-PrePost, the Delaunay Mesher will take the different parts of the surface and build a volume mesh. The parts have to define at least one closed surface. Apart from the closed surfaces other open surfaces may be inserted in the model to change the internal size of the tetrahedral mesh (see Fig. 1). The last image in Fig. 1 shows the partition obtained if a parallel computation will be carried out. One of the main objectives of this work will be to provide an MPP version of the fluid mechanics code. In a future work an MPP version of the mesh generator will also be provided.

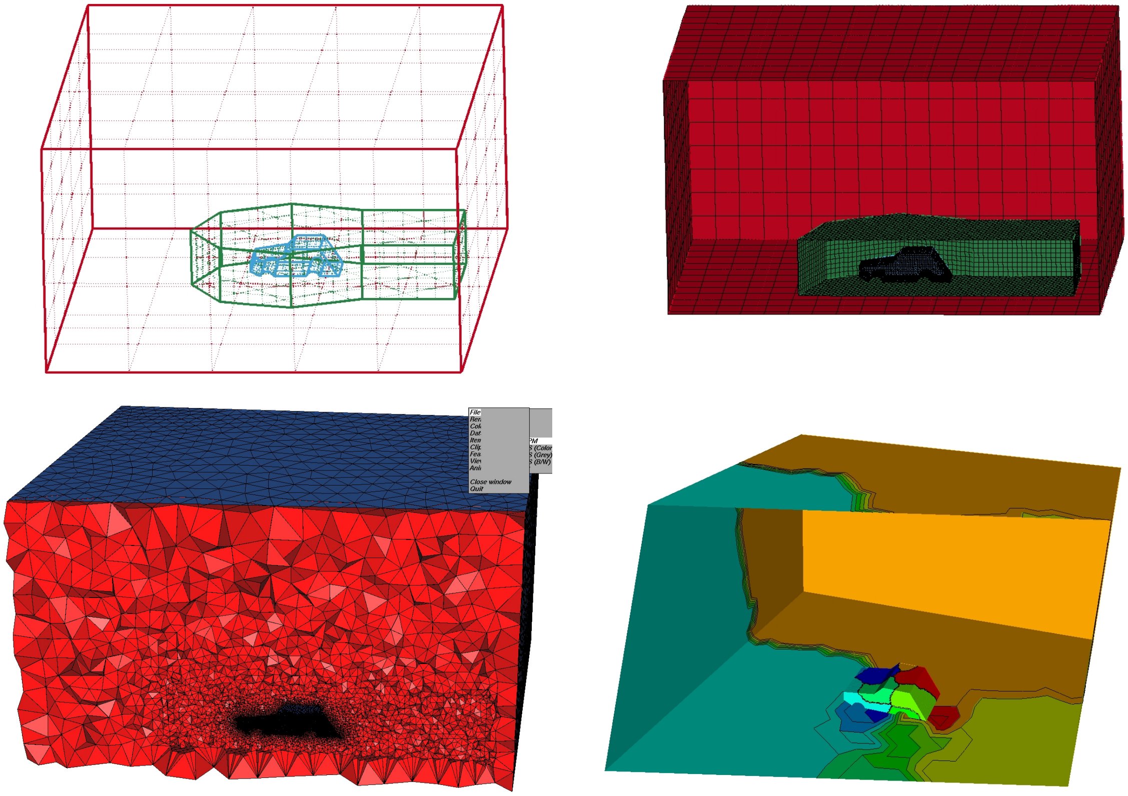

Figure 1: Sequence of steps for the mesh generation of a generic car model. The final volume mesh consists of 1M tetrahedral elements. It took one minute to generate in a standard PC computer. The minimum dihedral angle is 6 degrees and the minimum average is 74.2 degrees.

Fluid Mechanics Solver

For the fluid mechanics solver the Navier-Stokes equations for incompressible flows are solved. The space integration is carried out using a standard stabilized P1-P1 element and the integration in time will be based on a projection scheme [4,5,6,7]. In Fig. 2 an example of application is displayed.

