ICFD: Summary of Recent and Future Developments

Facundo Del Pin, Iñaki Caldichoury, Rodrigo R. Paz and Chienjung Huang
Livermore Software Technology Corporation

Abstract

Since its release in R7 the Incompressible CFD solver (ICFD) has been rapidly improving and increasing its functionality. In this paper a summary of the latest and current developments will be presented. The focus will be on four topics. First the steady state solver and its coupling capabilities for fluid-structure interaction (FSI) or conjugate heat transfer (CHT) will be presented. In second place the recent modifications to the boundary layer mesh generation will be introduced where some default parameters have changed. The possible implications of these changes in the solution will be mentioned. Third a short introduction to coupling ICFD with LS-OPT® for shape optimization will be presented. The idea is to use ANSA to morph the surface mesh driven by LS-OPT to provide an optimal solution. Finally some of the current developments will be enumerated like immersed interfaces, periodic boundary conditions, porous media through shell elements for parachute simulation, etc. These developments will be part of future LS-DYNA® releases.

Introduction

The demand for coupled multiphysics analysis has been steadily increasing in the past few years. In the wake of this many commercial solvers have rushed to provide engineers with solutions which many times carry the load of a stiff legacy implementation. Thus codes that were originally designed to deal with purely CFD analysis face the challenge of connecting their results to other solvers sometimes for other vendors to produce multiphysics results. This ends up in a cumbersome process of co-simulation which leaves much of the coupling burden to the engineer.

In LS-DYNA the ICFD module has been designed from the beginning to be a multiphysics solver that provides accurate and scalable CFD solutions and at the same time integrates easily to the other physics modules in LS-DYNA. Continuous work is devoted to integrate to newer modules, improve the coupling algorithm, increase efficiency and simplify the coupling process without assuming that the user is a CFD or Structural analysis engineer.

In this paper some of the latest advances in the implementation of coupled problems are presented. In particular the benefits of having a steady state solution will be discussed. It will be shown that steady state either Navier-Stokes or Potential flow solutions could significantly reduce the cost of rapid prototyping by maintaining a reasonable level of accuracy in some problems. The ability of LS-DYNA to seamlessly transition from a steady state CFD solution to a structural analysis greatly reduces the cost and complexity of non-linear solutions. Similar ideas will be shown for Conjugate Heat Transfer (CHT) analysis. The second part of this paper will show a simple but valuable optimization problem achieved using LS-OPT, ANSA and LS-DYNA for the shape optimization of a ground vehicle with the idea to improve the Down-Force to Drag ratio. The last part of the paper will deal with new developments that are expected to be available in future releases. Among these developments are periodic boundary conditions, sliding and overset mesh and immersed interfaces.

Steady state solutions and multiphysics coupling

One of the largest complexities of coupled solutions are non-linear effects which many times are responsible for reducing scalability and increasing the cost of simulations. Non-linear coupling is a highly complex phenomena that requires deep knowledge of the underlying physics of the problem to model them correctly. Nonetheless in some kind of problems or at some stages of the design process linearization could be a good compromise between accuracy, computational time and model complexity. Fortunately LS-DYNA has the right tools for both types of analysis. In this section we will focus on a linearization process that could be applied for coupled Fluid Structure interaction (FSI) analysis or CHT. The coupling arises from the forces, velocities or temperature fluxes computed in the CFD portion of the model at steady state. These field values could be seamlessly transferred to the structural part of the model during the same run. Furthermore by using the keyword *ICFD_DATABASE_DRAG the user is able to write a file in LS-DYNA format which could be included to the structural model to reproduce the fluid solutions as many times as needed without actually running the CFD model. This feature is highly valuable at the time of tuning the structural model, material properties, thickness, etc.
Two examples will be presented to illustrate these ideas in the field of FSI and CHT.

Fluid structure interaction on a ground vehicle
The first example addresses the FSI analysis of a ground vehicle where the roof panel deformation will be studied. The sketch of the model is shown in figure 1.

