Immersed Interface Development in Incompressible CFD

Facundo Del Pin, Iñaki Çaldichoury, Rodrigo R. Paz and Chien-Jung Huang
Livermore Software Technology LLC

Abstract

The pre-processing of complex geometries exported from CAD programs is a big challenge in the Finite Element analysis of fluid problems. There are many situations where a detailed high quality mesh is preferred and possible mandatory. Such is the case for problems where shear stresses are an important component of the total force, i.e. ground vehicle aerodynamics, aircraft drag prediction and some bio-mechanics applications where the stresses in the endothelium are need to predict the development of some diseases. There are many other applications though where pressure forces are enough in terms of accuracy or where rapid prototyping of engineering parts do not need the accuracy required in the final stages of engineering design. In these cases, the geometry could be simplified by approximating the domain walls immersing them inside a much simpler domain. This simplification becomes even more appealing in the presence of an internal structure that interacts with the fluid which in many engineering applications are modeled as thin shell structures. In the current work the sub-element interfaces of the geometry will be approximated by level set distance functions. The walls could be part of a flexible structure or they could be rigid and they may be "wet" on both sides. The pressure discontinuity across the wall (in the case of shell structural elements) will be approximated by discontinuous shape functions as described in [1]. One of the main advantages of this approach is that it is easily adapted to an existing solver since no additional degrees of freedom need to be added. The presentation will include details of the additions that the existing solver needed such as: 1) boundary recognition; 2) level set representation; 3) sub-element splitting; 4) computation of the new interpolation functions and integration; 5) assembly and solution.

Introduction

The pre-processing task of complex and dirty geometry is a tedious task that can be time consuming and which diverts the focus of engineers from analysis to meshing. Advanced meshing tools have made high progress in automating the process, but it still takes many human hours to produce a high quality mesh suitable for analysis. Immersed interface techniques are aimed at transferring the heavy lifting work from a human to a computer increasing the computational time due the detection of interfaces at analysis time but reducing the human intervention to a minimum during pre-processing. This is the main appeal of the method and the main reason why it is a good fit for multi-purpose analysis tools like LS-DYNA®. In the ICFD solver there is an ongoing effort to implement these methods with the double goal of simplifying pre-processing and assisting in some FSI simulations by preventing excessive re-meshing and allowing contact, penetration and erosion (topological changes). Many immersed interfaces techniques luck accuracy when the internal interfaces are thin shells with no thickness since they are based on Heaviside functions for smoothing which need thickness of interfaces for accuracy. Since FSI problems many times involve shells in LS-DYNA then the implementation of immersed interfaces has to robustly support thin structures and for that it is imperative that the method allows sharp discontinuities of state variables across the interface maintaining the high accuracy of the finite element method present in the rest of the domain away from the interface. All this constraints and features need to be part of our implementation for it to be successful and adopted by users. It goes with no saying that it has to be robust, fast and scale in parallel simulations.
This development has been taken seriously, and in this paper, we will introduce the principles of the method implemented in LS-DYNA ICFD as well as some preliminary results. At the time when this paper was written a beta version SMP and CFD only is being released in version R12 of LS-DYNA. The idea is not to replace the body fitted technique available in the solver but to create an environment where both techniques will work together and complement each other.

There are problems, i.e. ground vehicle aerodynamics, where the geometry is complex, and meshing can be challenging in areas where the flow is detached from surfaces like engine compartments. Those parts of the model are great candidates for immersed interfaces while in the same model areas of a vehicle like the external body where the flow is attached and where a boundary layer mesh is needed for accurate drag prediction will use a body fitted technique.
In what follows the immersed interface methodology under development will be presented in the first part. In the second part a first glimpse at how the user interface may look in LS-DYNA will be introduced. For the third and last part some examples of applications will be presented.

