New Development of the Gap Closure Feature in LS-DYNA ICFD

Chien-Jung (Peggy) Huang, Facundo Del Pin, Iñaki Çaldichoury, Rodrigo R. Paz

Livermore Software Technology LLC, an ANSYS company

1 Abstract

There is a great interest in the fluid-structure interactions (FSI) of flow mechanics around valves in the heart or in mechanical parts. The capability to allow flow blockage due to the valve closure is very important. A new feature for managing the contact of immersed structures in the LS-DYNA ICFD solver is presented and tested. In the framework of conformal mesh used in ICFD solver, when two structures come into contact, the possible overlap or intersection of surface meshes at the contact location would cause the simulation to fail. As a solution, a clearance between two contacting surfaces is necessary. In the gap detection process, if the proximity between two surfaces is below a threshold value, the two surfaces are considered in contact in ICFD solver. To eliminate the resulting artificial flow leakage, the flow control treatment is performed on the elements in the gap at the contact location so that the flow is obstructed when contact occurs. The FSI simulations of 2D pump and the 3D pipe are presented to demonstrate the cability of this new feature.

2 Introduction

In the FSI problems, the contact between two or more structures with fluid between them attracted a lot of attention in acdemia and industry. From the aspect of the fluid domain, when the immersed structures are in contact, it means a topological change to the simulation domain. Different types of fluid domain discretization deal with the topological change differently. The immersed boundary (IB) method with Eulerian formulation, which the Cartesian mesh remain fixed, is widely used in the FSI computation especially in the field of heart valve simulations [1]. However, the less accurate description of the fluid and structure surface limits the accuracy using IB method [2]. On the other hand, for the Arbitrary Lagrangian-Eulerian (ALE) formulation where the mesh is conformal to the boundaries, contact or penetration of structures can make elements collapse or invert. Few literatures have explored the FSI contact problem with ALE formulation. Spühler et. al. [3] proposed a unified continuum fluid-structure interaction model. At the location of collision, the fluid meshes between structures undergo local phase change from fluid to solid. Takizawa et. al. [4] excluded the collapsed elements at the contact location from the integration of the finite element formulation. They have successfully applied this technique on the heart valve flow calculation [5].

In this paper, a new technique called gap closure is developed to deal with the contact of multiple structures in FSI problems with LS-DYNA ICFD solver. The LS-DYNA ICFD solver uses finite element method with ALE formulation. To prevent overlap or intersection of surface meshes at the contact location, a gap needs to remain between two surfaces that are in contact. If the clearance between two surfaces is below a user-defined threshold value, these two surface are considered to be in contact. A gap detection algorithm is implemented to detect the occurance of contact and identify which elements are between the contacting surfaces. A flow control treatment inspired by Takizawa et. al. [4][5] is applied on these elements to block the flow.

In the next section, the new keyword and the gap closure treatment are introduced. The simulation results of two test cases of 2D piston pump and 3D collapsed pipe are shown in the Section 4. The discussion and summary are presented in Section 5 and 6, where progress and current limitations on this new feature will be discussed.

3 The ICFD solver and the new gap closure feature

3.1 Introduction

LS-DYNA ICFD is a numerical solver for incompressible flow [6] based on finite element formulation (FEM). It can be coupled with the structural solver to simulate FSI problems. The coupling can be done in one-way or two-way, where physical quantities on the interface are transferred in one direction or both directions between two solvers. On the FSI interface, the fluid mesh is Lagrangian, which means the mesh on interface moves along with the structures. This implies re-meshing is needed in some circumstances when the structure undergoes large movement or deformation.

To enable analysis of contact in FSI problem combining LS-DYNA structural and ICFD solvers, first they are connected through keywords **`*ICFD_BOUNDARY_FSI`** and **`*ICFD_CONTROL_FSI`**. The former keyword defines the FSI interfaces, whereas the latter keyword specifies the type of coupling. In order to prevent overlap or intersection of surface meshes causing volume mesh to invert or collapse, there should remain a gap between the two contacting structures. In the ICFD solver, a threshold h_{gap} is set so that two structures are considered in collision if their distance is below h_{gap}. For the structure, the contact between two objects is managed by the **`*CONTACT`** keyword. The contact thickness in the **`*CONTACT`** keyword can be set to maintain a gap between colliding structures. The contact thickness and h_{gap} don't need to have the same value. They can be two similar values carefully chosen according to the problem.