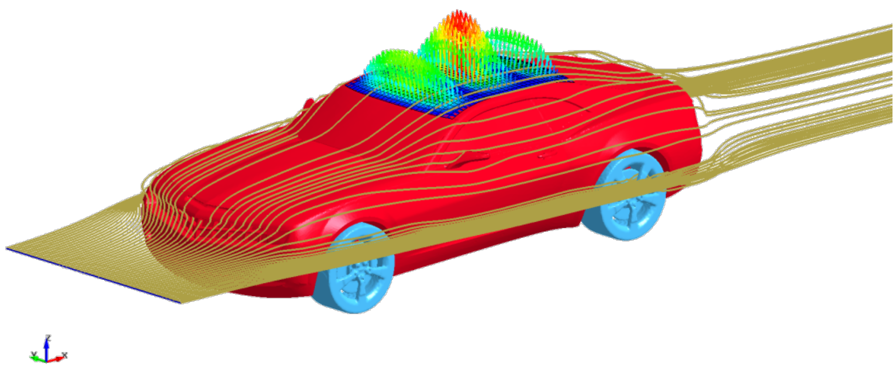

Figure 1. Vehicle model used to study the structural deformation of the roof panel.

This model has been solved using three different approaches. The first approach is a non-linear transient coupling where the structural and fluid solver are strongly coupled and the solution is performed implicitly. This should results in the most accurate type of coupling and thus it should be considered as the reference solution. The second approach uses a steady state potential flow solver with a subsequent non-linear structural solution at the end using the pressure field from the potential flow solution. Using the potential flow solver assumes that the flow in the region of the domain under consideration is attached to the walls and laminar. In the real case the flow is of course not laminar but it is attached and so the pressure values are expected to be within a reasonable range to the Navier-Stokes solution to be used in the prediction of the structural displacement. In the third approach the force computed in the non-linear coupling is written to an LS-DYNA input deck for each load segment in the structural model. So just by including this file in the structural input deck a user can run s structural only analysis using the force computed in the expensive non-linear analysis. This results in a very fast run that allows engineers to easily change and test their designs until a model is ready for a new non-linear run. Figure 2 shows a comparison of the flow velocity for the Potential Flow vs Navier-Stokes solver.

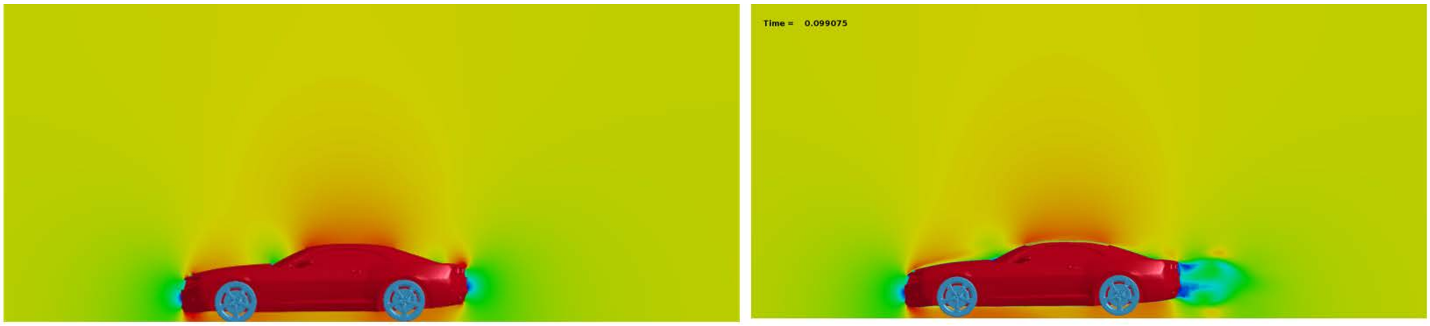

Figure 2. Velocity comparison for the Navier-Stokes and Potential flow solver.

The area of interest is the roof of the vehicle. In Figure 3 the result of the three different approaches is depicted. It is observed that the three solutions predict very similar displacements. It is also interesting noting the running time for each approach which is the reason that balances the potential lack of accuracy of the simpler models.