Immersed interface method

The choice of methodology to implement immersed interface had to fit our finite element framework and parallel implementation. We decided to adapt and test a development that was already part of the code in the field of multiphase flows with surface tension. Indeed this kind of problems present pressure discontinuities which we solve using a discontinues finite element space for the pressure field. The idea was to replace the immersed interfaces by a level set function in the same way that a fluid interface is identify in multiphase flows. In Fig. 1 there is a schematic representation where a level set function is used for a free surface model like a dam break to identify the fluid interface and in a fluid structure interaction (FSI) problem to represent the structural interface.

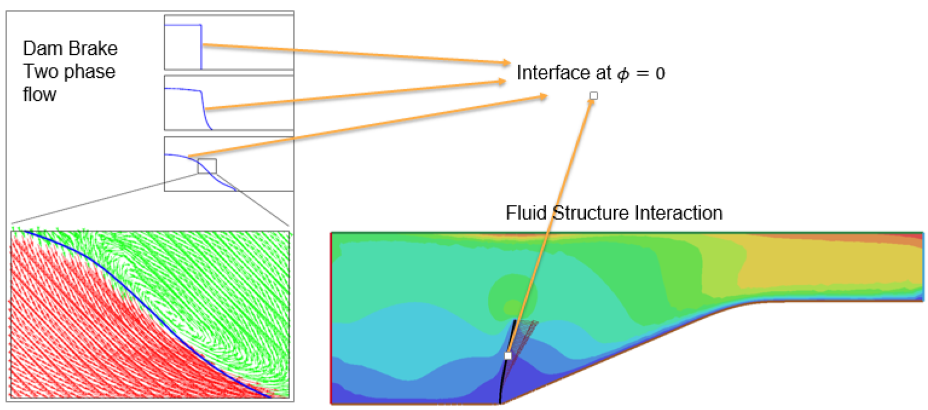

Fig. 1: for FSI problems use a level set function to represent the solid structures and use the new discontinuous pressure space at the interface elements.

The discontinuous pressure space was introduced in [1]. It has several appealing properties:
1. no additional degrees of freedom are incorporated,
2. pressure shape functions are modified so as to capture discontinuities,
3. the modifications are local and can be computed element-by-element,
4. the modified functions are piecewise linear on each side and only discontinuous at the interface,
5. they form a nodal basis, take the value one at their corresponding node and zero at the other nodes.

Property number one means that the structure of the lineal systems will not change allowing us to use exactly the same technology as for body fitted mesh. The implementation in an existing finite element code is straight forward since all we have to do is replace the approximation space in those elements that are intersected by an interface. A picture with the new discontinues functions is shown in Fig. 2. Without going into the details, the generation of these functions is very simple. To define them we simply "carry" the value at each node towards the intersection of any edge emanating from it with the interface. The elements at the interface will have to get subdivided into sub-elements to perform the integration. There are several possibilities for cutting a tetrahedral elements which are shown in Fig. 3.

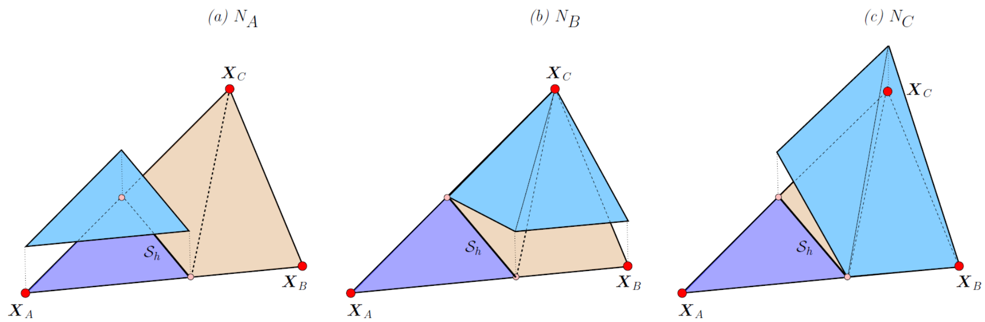

Fig. 2: new discontinues finite element space use for the pressure on elements cut by the immersed interface.

