FSI Simulations with LS-DYNA ICFD Solver: Capabilities, and Best Practices

Satish Kumar Meenakshisundaram[1], Facundo Del Pin[2]

[1]Ansys, Application Engineer
[2]Ansys, R&D Engineer

1 Abstract

This document presents an overview of the LS-DYNA ICFD (Incompressible Computational Fluid Dynamics) solver and best practices for its efficient use in solving complex Fluid-Structure Interaction (FSI) problems from a user's perspective. LS-DYNA ICFD solver is a powerful tool for simulating the interaction between fluids and solid structures, allowing for accurate predictions of real-world phenomena. This paper will cover the underlying principles of the ICFD solver, its unique features, and practical applications, along with tips for efficient use. The content presented is crafted to ensure that engineers, regardless of their level of experience with FSI simulations, can easily understand and benefit from it. This document aims to provide a clear and concise guide for researchers and engineers who seek to utilize LS-DYNA ICFD solver for their FSI simulations, enabling them to achieve the expected throughput and save time in the process.

2 Introduction

LS-DYNA, a widely used and powerful finite element analysis software, is renowned for its versatility in simulating complex and dynamic engineering problems. While it is primarily recognized for its capabilities in modeling structural mechanics and transient dynamics, it also offers a robust incompressible flow solver. This incompressible flow solver enables engineers and researchers to study fluid flow problems where the assumption of constant density holds, such as most liquids and gases at low Mach numbers.

The incompressible flow solver in LS-DYNA employs finite element method to accurately discretize, predict and visualize fluid behavior. By solving the Navier-Stokes equations, which govern the motion of incompressible fluids, LS-DYNA can simulate a wide range of fluid flow scenarios, including steady-state and transient flows, laminar and turbulent flows, as well as free-surface and multiphase interactions.

LS-DYNA's incompressible flow solver offers a comprehensive set of capabilities that make it an effective tool for solving complex Fluid-Structure Interaction problems. Its ability to handle strong coupling, arbitrary mesh interfaces, and a wide range of FSI scenarios allows engineers and researchers to study intricate interactions between fluid flow and deformable structures, leading to deeper insights and improved designs in various engineering disciplines. In addition, the structural solver's capability to handle highly non-linear problems provides a significant advantage for simulating challenging FSI scenarios. By accurately representing material, geometric, and contact non-linearity, it empowers users to gain deeper insights, make informed design decisions, and optimize engineering solutions in various disciplines, ranging from automotive and aerospace to civil and biomedical engineering.

This introduction serves as a gateway to explore the capabilities of the incompressible flow solver in LS-DYNA. We will delve into its underlying mathematical principles, the discretization techniques utilized, and the array of boundary conditions available for modeling real-world fluid flow phenomena. Additionally, we will discuss the challenges and limitations associated with incompressible flow simulations, as well as best practices to ensure accurate and reliable results.

Whether you are an experienced LS-DYNA user seeking to extend your simulations to fluid dynamics or a newcomer eager to explore the world of incompressible flow analysis, this exploration of LS-DYNA's incompressible flow solver will undoubtedly provide valuable insights and pave the way for innovative and insightful fluid-structure interaction simulations.

3 Implementation of ICFD in LS-DYNA

3.1 Incompressibility Condition in Fluids

Incompressibility is an essential property of certain fluids, particularly liquids and gases, that refers to the inability of these substances to change their volume significantly under the influence of external forces or pressure. When a fluid is said to be incompressible, it means that its density remains constant, and any change in pressure only results in a change in its velocity and not its volume.

Mathematically, we express the incompressibility condition using the continuity equation:

$$\nabla \cdot \mathrm{v} = 0$$

where, $\nabla \cdot \mathrm{v}$ represents the divergence of the fluid velocity vector v.

This equation states that the divergence of the fluid velocity field is zero. In simpler terms, it means that the fluid flow behaves like an incompressible substance, and any fluid elements moving together in a region will remain together.

3.2 Navier-Stokes Equations for Incompressible Fluids

The Navier-Stokes equations are fundamental equations in fluid mechanics, describing the motion of a fluid in response to external forces. For incompressible fluids, these equations will be described in the following sections.

3.2.1 Conservation of Momentum:

$$\rho\left(\frac{\partial v}{\partial t} + v.\nabla v\right) = -\nabla P + \mu\nabla^2 v + \rho g$$

where,
ρ is the density of the incompressible fluid,
v is the fluid velocity vector,
t is time,
P is the pressure,
μ is the dynamic viscosity,
∇ is the gradient operator,
∇^2 is the Laplacian operator,
g is the acceleration due to gravity.