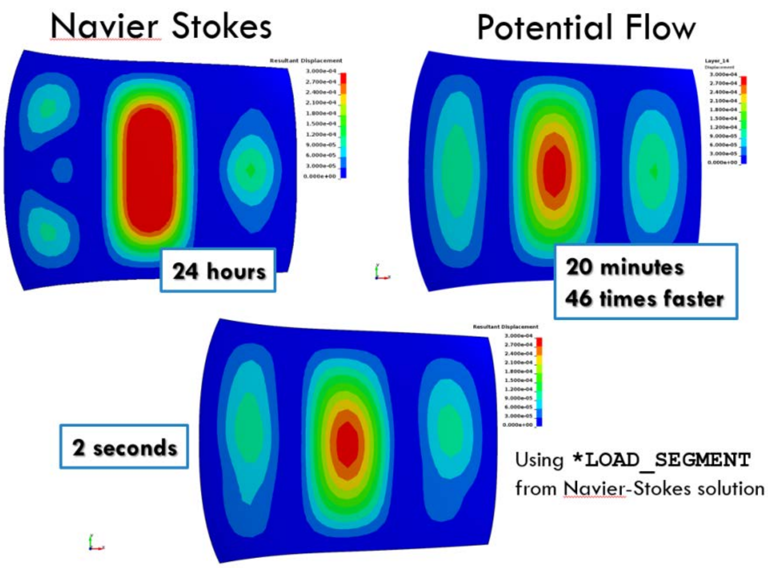

Figure 3. Structural displacement for each approach and running time.

Cooling problem

This second example involves a Conjugate Heat Transfer problem where fluid is used to cool down a tool in a stamping simulation. The setup of the model is sketched in Figure 4.

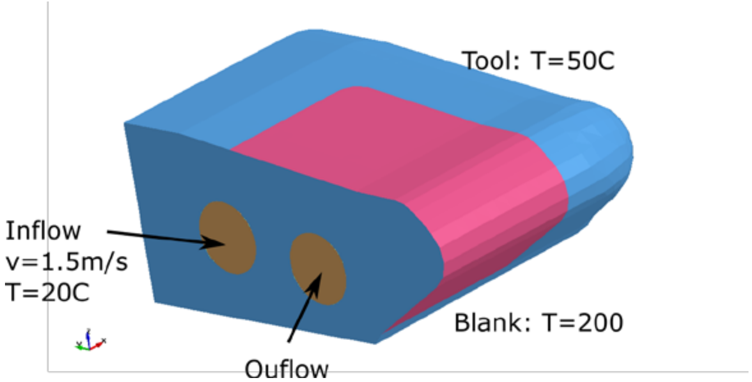

Figure 4. Sketch of the model used for cooling simulation.

In cooling problems fluid flows in a pipe inside a die at a lower temperature than the die is. As the fluid runs along the pipe it will slowly heat up as the die cools down. In the absence of a heat source the fluid and the die will reach the same temperature as the fluid inflow temperature. The velocity of the flow plays an important role in maintaining the appropriate temperature profile inside the pipes. In Figure 5 the velocity profile for the Navier-Stokes and the Potential flow solver are compared. The largest differences are at corners and areas of detachment/recirculation.

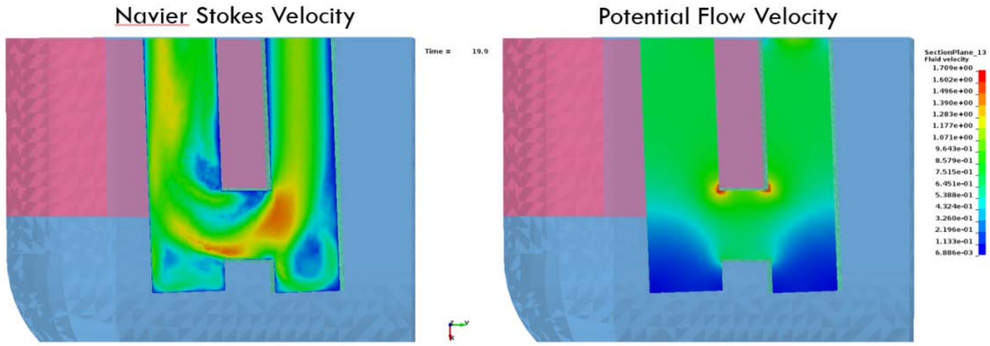

Figure 5. Velocity comparison for Navier-Stokes and Potential flow solution.

