Review and Advances of Coupling Methods for the ICFD solver in LS-Dyna

Facundo Del Pin, Iñaki Caldichoury and Rodrigo R. Paz

Livermore Software Technology Corporation

Abstract

The ICFD solver in LS-Dyna specializes in the solution of Incompressible fluid flows. The main goal is to accurately predict the values of pressure and velocity subject to the constraint that Div(v)=0 where v is the fluid velocity. One area of applications is the prediction of lift and drag in aerodynamics of external flows, detachment or recirculation for internal flows, etc. These are what could be regarded as typical CFD applications. The real potential of the ICFD solver arises when it is coupled to other solvers within LS-Dyna. In this paper the numerous existing coupling methods available for the ICFD solver will be reviewed. Also in this paper the latest advances regarding coupling methods will be presented. In terms of coupling ideas the following will be discussed: Fluid Structure Interaction (FSI) weak and strong, Conjugate heat transfer weak and strong and the addition of electromagnetism as part of the coupled solution. Also the coupling with the Discrete Element Method (DEM) will be introduced enumerating the different features available in the coupling process. Finally a new variation that incorporates the steady state solver coupled to a static linear or non-linear structural solution will be presented.

1 Introduction

The Incompressible CFD solver in LS-Dyna specializes in the solution of fluid flows at low Mach number (Ma<0.3) using implicit time integration. Although it's main objective is the accurate solution of flows it was designed from scratch to provide a framework for the solution of coupled multiphysics problems. This enables the ICFD module to incorporate or couple to the fluid problem the solution of other modules in LS-Dyna such that the solution of the coupled system differs from the solution of the modules when solved independently from each other.
Coupled problems may come in different flavors: strong coupling, weak coupling, one-way or two-way coupling, indirect coupling and monolithic coupling. In the next section the different types of coupling mechanisms that are available will be described. Those mechanisms will be applied to three modules in LS-Dyna namely the structural mechanics solver, the thermal solver and the Discrete Element Method (DEM) solver. The subsequent sections will provide details on how these modules interact with the ICFD module when the coupling occurs.

2 Types of coupling problems

A coupled solution that involves the interaction between two modules will typically involve the exchange of some sort of information at the coupling interface. Indeed the way that the interface is treated is key for an accurate solution of the system. The interface treatment will be dictated by the type of problem that is intended to be solved, the accuracy needed for the solution and the computational power available. Bellow there is an explanation of the different types of coupling and how they could be combined.

2.1 Strong, weak and monolithic coupling

Basically strong coupling means that within one time step two modules will iterate exchanging information at the interface until the residual of the interface is bellow a given tolerance. On the other hand weak coupling means a single exchange of information between the modules per time step. The case of monolithic coupling is a step further from strong coupling and it means solving in a single iteration all the degrees of freedom of all the systems coupled by resolving a single linear or non-linear matrix system. For some problems like aero-elasticity if the time step is sufficiently small the residual left at the interface does not greatly affect the final solution. The same happens in linear problems for which weak coupling is a good option. Since a single sub-iteration is performed weak coupling is generally much faster to solve than strong coupling and if the problem allows it this should be the preferred option. It is very important to pick the correct coupling. If in doubt strong coupling will generally provide more accurate and robust results at the expense of more computational intensive solutions.

2.2 One-way or two-way coupling

When two modules are exchanging information at the interface then one-way coupling means that only one module is transferring information to the other. An example where one-way coupling may be used is when rigid bodies move with an imposed velocity inside a fluid. The rigid body velocity will be transferred to the fluid to compute a new pressure but the fluid does not need to transfer forces back to the rigid body since doing that does not change the state of the rigid body. Applying one-way coupling can save computational time without affecting the solution.
On the other hand two-way coupling means that both modules exchange information and that the result of one will change the state of the other and vice versa. This is the most classical type of coupling which may be computed using weak or strong coupling.

2.3 Indirect coupling

The last type of coupling is the indirect coupling. This occurs when two modules are connected through a third module. An example of this is the case of the electromagnetism module heating up a coil through joules heating which in turns heat up fluid around it. The fluid solver and the electromagnetism solver are indirectly coupled through the structural thermal solver.

3 Coupling to the structural mechanics solver

The coupling of CFD with structural solvers is generally referred as fluid structure interaction (FSI). The structural mechanics solver in LS-Dyna has two time integration schemes: explicit and implicit. The ICFD solver can couple to both of them but the type of coupling will differ. The coupling between ICFD and the explicit mechanics solver results in a weak coupling. On the other hand the coupling with the implicit solver results in a strong coupling. This will greatly impact the type of problems that each can tackle and thus choosing between explicit or implicit structural solvers is problem dependent. The user has to have good knowledge of the application that they are trying to simulate. A good rule of thumb is that light structures with heavy fluids (like water-rubber) should use strong coupling and thus solve the structure implicitly while light fluids coupled to heavy structures (like air-steel) work well in general with weak coupling and thus explicit mechanics should be enough.