This equation states that the change in momentum of a fluid element over time is equal to the sum of the pressure gradient, viscous forces, and the force due to gravity.

3.2.2 Conservation of Mass

$$\nabla \cdot \mathrm{v} = 0$$

This equation is the continuity equation we discussed earlier, expressing the incompressibility condition.

3.2.3 Associated Boundary Conditions

To fully solve the Navier-Stokes equations for a specific fluid flow problem, we need to apply boundary conditions, which describe the behaviour of the fluid at the boundaries of the flow domain. For incompressible fluids, common boundary conditions include:

a) No-Slip Boundary Condition

At a solid boundary (e.g., a solid wall), the fluid velocity is equal to the wall velocity. This condition implies that the fluid "sticks" to the wall, and there is no relative motion between the fluid and the boundary surface.

v = *v_wall* at the solid boundary

b) Free-Slip Boundary Condition:

At certain boundaries, such as the interface between two immiscible fluids or symmetry planes, the tangential velocity component is set to zero or to the wall velocity in the case of moving boundaries. It means that the fluid can flow freely along the boundary, but there is no relative motion perpendicular to it.

$$v \cdot n = v\_wall \cdot n \text{ at the free-slip boundary}$$

where n is the normal vector to the boundary.

c) Inflow/Outflow Boundary Condition:

At the boundaries where fluid enters (inflow) or exits (outflow) the flow domain, the velocity is prescribed based on the specific flow problem or experimental data.

$$v = v\_inflow \text{ or } v = v\_outflow \text{ at the inflow/outflow boundary}$$

These boundary conditions, along with the Navier-Stokes equations and the continuity equation, form a set of equations that can be solved numerically or analytically to determine the fluid flow behaviour in various practical scenarios.

3.3 Fluid Structure Interaction

3.3.1 Fluid-Structure Interaction (FSI)

Fluid-Structure Interaction (FSI) is a fascinating phenomenon that occurs when a fluid and a solid structure interact with each other, resulting in mutual influence and deformation. In various real-world scenarios, such as a flag fluttering in the wind, a fish swimming through water, or the behavior of blood flow in blood vessels, FSI plays a crucial role in shaping the dynamics and behavior of both the fluid and the solid.

In FSI, the fluid's motion affects the deformation of the solid structure, while the solid's movement, in turn, alters the flow characteristics of the fluid. This two-way coupling introduces complex challenges in modeling and simulation, requiring sophisticated algorithms to accurately capture the interactions between the fluid and the structure.

3.3.2 General Fluid-Structure Interaction Schema

The fluid-structure interaction algorithm is a numerical approach used to solve the coupled equations governing fluid and solid dynamics in FSI problems. It involves solving the Navier-Stokes equations for the fluid and the equations of motion for the solid simultaneously. The algorithm typically follows these steps:

Step 1: Discretization of the Domain
The computational domain is divided into a mesh for both the fluid and the solid. This mesh discretization allows us to represent the continuous equations as discrete equations that can be solved numerically.

Step 2: Time Integration
The algorithm uses time-stepping methods to progress the simulation in small time increments. The choice of time integration scheme can significantly impact the stability and accuracy of the FSI simulation.

Step 3: Fluid Solver
The fluid equations, such as the Navier-Stokes equations, are solved using appropriate numerical methods, such as Finite Difference, Finite Element, or Finite Volume methods.

Step 4: Solid Solver
The equations governing the motion and deformation of the solid structure are solved using techniques like Finite Element Analysis or Finite Difference methods.

Step 5: Interface Coupling
To couple the fluid and solid domains, the forces and displacements at the fluid-structure interface are exchanged between the fluid and solid solvers. This exchange of information ensures that the interactions between the two domains are properly accounted for.

Step 6: Iteration
The fluid and solid solvers are iteratively solved until convergence is achieved. This ensures that the mutual interactions between the fluid and structure are accurately captured.

Step 7: Post-Processing
Once the simulation is complete, post-processing is done to analyze and visualize the results, which may include fluid flow patterns, solid deformations, and other relevant quantities.

3.3.3 Fluid Structure Interaction in LS-DYNA ICFD

LS-DYNA ICFD uses a partitioned or staggered method to solve for fluid-structure interaction. A partitioned approach is where the fluid and solid equations are decoupled and solved. This offers efficieny by enabling us to solve small and better conditioned subsystems instead of a single (coupled) problem.

