An Implicit Incompressible CFD Solver in LS-DYNA® for Fluid-Structure Interaction Problems

Facundo Del Pin
Livermore Software Technology Corporation

Abstract

The present work is an introduction to a new CFD solver in LS-DYNA®. This solver is part of the efforts put in LS-DYNA with the objective to expand the capabilities into new challenging problems in industry. The new CFD solver will focus on fluid-structure interaction (FSI) applications where incompressible fluids interact with the existing structures in LS-DYNA. Incompressible flows are present in a great variety of industrial applications from sloshing problems to aerodynamics at low Mach numbers.

This new incompressible CFD solver coupled to LS-DYNA mechanics will provide an implicit time integration scheme allowing larger time steps and faster convergence to steady state. One of the main features of the solver is the Lagrangian representations of all FSI interfaces providing exact imposition of boundary conditions. In this way the fluid mesh and the solid mesh are tightly coupled such that the fluid domain deforms following the Lagrangian structure displacements. The rest of the domain follows an Arbitrary Lagrangian-Eulerian formulation. The image bellow shows the mesh movement and the conformity of the fluid mesh to the solid mesh.

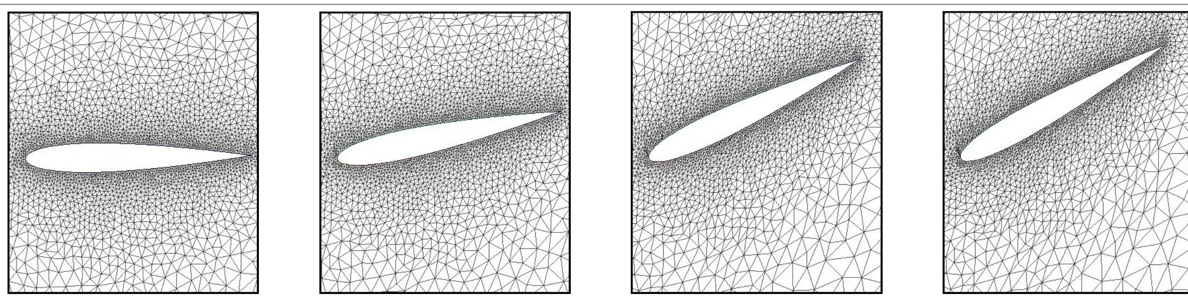

Fig. 1: Mesh movement with interface nodes attached to the Lagrangian structure.

In the case of large deformations the mesh may suffer extreme distortions that in some cases will lead to meshes with bad element quality or even element inversions (negative Jacobians). The solver provides a tool to automatically re-mesh the fluid domain. For this type of problems the user will have to provide an initial boundary mesh that the fluid mesher will use to constrain it's boundaries and re-construct the volume mesh. The user provided boundary mesh will deform following the structure displacements and may even be re-meshed and adapted. Since a re-meshing tool is available an error estimator may also be employed to provide a mesh with a constant error within user specified limits. The figure bellow shows a problem of a flow past a moving sphere where the size of the mesh has been adapted to local error estimates.

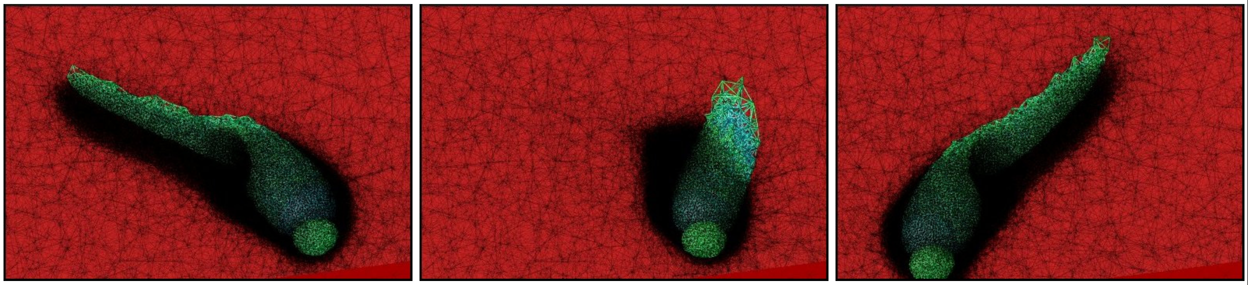

Fig. 2: Adapted mesh around a moving ball with Re=800.

When the mesh displacement becomes so large that it compromises the quality of the elements and the mesh size distribution a full re-meshing step will automatically take place. The image bellow depicts a flexible valve immersed in a incompressible flow. The problem set up follows the conditions that my be found in a human heart.

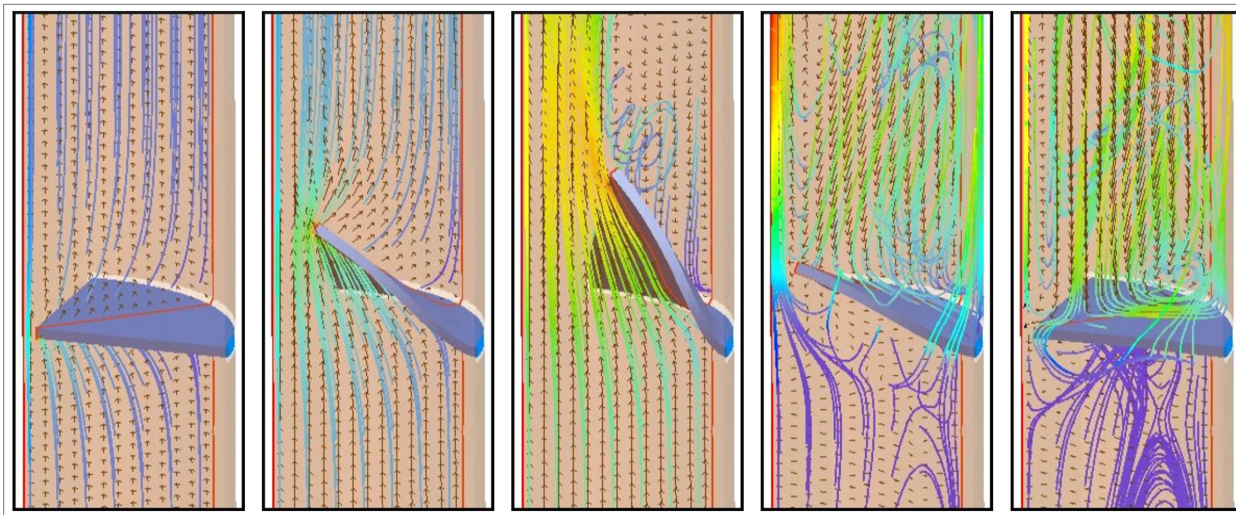

Fig. 3: Streamlines colored with the velocity field for the case of a flexible valve undergoing large deformations.

The CFD solver will also include the solution of flows with free-surfaces where the surface may be represented as a Lagrangian interface or using a Eulerian mesh and an interface tracking technique. Some applications for this features will include sloshing tanks and flows with breaking waves, fragmentation and coalescence of fluid. The solver will also support flow mix without chemical reactions. All these features will always be coupled to the solid solver.