In Figure 6 the temperature profiles are shown for both Navier-Stokes and Potential flow solutions. Both profiles are in very good agreement.

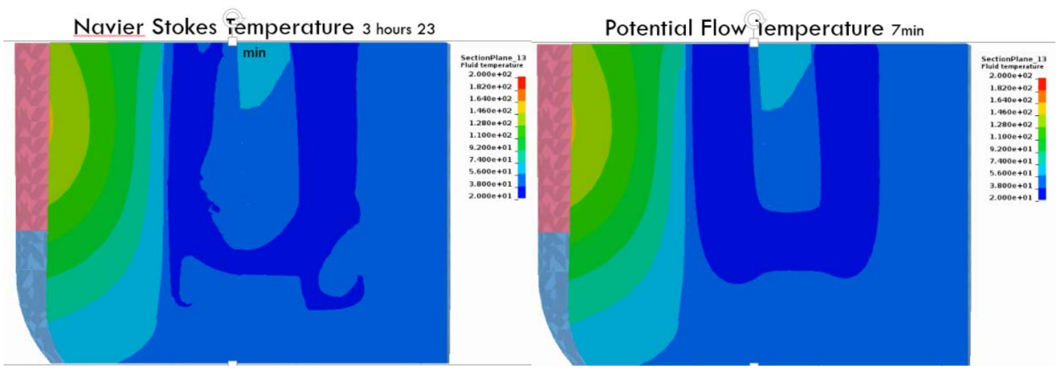

Figure 6. Temperature profile at steady state for Navier-Stokes and Potential flow solver.

Shape Optimization using LS-OPT

One of the latest achievements related to ICFD has been using LS-OPT and the meshing package ANSA (by BETA-CAE) for shape optimization. The idea is to modify an initial geometry to optimize a functional. For instance in the case of ground vehicle aerodynamics the vehicle geometry is changed to maximize the down-force to drag ratio. In Figure 7 an initial geometry of a generic vehicle is presented. The optimization process will focus on the tail of the vehicle.

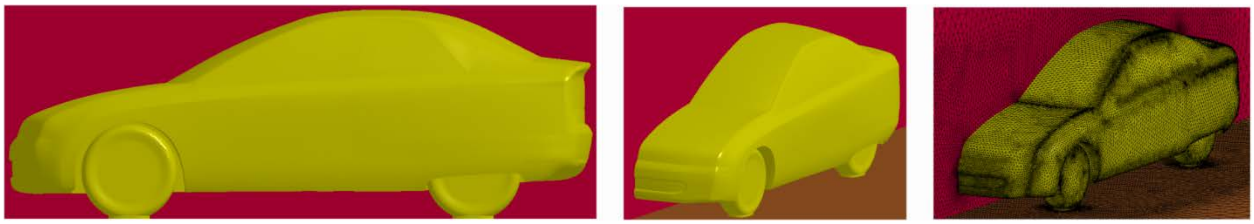

Figure 7. Initial geometry for the optimization loop.

The idea is to use ANSA morphing capabilities to change the geometry using some parameters obtained from LS-OPT. Figure 8 shows the morphing boxes and the parameters to modify together with the LS-OPT optimization loop.