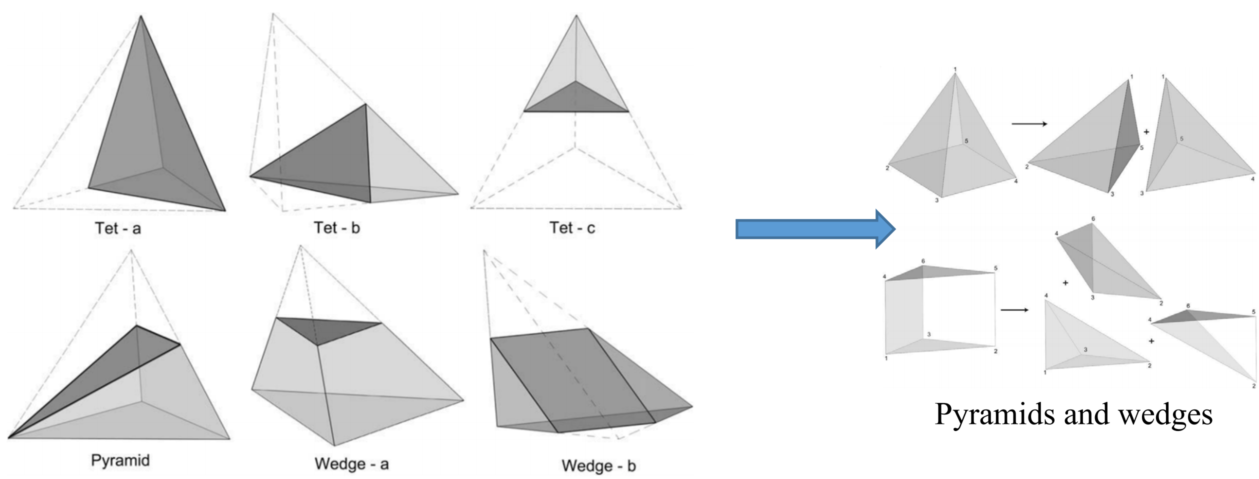

Fig. 3: element subdivision for finite element integration.

A robust and fast edge to face intersection algorithm is needed as part of the implementation. There are many data structures available to do this. We used a bucket sort since it was already implemented in the code to make the search $O(n+k)$ with n the number of edges and k the number of buckets.

Keyword interface

Since this development is being released in beta mode the keyword interface is subject to change after user feedback. In our first round of iterations we added a new subset of `*MESH` keywords. The immersed interface will be defined either by a structural element for FSI or by a `*MESH_IMM_ELEMENT/NODE`. The elements will be triangles in 3-d and line segments in 2-d. The keyword `*ICFD_CONTROL_IMMERSED` is used to tell the solver that the immersed interface solution is active and to indicate if it is a static interface which does not change in time or dynamic. Only static is supported at the time this paper was written.

Numerical tests

In this section three numerical tests will be presented to evaluate the accuracy of the method as well as how it compares to a body fitted approach.

Flow past a cylinder

This is a classical laminar example of flow at $Re = 100$ around a static cylinder with non-slip boundary conditions. A schematic of the problem is presented in Fig. 4 showing the geometry and the boundary conditions.

The same figure also shows the mesh used in the analysis. There is a coarse background mesh with larger elements used to represent the shape of the cylinder while a much finer mesh is used to compute the solution.