3.2 Gap closure in ICFD

The gap closure feature is activated using the keyword **`*ICFD_CONTROL_GAP`**. The value of h_{gap} is specified in the first card, and all the PIDs that are involved in the gap closure treatment are listed in the second card. Once the gap closure is activated, the ICFD solver will first go through gap detection process. The distance from these involved surfaces to their nearby nodes are calculated. For a particular node, if its distance to surface A and B are less than h_{gap}, which means

$$d_A + d_B \leq h_{gap}, \quad (1)$$

this node is marked as a gap node. Then gap elements are identified according to the gap nodes. The gap elements are excluded in the continuity and momentum equations, and the flow velocities on the gap nodes are set to be zero to achieve flow blockage at the contact location. Since the pressure field will not be solved on some gap nodes, when two surfaces get into contact and separate later, there are unrealistic pressure values on these nodes. To overcome this problem, after solving for the flow field, the pressure field is reconstructed on these gap nodes using the pressure value of their closest non-gap nodes. After the gap closure treatment, the pressure field on two sides of the contact location have two distinct pressure values.

4 Numerical Results

Two verification cases are shown to demonstrate the gap closure feature, including a two-dimension piston pump problem and a three-dimensional pipe compressed by two rigid cylinders.

4.1 2D piston pump with two check valves

The pump has a rectangular chamber with a moving piston at the bottom as shown in Fig 1. There are two openings at the top, with two check valves fixed at their left end and free at the right end. The left valve is only allowed to open outward, while the right valve is only allowed to open inward.

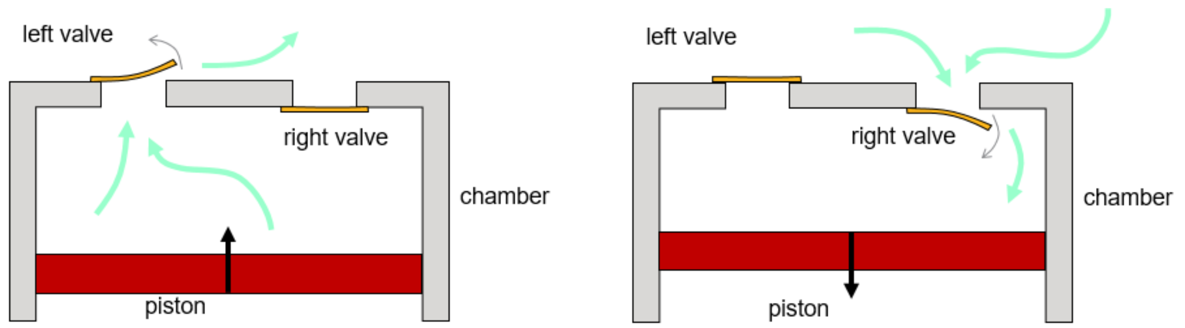

Fig.1: Illustration of the 2D piston pump problem.

The chamber is 15 mm in height and 30 mm in width, and the piston and chamber are rigid bodies. The piston moves up and down in a sinusoidal motion (see left figure in Fig 5). The valve is 7 mm long and

0.078 mm thick and is considered an elastic material with density 7.8×10^3 kg/m^3, Young's modulus 100 GPa and Poisson ratio 0.3. The fluid has the same properties with air.

The Fig 2 shows the simulation domain. The zero pressure boundary condition is set at far field of the fluid domain. For the two valves, their left sides are mounted on the rigid wall, and their free sides are 0.15 mm away from the wall. The contact thickness in structural solver is set to be 0.13 mm, whereas the h_{gap} is chosen to be 0.138 mm in ICFD solver. The simulation time is T=0~0.02 s and the time step size is Δt=10^{-5} s.