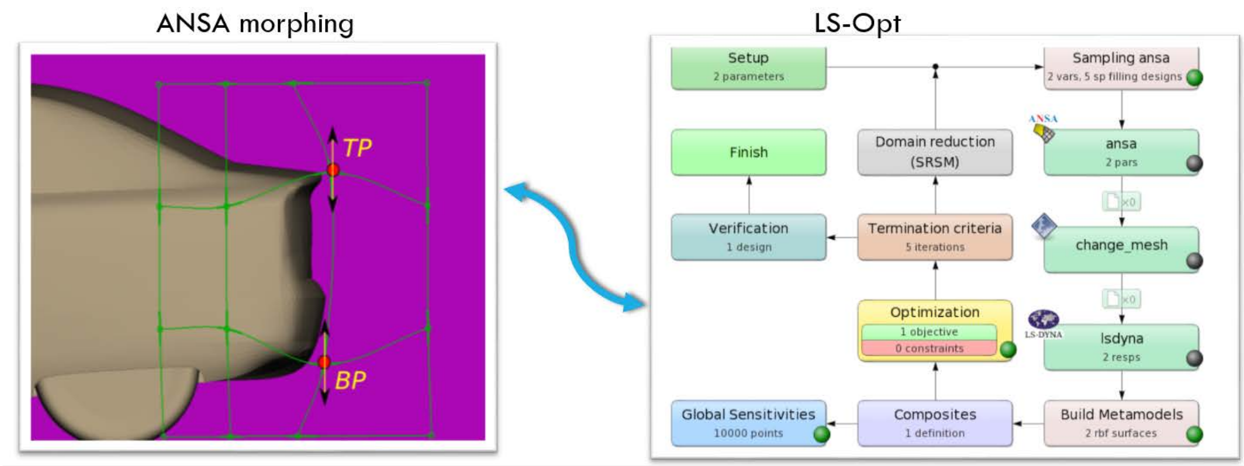

Figure 8. Morphing boxes and parameters.

One of the benefits of using LS-DYNA for the optimization process is that the volume mesh is built at run time. So the morphing needs to take place only on the surface of the vehicle greatly simplifying the process. Morphing a volume mesh is expensive and less reliable especially in the presence of boundary layer mesh. The final results or shown in Figure 9 which show the predicted and most optimal configurations.

Optimal design for (Down Force)/Drag ratio.

	Initial design	Optimal design (6.1)	„Best“ design (5.2)
bp	0.0	0.022	0.062
tp	0.0	0.105	0.132
LD predicted	-	0.00376	0.00366
LD computed	0.00094	0.00364	0.00384

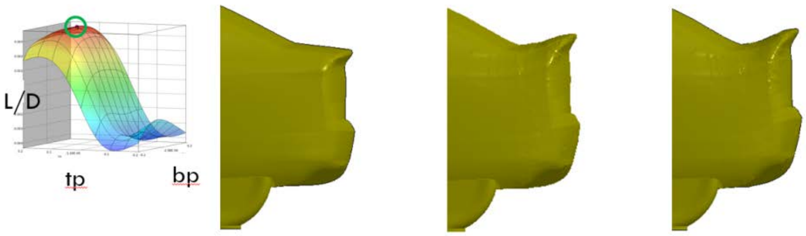

Figure 9. LS-OPT predicted and most optimal results.

New Developments

In this section some of the current latest developments will be briefly presented. These developments are currently not part of the release version of LS-DYNA although a beta version is expected at some point this year.

Periodic Boundary Conditions

Periodic boundary conditions are often used in numerical methods to represent a large domain by modeling just a fraction of it. These boundary conditions are heavily used in rotating systems like in turbomachinery. The implementation involves the use of linear constraints that guarantee the continuity and conservation of flow across the periodic boundaries. Figure 10 is a setup example of a problem with periodic boundaries.

