Advances on the Incompressible CFD Solver in LS-DYNA®

Facundo Del Pin
Livermore Software Technology Corporation

Abstract

The present work will introduce some of the resent developments in the Incompressible CFD (ICFD) solver currently under development in LS-DYNA. The main feature of this solver is its ability to couple with any solid model to perform Fluid-Structure interaction (FSI) analysis. Highly non-linear behavior is supported by using automatic re-meshing strategies to maintain element quality within acceptable limits. In this work we will introduce the additional features for conjugate heat transfer, turbulence model, biphasic flow, some new feature in terms of mesh generation like boundary layer meshing and MPP.

Introduction

Incompressible flows cover a vast number of engineering problems ranging from car aerodynamics to arterial flows and parachute simulation. As a rule of thumb a flow may be considered incompressible if the Mach number presented in any part of the domain is not larger than 0.3. In LS-DYNA there are other two options to do CFD depending upon the kind of problem. The CESE solver is highly accurate CFD solver for compressible fluids. The ALE solver has support for both compressible and incompressible and it is a good option for highly transient problems.

Due to the requirement of industrial applications a number of new features have been added to the ICFD solver prior to the release version. They will be briefly described bellow.

Turbulence Models

The majority of the problems that involve real life applications fall in the category of high Reynolds number problems, where turbulent effects play an important role. Since full resolution of the problem is not possible due to resource limitations robust turbulent models are critical to provide realistic results. In ICFD turbulence is and will continue being a work in progress due to the continue evolution of the field. At the moment three classical approaches have been incorporated namely K-e model, Smagorinsky LES model and a variational multiscale model which is still part of research work. The user will be able to modify some parameters of the models from the input deck to adjust it to some particular problem.

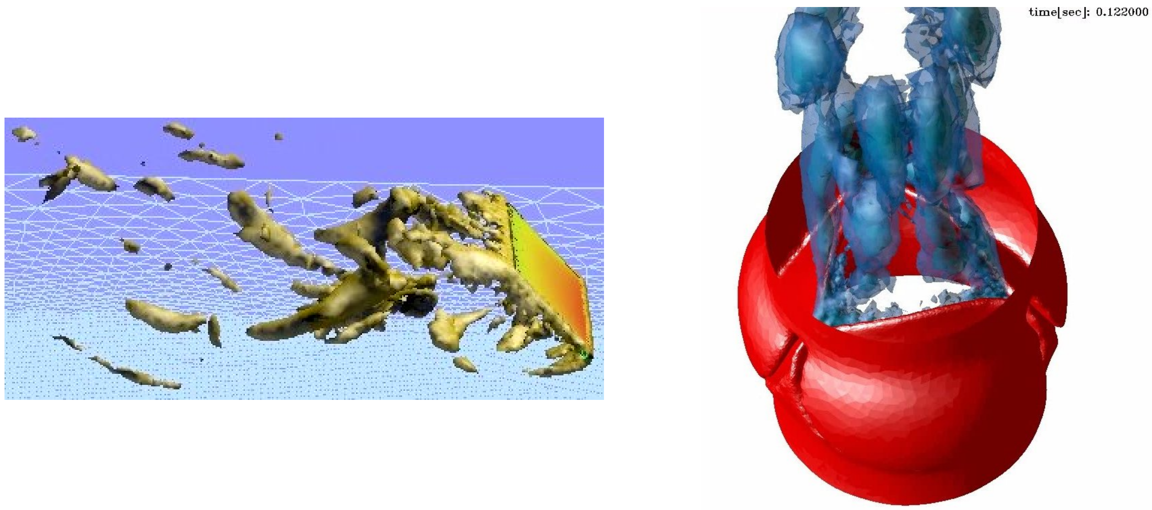

Figure 1: the image shows a flow past a flat plate in an angle on the left, simulating the wind effect on a solar panel. The image on the right is an FSI problem of a opening valve and the flow is shown and maximum aperture when the flow rate peaks. In both cases the turbulence was modeled using a LES approach.