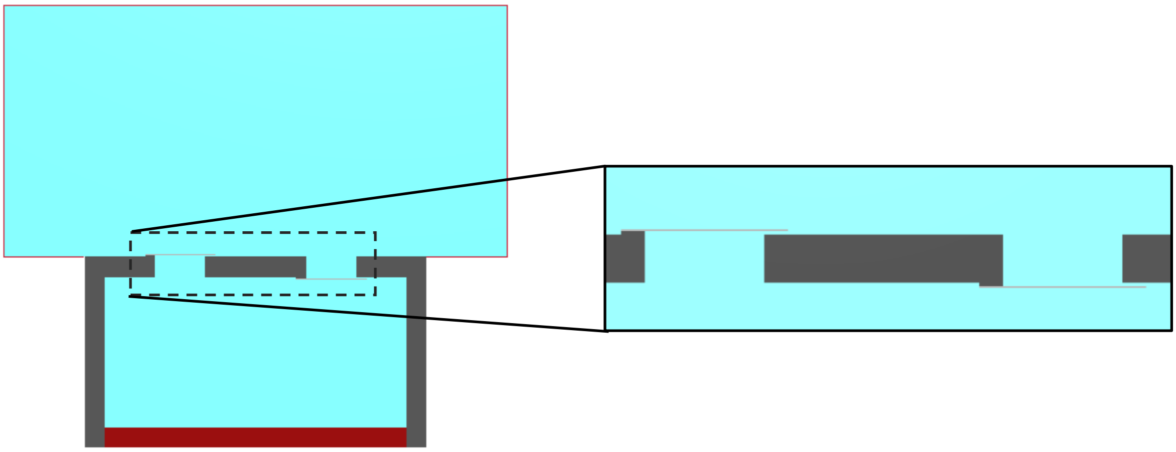

Fig.2: Simulation domain and a closer view near the two valves.

The computed flow fields with and without the gap closure treatment are shown in the Fig 3 and 4 at T=0.003. At this time, the piston is moving upward, and it is expected that the piston movement raises the fluid pressure in the chamber. The left valve is expected to open, and the right valve should be in contact with the chamber and remain closed. It can be observed that without the gap closure treatment, flow leakage occurs at gap of the right tip of the right valve. With the gap closure technique, the contact between valve and wall is detected in ICFD and no flow goes through the contact location. This leads to higher pressure in the chamber and drastically changes the motion of the valves. In right plot of Fig 5, the obtained displacements of the valves with gap closure treatment are larger compared with the ones without the treatment.

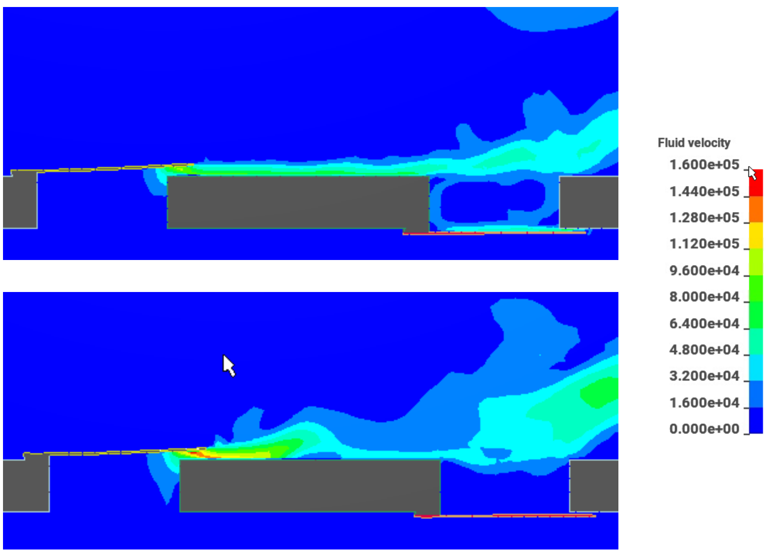

Fig.3: The velocity fields around the valves at T=0.003(s) without (top) and with (bottom) gap closure treatment. (The unit of the velocity in the legend is mm/s)