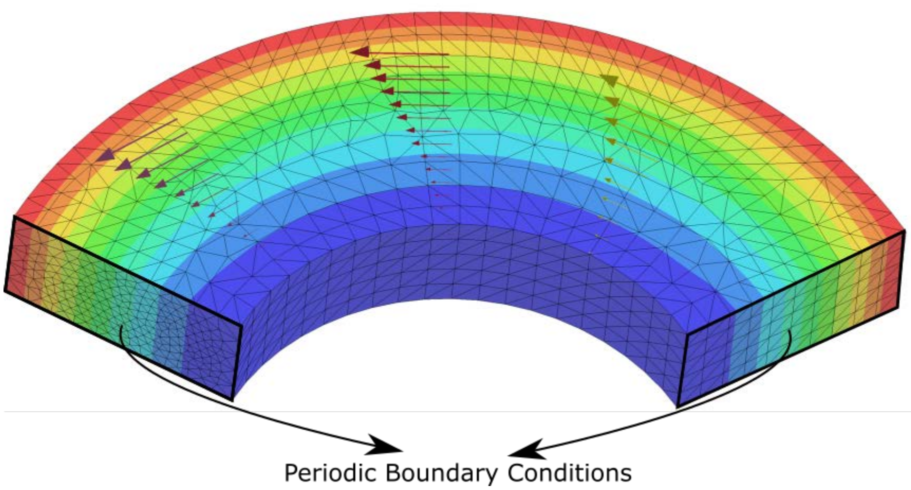

Figure 10. Periodic boundary conditions problem setup. It is important to note that the mesh for the boundaries with periodic conditions do not need to match.

Sliding mesh

Sliding mesh is a technique that allows the simulation of transient rotating mechanisms without re-meshing. When sliding meshes are used typically the domain is split into at least two volume meshes. One mesh will have the rotating components and the other the rest of the domain. The interface between the two volume domains is the sliding mesh. The solutions on both domains are solved simultaneously by using linear constraints. Figure 11 shows an application example.

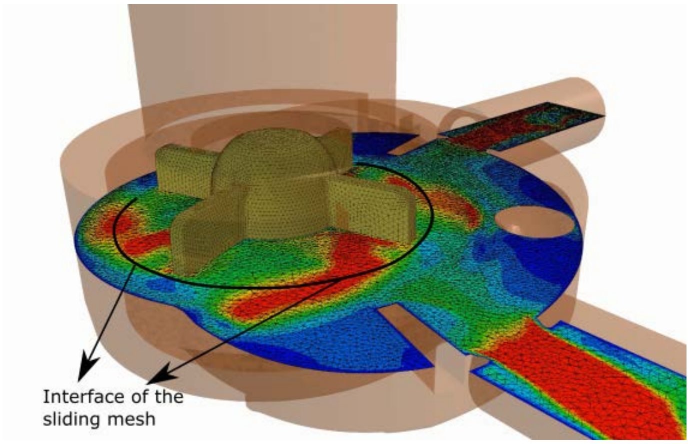

Figure 11. Sliding mesh application example. The black line indicates the interface between the rotating domain and the rest.

Immersed interface

Immersed interfaces simplifies the pre-processing of complex geometries. The goal is to have a hybrid immersed interface / body fitted mesh depending on the types of flow in different parts of the model.
This new approach is based on discontinues finite element approximations which provide sharp interfaces and allow structural contact.
A typical example is shown in Figure 12 which involves a lobe pump where the contact between the lobes makes it challenging for the more classical re-meshing approach.

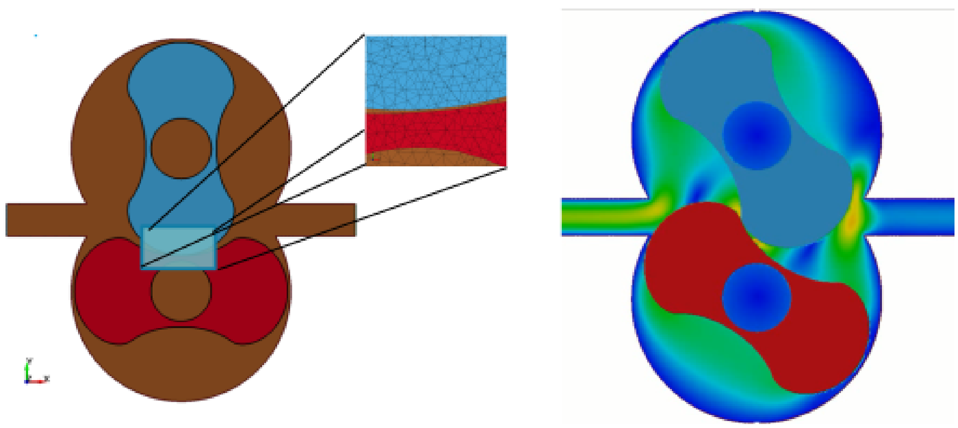

Figure 12. This image shows the lobe pump geometry on the left with enlarged region that shows the mesh and the contact between the components. On the right is a velocity profile.