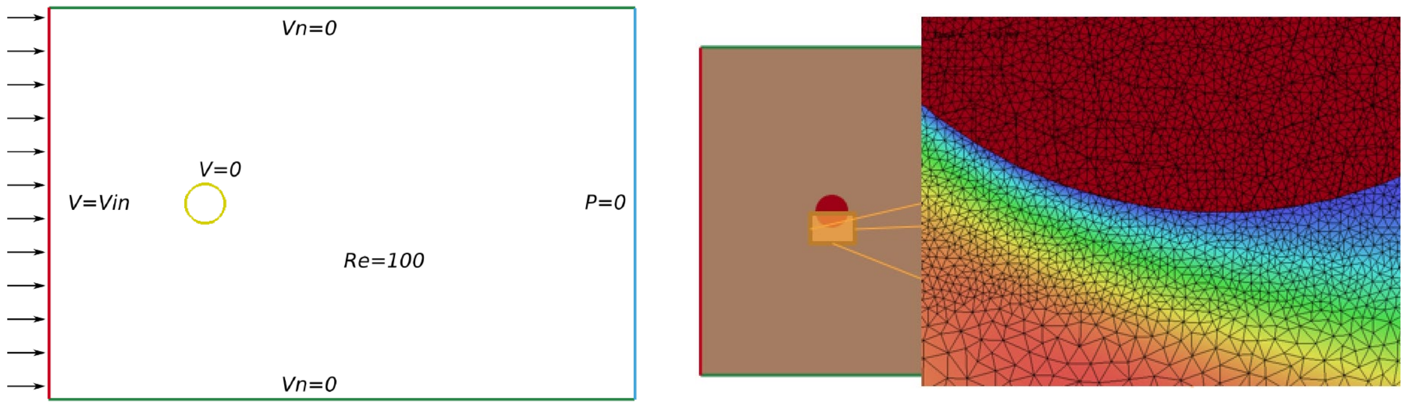

Fig. 4: geometry and boundary conditions for the flow past a circular cylinder at $Re = 100$ (left). Mesh used in the immersed interface solution (right).

The results are presented in Fig. 5 where the forces on the cylinder are compared for the immersed interface and the body fitted approach. Both results match almost exactly. In the same figure there are contours of velocity magnitude that also look in accordance with the body fitted method.

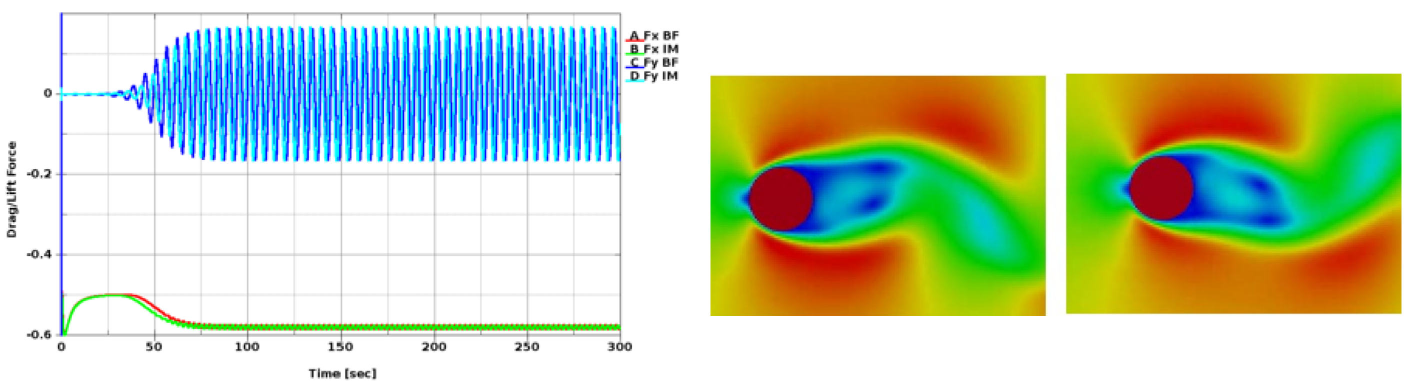

Fig. 5: Comparison of drag and lift forces for the cylinder between the immersed interface and the body fitted technique (left). Contours of velocity magnitude for the immersed interface on the right.

Flow past a sphere

This a very similar test case as the one presented above but now in three dimensions. The cylinder has been replaced by a sphere and the results are compared to experimental values for a flow at $Re = 100$. The domain is shown in Fig. 6 with the dimensions expressed as a multiple of the sphere diameter D. For the result we compare the wake length of the flow behind the sphere and the drag coefficient to those available in [3]. The results are presented in Fig. 7 and both values are in good agreement with the reference solution.