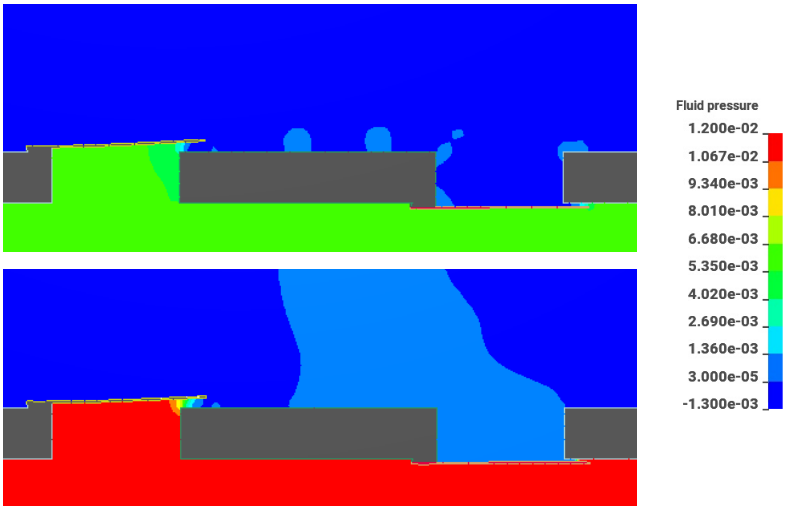

Fig.4: The pressure fields around the valves at t=0.003 without (top) and with (bottom) gap closure treatment. (The unit of pressure in the legend is MPa)

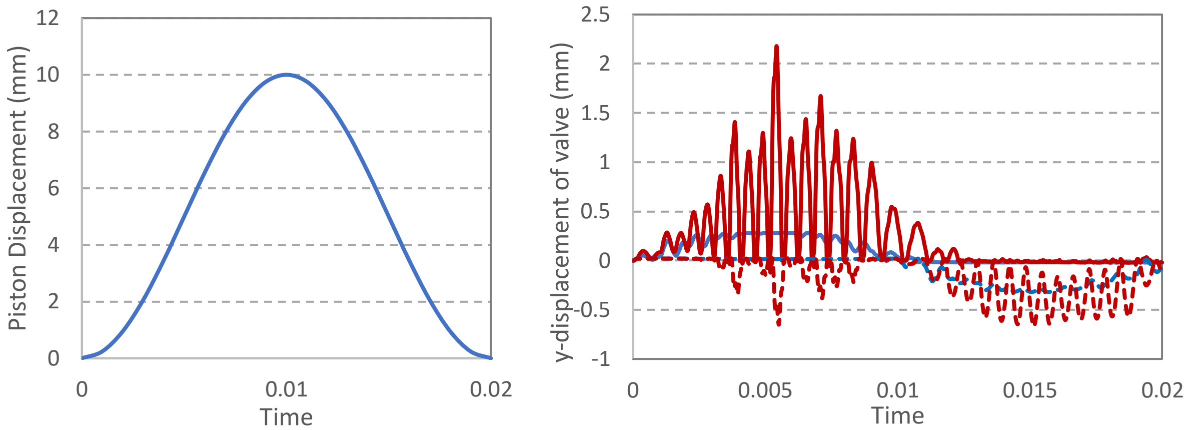

Fig.5: Left: piston displacement over time. Right: the y-displacement at the free tip of left (solid) and right (dotted) valves with (red) and without (blue) activation of gap closure treatment.

4.2 3D pipe collapse

A flow in a 3D collapsing pipe is simulated to examine the flow blockage ability after the gap closure treatment. The circular pipe with diameter 20 mm and length 500 mm is compressed by two rigid circular cylinders. The pipe is set to be elasto-plastic material with same density and Young's modulus as steel. The two rigid cylinders move toward each other in a constant speed between time T=0~1.5 s. They stop at T=1.5 and remain stationary after that. The pipe is considered to be collapsed after T=1.5 s. There is water driven by pressure difference (=1Pa) in the pipe.

The FSI simulation is set to be one-way coupling, only the solid solver transfers the movement of the structure to the ICFD. The simulation ends at T=3 with time step size 0.01. As to the mesh, the mesh is refined around the contact location, as shown in Fig 6. The minimum height at the contact location is 1.5 mm and the minimum mesh size there is 0.5 mm. The h_{gap} is chosen to be 2.2 mm.