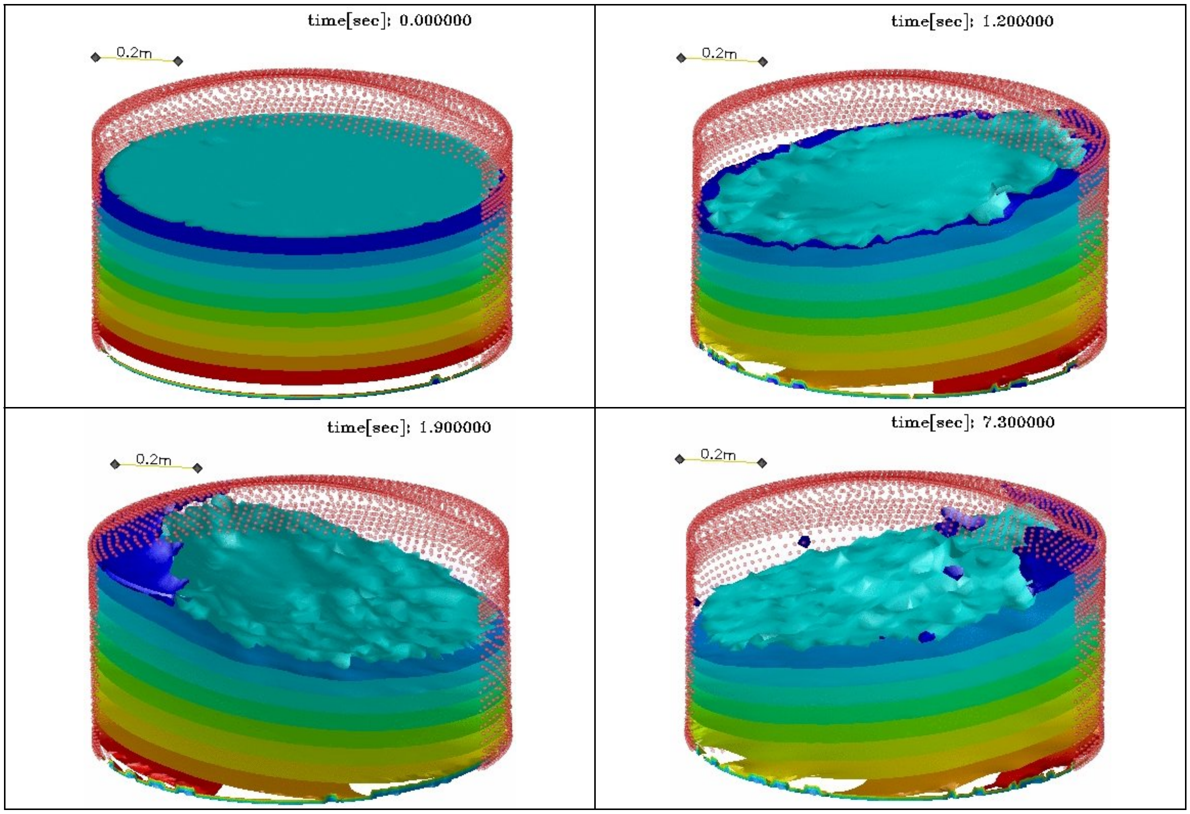

Figure 2: Example of large boundary displacements that impose large deformations in the fluid domain. In this case a free surface has to be computed due to breaking waves inside the sloshing tank.

Adaptivity

The adaptivity process will take the mesh generated as described in section 2 and the results obtained from the mechanics equations and estimate areas of large errors and lower errors. The maximum error could be a user input data. In this way the mesh will be adapted to provide a constant error over the whole domain. Some times the error is known a priori and the mesh is built accordingly from the beginning of the computation. But there are other problems where the mechanics are not so well established and thus an adaptivity process will automatically build a proper mesh for the specific type of flow. This process is described in Fig. 3 using as an example 6 rotating propellers.