High Reynolds flow past a bluff body

In this last example the solution of the flow past a bluff body at $Re = 40000$ will be explored using immersed interface. For this case only qualitative results are compared to see if the main fluid structures are found in the immersed solution. The results for this simulation are presented in Fig. 8. We observe that the *trumped vortex*, *horseshoe vortex* and *bound recirculation* are all found in the simulation.

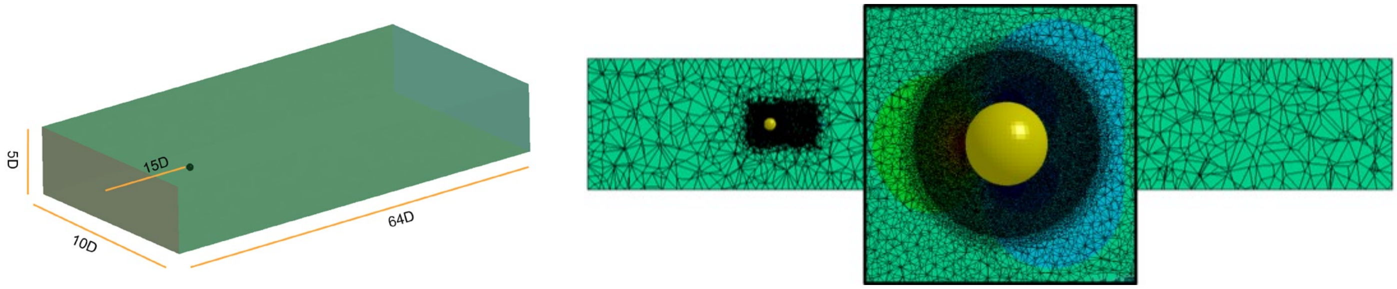

Fig. 6: fluid domain used for the immersed interface problem of a flow past a sphere (left). Fluid mesh with the detailed refinement on the right.

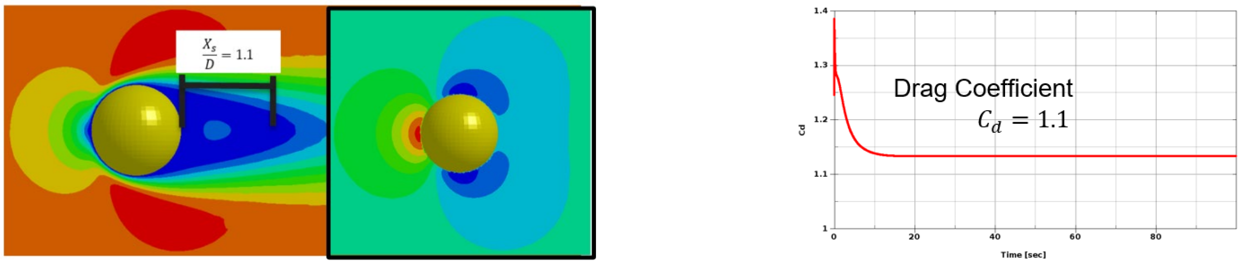

Fig. 7: results for the three dimensions flow past a sphere. On the left the velocity and pressure are shown and the non-dimensional wake size is depicted. The pressure is also shown. On the right the drag coefficient.