Conjugate Heat Transfer

Problems involving heat conduction in solid material have been part of LS-DYNA for a long time. Now we can couple the solid material to the fluid in an implicit way to solve conjugate heat transfer problems. This kind of coupling will also allow to solve thermo mechanical problems, providing a two-way coupling between the fluid and the structure to solve fluid-structure interaction problems.

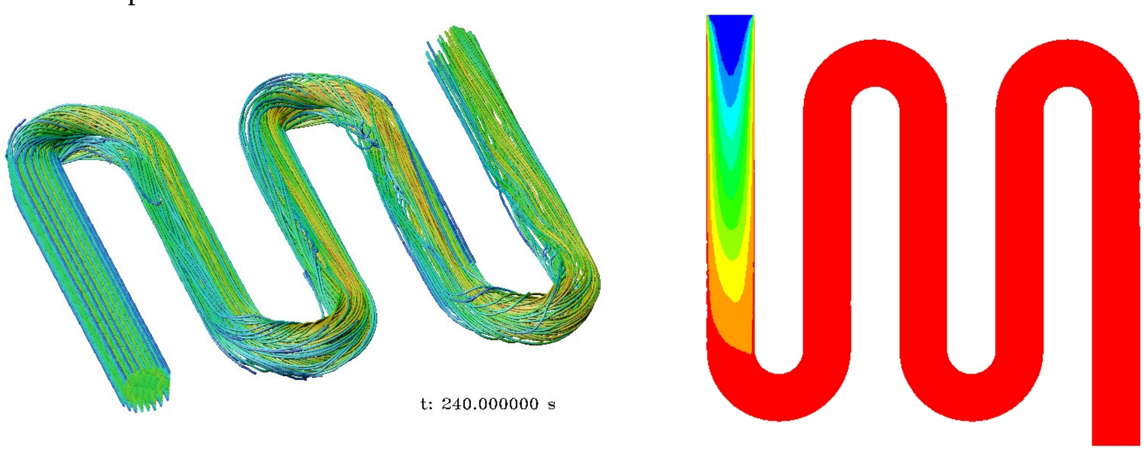

Figure 2: in this problem a cooling serpentine is used to cool down a solid part (not shown in the picture). On the left image the streamlines of the flow are shown after the flow has reached steady state. The image on the right shows the flow temperature profile at steady state.

Multi-Phase Flow

One of the new additions to the solver is the possibility to approximate two phase flow while preserving a sharp interface even for under resolved problems. This approach will preserve sub-grid features of the interface in areas where the finite element mesh is under-resolved and restore this features to the solver later in areas of the mesh with better resolution.