4 Coupling to the thermal solver

The coupling of CFD with the thermal solver is normally referred to as conjugate heat transfer (CHT). In the case of the thermal solver only the implicit time integration scheme is available. The degrees of freedom are the temperatures on the fluid and structural sides of the domain (Fig. 1). There are two ways for solving CHT problems namely using weak or monolithic coupling. The weak coupling is a good approach when the solution does not require high accuracy at the fluid-solid interface, namely when the gradients are small. For higher gradients weak coupling needs a finer more resolved and possibly a boundary layer type of mesh. The advantage of using weak coupling is that it is faster to compute. On the contrary the monolithic coupling is more computationally intensive but it provides accurate results at the interfaces even in the presence of high temperature gradients.

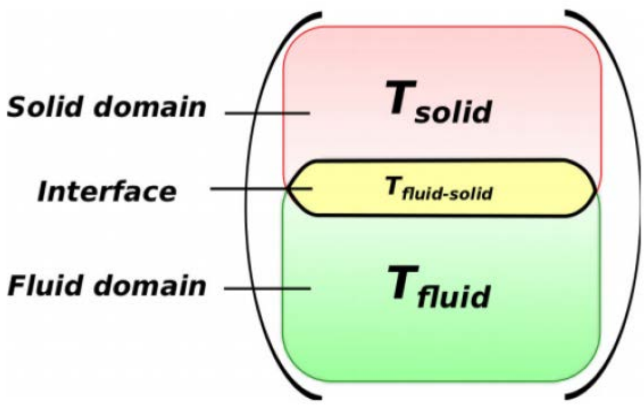

Fig.1: The fluid and thermal solvers exchange temperatures at the interface through heat flux using weak coupling or by using a monolithic approach.

5 Coupling to the DEM solver

The ICFD solver may also be coupled to the DEM spheres to approximate the motion of particles subject to the fluid drag force. The type of coupling used is weak coupling since DEM always run in explicit mode. Depending on the problem the coupling could be one-way or two-way so the particles could also provide a body force that is applied to the fluid system to evaluate the effect that particles have in the motion of the fluid.

6 Setting up a coupled problem

A main focus during the implementation of the ICFD solver has always been to keep the coupling procedures as simple as possible. This means that having a working ICFD model and a working structural/thermal/DEM model the coupling should be done with minimal effort for the user using the minimum amount of keywords or parameters that provides a robust coupling.

6.1 Fluid Structure Interaction

Solving a fluid structure interaction problem means that the user has a working structural mechanics model made of deformable or rigid body parts. The geometry that defines these parts may be discretized using shell elements, solid elements or beams. Typically the user will want to evaluate the effect of the fluid forces on the parts of the structure that are “wet”. These parts form the interface where forces/displacements are transferred and they could be made of shell elements or in the case of solid elements the external faces of the solid elements that are in touch with the fluid. So the very first most
important concept for setting up an FSI model is that the user needs to recognize these wet zones where fluid loads will be transferred to the structure. This is important because those external wet surfaces of the structure will need to have a fluid mesh which is body fitted to the structural objects. In general all parts of the structural model that are in touch with the fluid will need a fluid mesh. The second most important step is to define the mesh resolution that is needed to accurately capture the physical features of the problem. In general terms the fluid mesh will need a higher resolution or at least the same resolution as the structural mesh. In one extreme one can imagine the case of a rigid body cube which can be defined by six quadrilateral shell elements coupled to a high Reynolds number flow with strong turbulent effects. An accurate solution of this problem means that each quadrilateral face of the cube will probably need several thousand fluid elements (Fig. 2). The opposite may happen in the presence of a highly viscous fluid flowing in a slow motion coupled to a highly flexible structure. So it is important for accuracy and efficiency to have an optimal number of elements on each side of the model. It is always advisable to test the fluid and structural models independent from each other to see if they behave as expected before solving a coupled simulation. Once both simulations run fine and the interface mesh is correct then the two input decks may be coupled by telling the ICFD solver which PIDs of the fluid mesh will form the FSI interface. In other words which parts of the fluid surface mesh is touching the structure. The keyword is **`*ICFD_BOUNDARY_FSI`** followed by the PIDs of the surfaces in the next line. There is no need to modify the structural model. In addition the keyword **`*ICFD_CONTROL_FSI`** may be used to modify the default values for the coupling i.e. changing from two-way to one-way coupling.
A variation of the typical transient coupling could be to solve a steady state problem in ICFD and once the convergece has been achieved apply the fluid loads to a structure to solve a linear problem, an eigenvalue analysis, etc. If a structural transient problem is solved after the steady state ICFD analysis then the load applied to the structure will be constant in time and equal to the value at steady state.