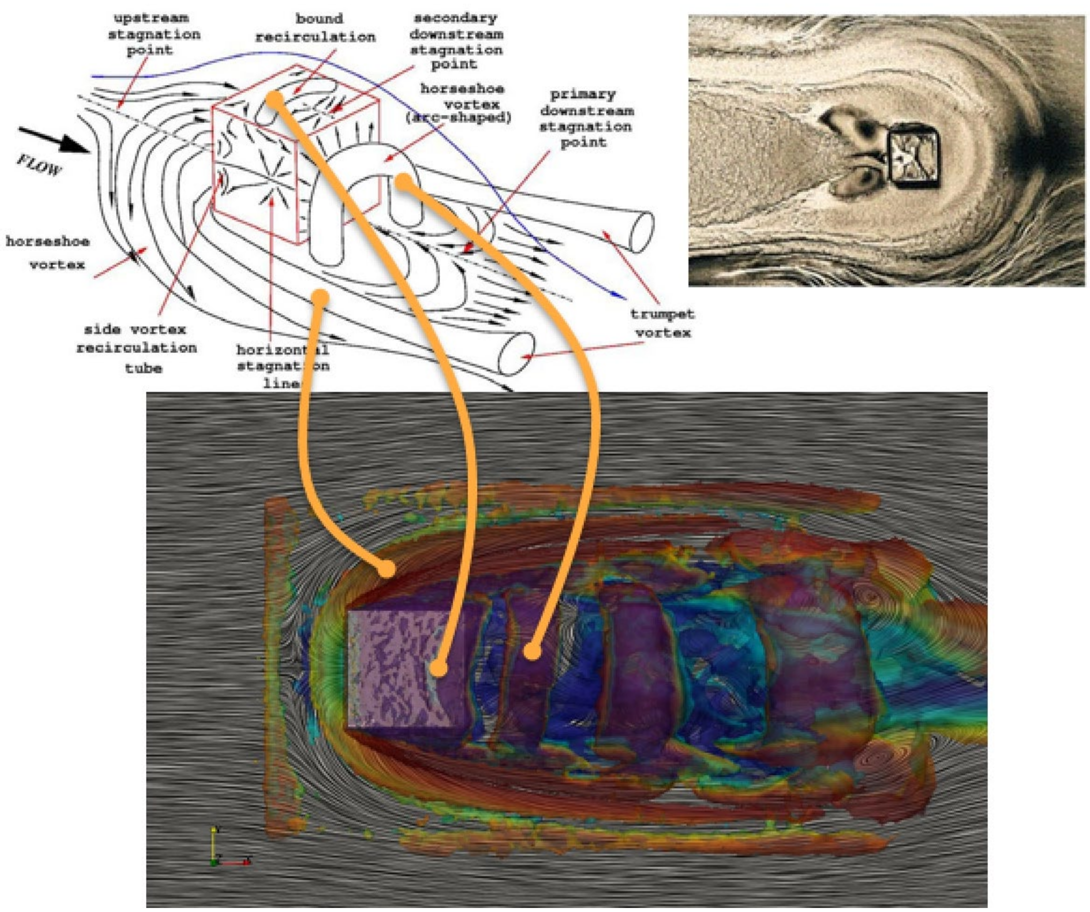

Fig. 8: results from a high Reynolds number problem for the flow past a bluff body. Identification of typical flow structures observed in experimental results.

Conclusions

In this paper a new methodology to represent wall bounded flows using immersed interfaces in the ICFD solver has been presented. The motivations behind the current development were explained and it was emphasized that the method had to perform well when the internal walls were represented by thin shell structures. It was also mentioned that the goal is to have a hybrid body fitted / immersed interface approach to take advantage of the benefits that both methods bring to the analysis of complex flows. The numerically methodology was briefly discussed where a discontinuous finite element space for the pressure was introduced. Finally, three test cases were presented showing the performance of the method at low and high Reynolds number in problems with static walls.

References

[1] R.F. Ausas, F.S. Sousa, G.C. Buscaglia, An improved finite element space for discontinuous pressures, Comput. Methods Appl. Mech. Engrg. (2009), doi: 10.1016/j.cma.2009.11.011

[2] Ramsharan Rangarajan, Adrián Lew, Gustavo C. Buscaglia, A discontinuous-Galerkin-based immersed boundary method with non-homogeneous boundary conditions and its application to elasticity,Computer Methods in Applied Mechanics and Engineering, Volume 198, Issues 17–20, 2009, Pages 1513-1534, ISSN 0045-7825, https://doi.org/10.1016/j.cma.2009.01.018.

[3] Campregher, Rubens, Militzer, Julio, Mansur, Sérgio Said, & Silveira Neto, Aristeu da. (2009). Computations of the flow past a still sphere at moderate reynolds numbers using an immersed boundary method. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, *31*(4), 344-352.