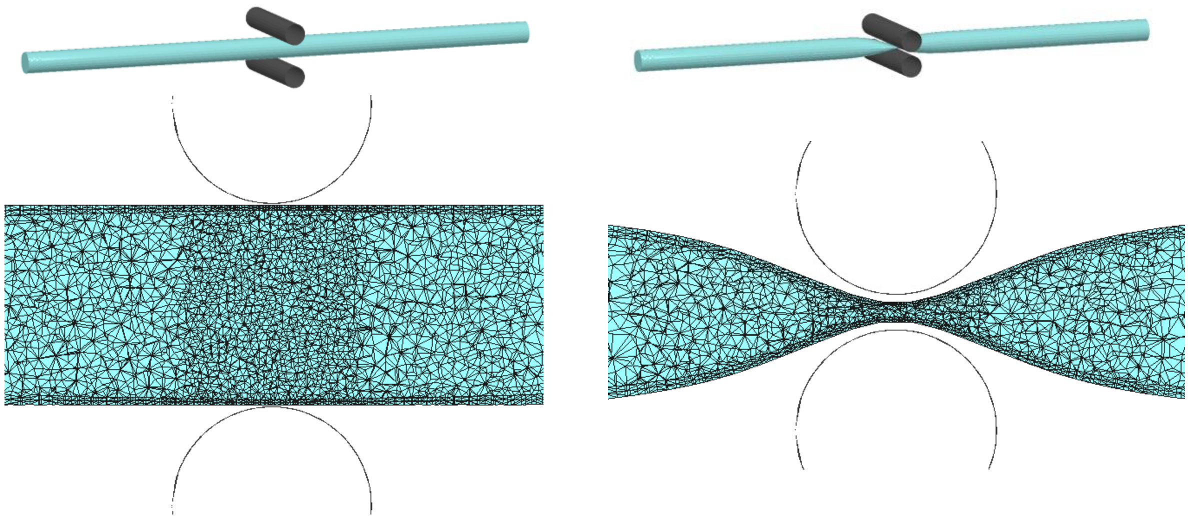

Fig.6: Initial (left) and final (right) configuration and mesh at the contact location.

With the gap closure treatment, after T=1.5, the gap elements are identified at the collapse location as shown in the Fig. 7. The flow velocity drops to zero at the collapse location. There are circulation flows remain in both sides of the collapsed pipe. There are two distinct pressure values across the collapse location corresponding to the pipe inlet and outlet pressure (Fig. 8). Fig. 9 shows the differences in the volume flow rate at the pipe inlet after collapse with and without the activation of gap closure treatment. Before the collapse, the flow is squeezed out of the pipe leading to negative fluxes. After the collapse, if without gap closure, flow still leaks from the gap due to pressure gradient resulting a positive flux. The flux after collapse is smaller with the gap closure treatment.

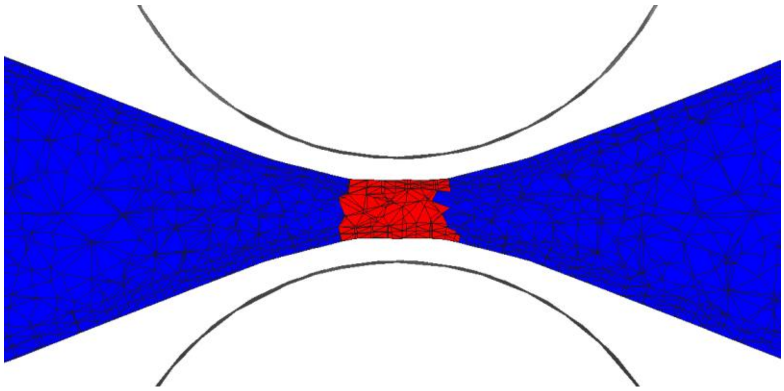

Fig.7: The gap elements (red) identified at the pipe collapse location according to the selected h_{gap}.

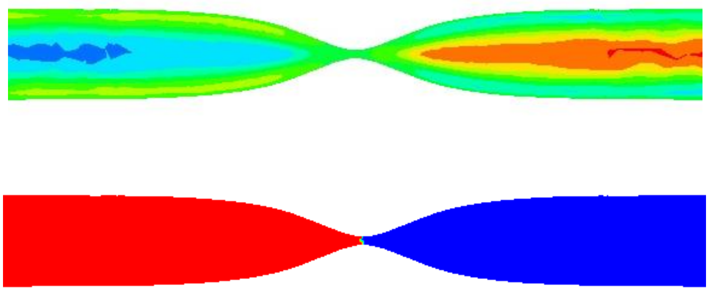

Fig.8: The flow velocity along the pipe direction (top) and the pressure field (bottom) at T=3. (The velocity contour ranges from -3 to 3 mm/s, and pressure contour ranges from 0 to 1 Pa.)