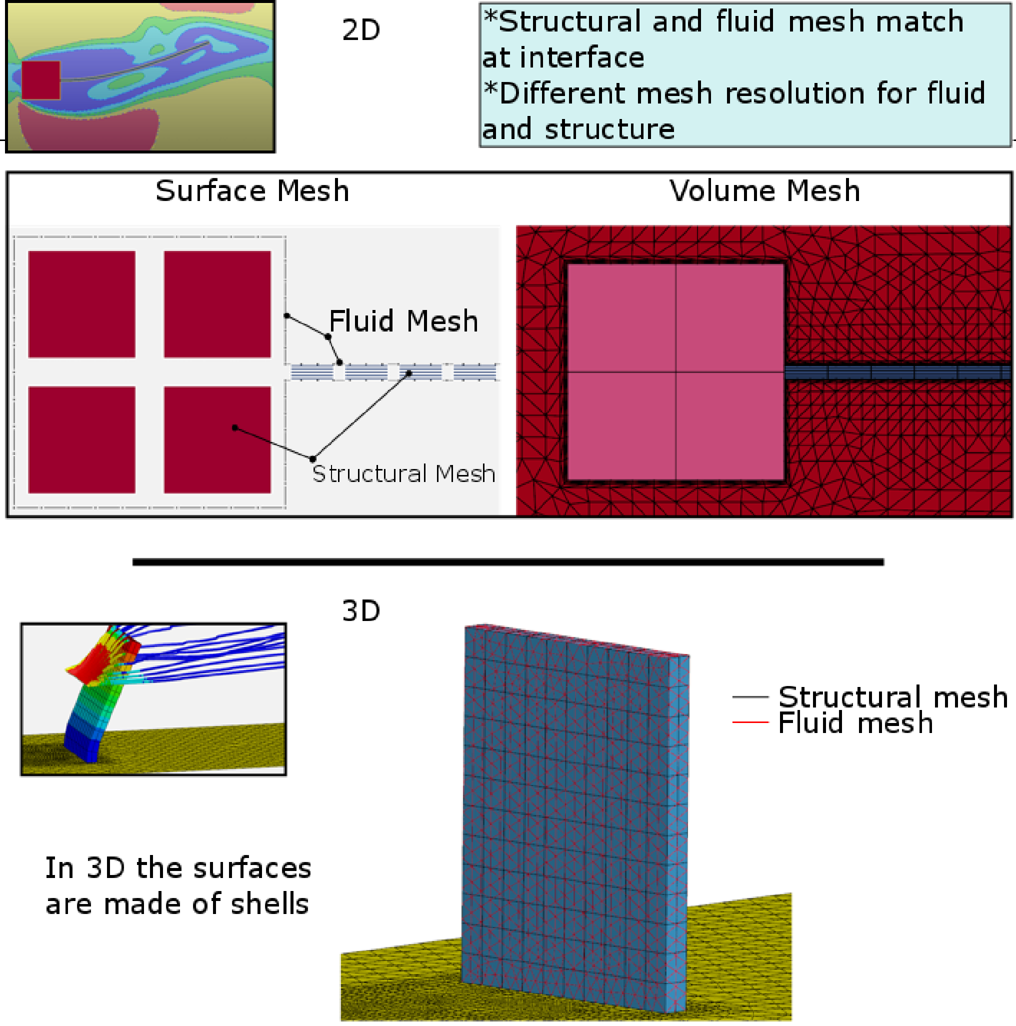

Fig.2: Sketch showing the mesh settings for a 2-D and 3-D FSI model. Note how the meshes for the fluid and structure at the interface will not match although both meshes are body tight. In 3-D the volume mesh is not shown for clarity.

6.2 Conjugate heat transfer

Many of the principles described previously for FSI apply to the conjugate heat transfer coupling. In terms of meshing the solvers need to satisfy very similar requirements. The need to be body fitted meshes and having a resolution that is fine enough to guarantee some accuracy requisites.
The conjugate heat transfer solution also needs the keyword ***ICFD_BOUNDARY_FSI** to indicate the PIDs of the surfaces that form the interface. In addition the user also needs to add the keyword ***ICFD_BOUNDARY_CONJ_HEAT** to indicate the solver which are the boundaries where the thermal solutions will be coupled. To modifiy the default values i.e. one-way or two-way coupling, etc., the keyword ***ICFD_CONTROL_CONJ** is used to switch between weak and strong coupling.
In problems where the fluid flow reachs steady state the fluid part of the problem can be turned off using the keyword ***ICFD_CONTROL_TIME** leaving only the conjugae heat transfer problem running. This can save time since the fluid solution is not being computed.

6.3 DEM

The ICFD solver can also be coupled to the discrete element method solver. In DEM the discretization is done using spheres which interact with each other when they come in contact by means of springs. The resultant force makes the particle move. Each particle has a given mass, radii and a position in space. When DEM is coupled to ICFD each sphere will be assigned a host finite element on the fluid mesh and a force will be added to the sphere. The force is computed using the drag force of a sphere: $F_D = (V_f^2 A \rho_f C_d)/2$ where $A = \pi r^2$ is the projected area of the sphere and V_f the fluid velocity. If the coupling is two-ways then a volumetric force is computed to assign it to the fluid force vector.
The only keyword needed to define the coupling is ***ICFD_CONTROL_DEM_COUPLING** which not only activates the coupling but also indicates if the coupling is one-way or two-way.

7 Summary

The ICFD solver in LS-Dyna is a tool to study the behavior of incompressible flows. The solver is coupled to other tools in LS-Dyna to study the behavior of coupled systems that arise in many engineering applications. This paper provided some insight into the different coupling techniques with examples of when each technique is more applicable. After that it was discussed the three main coupling options available with the keywords that activate and control them.