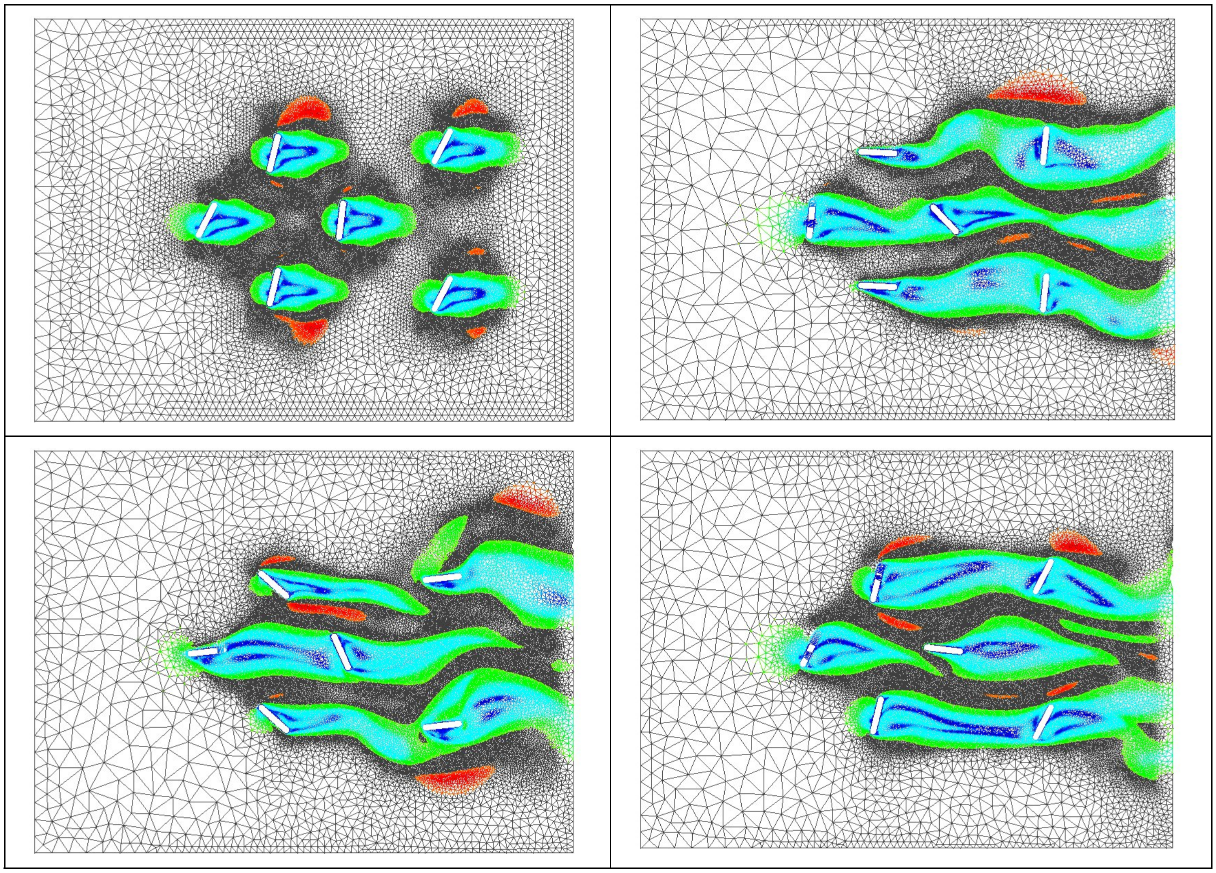

Figure 3: this figure depicts the use of adaptivity for highly complicated flow patterns.

Conclusions

We believe that the present algorithm will incorporate some new features to LS-DYNA in the field of fluid-structure interaction when large deformations take place. The use of automatic re-meshing will simplify the meshing task minimizing user inputs. An adaptivity process will also be integrated in the solver to provide error control in the fluid mechanics solver.

References

1. Idelsohn, S.R., Onate, E., Calvo, N. and Del Pin, F. (2003a). The meshless finite element method. Int. J. Num. Meth. Engng., 58, 6: 893--912.

2. Idelsohn, S.R., Onate, E. and Del Pin, F. (2004). The particle finite element method a powerful tool to solve incompressible flows with free-surfaces and breaking waves. Int. J. Num. Meth. Engng., submitted.

3. Idelsohn, S.R., Onate, E. and Del Pin, F. (2003b). A Lagrangian meshless finite element method applied to fluid-structure interaction problems. in Computer and Structures, 81: 655--671.

4. Idelsohn, S.R., Onate, E., Del Pin F. and Calvo, N. (2002). Lagrangian formulation: the only way to solve some free-surface fluid mechanics problems. Fifth World Congress on Computational Mechanics, Mang HA, Rammerstorfer FG and Eberhardsteiner J. (eds), July 7--12, Vienna, Austria.

5. A.J. Chorin, A numerical solution for solving incompressible viscous flow problems J. Comp. Phys., 2, 12--26, 1967.

6. R. Codina and J. Blasco, Stabilized finite element method for the transient Navier-Stokes equations based on a pressure gradient operator, Comput. Methods in Appl. Mech. Engrg., 182, 277--301, 2000.

7. R. Codina, Stabilized finite element approximation of transient incompressible flows using orthogonal subscales, Comput. Methods Appl. Mech. Engrg., 191, 4295--4321, 2002.

8. R. Codina and O. Soto, Approximation of the incompressible Navier-Stokes equations using orthogonal-subscale stabilization and pressure segregation on anisotropic finite element meshes, Comput. Meth. Appl. Mech. Eng., to appear.