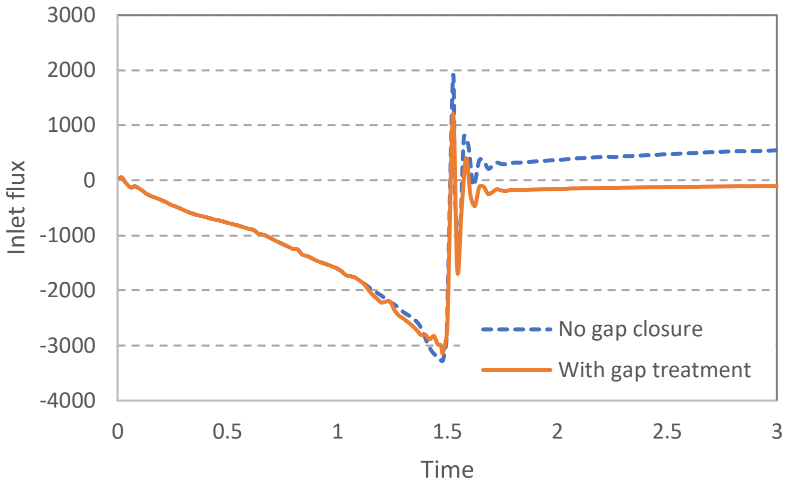

Fig.9: The flux (mm^3/s) at pipe inlet with and without gap closure treatment.

5 Discussions

The two test problems demonstrate that the proposed gap closure feature in ICFD is capable of detecting the occurrence of the contact of surfaces and applying flow control on the identified gap elements. However, there are still some limitations of the current gap closure feature. For the gap detection process, it cannot detect the occurrence of the single surface contact since it uses the criteria of Eq.(1). If contact within a single surface is expected, two PIDs need to be set on this surface in order to obtain the correct gap detection outcome. Also, the element near the corner of two surfaces may be falsely identified as a gap element. Moreover, as to the problem of pipe collapse, it can be observed that after the pipe collapse, the final flow rate is not exactly or close to zero. This may be because of several reasons. Firstly, since this problem is associated with sudden change of the flow field, the stabilization schemes may introduce compressibility to the flow. Secondly, the flow control technique can be further improved. Lastly, some elements around the intersection of top and bottom surfaces are falsely identified as gap elements and may affect the result of obtained flux.

6 Summary

In this paper, the new feature of gap closure in ICFD has been introduced to handle the contact in FSI problems. To avoid intersection or overlap of surface meshes at contact location, a gap should remain between surfaces in contact. This can be achived by setting the contact thickness in the structural solver. For the ICFD part, the gap closure is activated with keyword `*ICFD_CONTROL_GAP`. The gap threshold value and all the involved surface PIDs are specified in this keyword. A 2D piston pump and a 3D collapsing pipe have been simulated. With the gap closure activated in ICFD, the occurance of surface contact has been detected and the gap elements has been identified. The flow control was applied on the gap elements so that there is no flow leakage through the contact location, and the pressure field in the gap elements were reconstructed. The obtained flow fields showed the gap closure treatment can tackle with FSI contact problem correctly compared with the one without the treatment.

7 Literatures

[1] Borazjani, I., "Fluid–structure interaction, immersed boundary-finite element method simulations of bio-prosthetic heart valves." *Computer Methods in Applied Mechanics and Engineering*, *257*, 2013, 103-116.

[2] Bavo, Alessandra M., et al. "Fluid-structure interaction simulation of prosthetic aortic valves: comparison between immersed boundary and arbitrary Lagrangian-Eulerian techniques for the mesh representation." *PloS one* 11.4 , 2016, e0154517.

[3] Hiromi Spühler, J., & Hoffman, J. "An interface-tracking unified continuum model for fluid-structure interaction with topology change and full-friction contact with application to aortic valves. " *International Journal for Numerical Methods in Engineering*, 2020

[4] Takizawa, K., Tezduyar, T. E., Buscher, A., & Asada, S. "Space–time interface-tracking with topology change (ST-TC). " *Computational Mechanics*, *54*(4), 2014, 955-971.
[5] Takizawa K., Tezduyar T.E., Terahara T., Sasaki T. "Heart Valve Flow Computation with the Space–Time Slip Interface Topology Change (ST-SI-TC) Method and Isogeometric Analysis (IGA). *Biomedical Technology. Lecture Notes in Applied and Computational Mechanics*, 84, 2018, 77-99
[6] LS-DYNA ICFD theory manual, https://www.lstc.com/applications/icfd/documentation