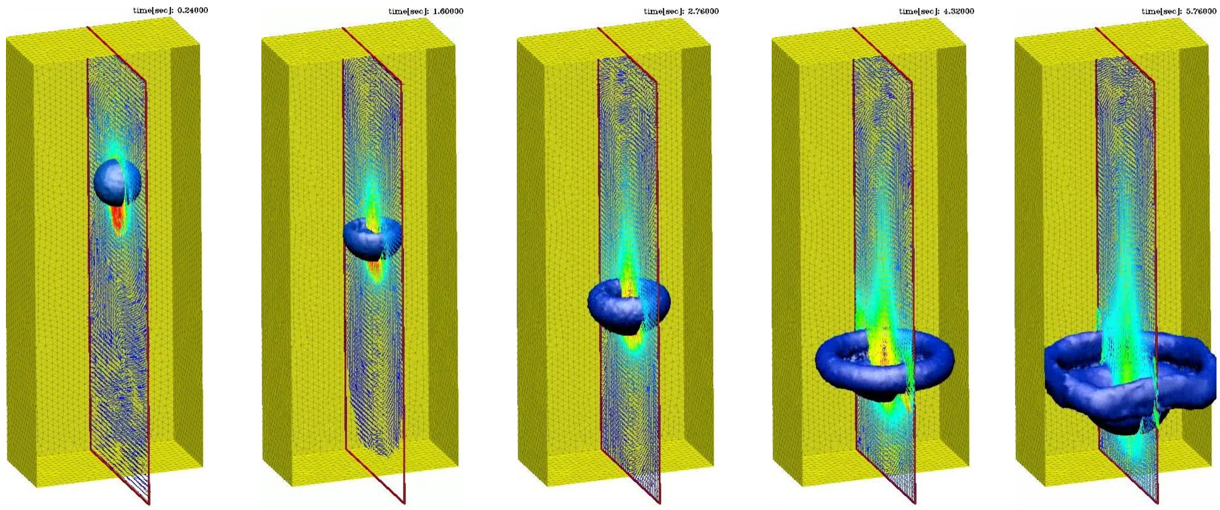

Figure 3: the image series show a bubble dropping into a lighter fluid. The mesh on the surface gives an idea of the mesh resolution used. In despite of the coarse mesh the interface is well captured.

Boundary Layer Meshing

In terms of volume meshing the solver supports automatic mesh generation which is perform internally at run time. The mesh may also be modified by an error estimator to do adaptive mesh refinement. Recently a boundary layer mesher was added to the solver to further resolve the areas of the domain close to the walls to provide a better approximation of the shear stresses. In problems involving drag calculation boundary layer meshing is a requirement. This kind of problems involve aerodynamics of bluff bodies like the one shown in the picture bellow.

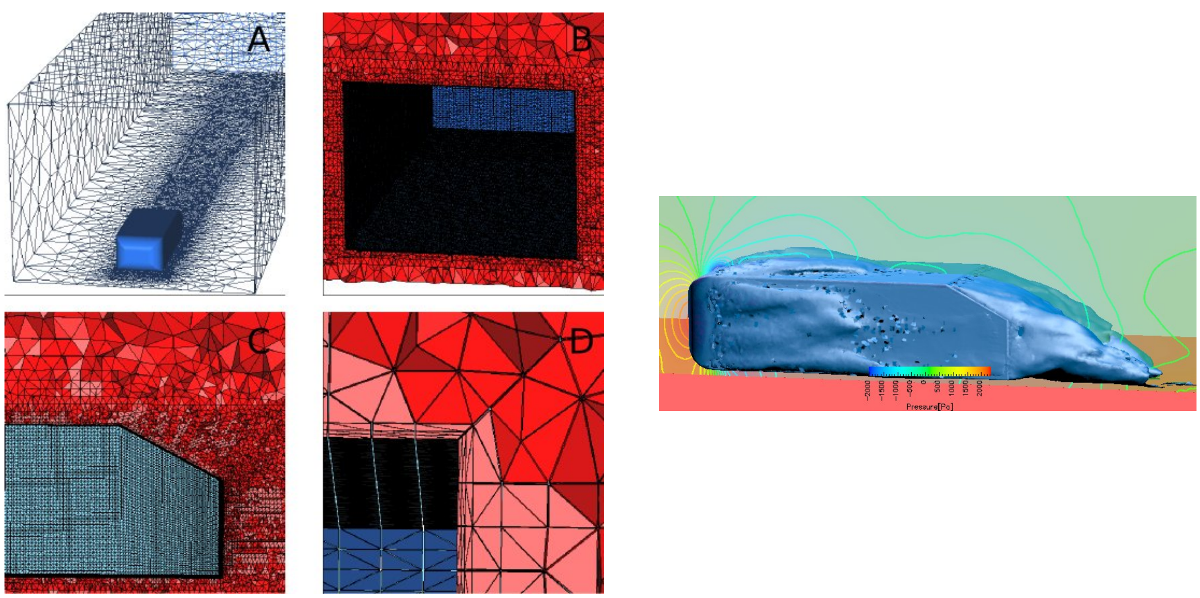

Figure 4: this is a classical problem to benchmark drag around bluff bodies. It is called the Ahmed body problem. On the left a detail of the mesh is show. Image A shows the full domain with mesh refinement in the wake of the body. B and C are a closer look at the mesh around the body and C shows the boundary layer mesh next to the body wall.

Parallel Computing

To improve the productivity of the solver and to satisfy the demand for high performance computing all the features of the solver have been implemented in parallel. The parallel CFD solver can be coupled to the parallel solid mechanics solver to do parallel FSI. In the same way the coupling may be done in parallel with the thermal solver to do conjugate heat transfer.