LSDYNA's solution schema can be classified as: loosely (or weakly) coupled and strongly coupled. Loosely coupled scheme requires only one solution from each solver (fluid and solid) per time step in a sequentially staggered manner which allows solving the structural equations using the explicit structural solver and fluid equations using the implicit ICFD solver.

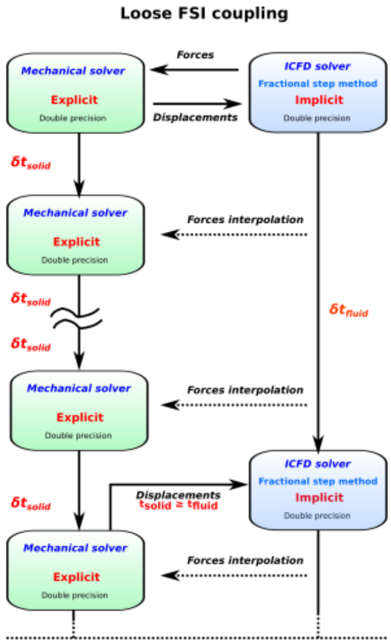

Fig.1: Weak FSI Coupling.

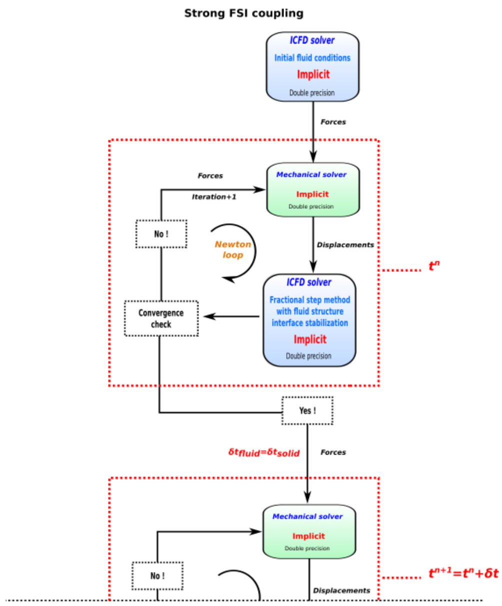

Fig.2: Strong FSI Coupling.

A strongly coupled scheme on the otherhand requires the need for convergence at the fluid-solid interface. This is to establish force balance throughout the solution process to overcome numerical instabilities. Hence the structural equations are solved using an implicit structural solver and fluid equations are solved using the implicit ICFD solver.

3.3.4 Level Set for Multiphase Flow FSI

In certain scenarios, computational fluid dynamics may involve simulating the interactions between two different fluids. These multiphase flows could include situations such as air and water interactions, oil and gas interfaces, or even more complex interactions. To accurately represent and analyze these dynamic fluid-structure interactions, the Level Set method offers a robust and versatile approach.

In FSI simulations involving multiple fluid phases, such as air and water or oil and gas, the Level Set method provides a convenient way to represent and track the fluid interfaces. The Level Set method utilizes a scalar field, called the Level Set function, to represent the position of the interface between the two fluid phases. The Level Set function is defined as a signed distance function from the interface, with negative values representing one fluid phase and positive values representing the other. The interface itself is located where the Level Set function equals zero.

The key advantage of the Level Set method in FSI simulations is that it can handle complex topological changes of the fluid interface, such as merging, splitting, and the formation of new interfaces. This makes it particularly useful for simulating fluid-structure interactions involving free surfaces or moving boundaries.

4 Key Capabilities

4.1 Volume Mesher

The ICFD solver makes fluid simulations easier by automatically creating the 3D space where fluids flow. It needs a good starting mesh on the surfaces to work well. For cases with large wall deformation in simulations involving solid-fluid interaction (FSI), the solver can adjust the mesh automatically to keep it accurate.

Here's how it works:

1. Starting with Surfaces: The solver looks at the surface mesh the user provides. The surface meshes can't overlap or have gaps; it should be watertight. The edges where two surfaces meet should match up perfectly (i.e., the boundary nodes need to be merged and share topology).

2. Building the First Mesh: Using the surface mesh, the solver connects the nodes to make a rough 3D mesh. It uses triangles or tetrahedrons that follow certain rules to fit together following a Delaunay technique.

3. Adding More Detail: The solver then adds more nodes to the volume mesh. Whenever it adds a node, it breaks the nearby triangles or tetrahedrons into smaller ones. This helps make the mesh more detailed and accurate. It does this again and again until the right mesh is achieved.

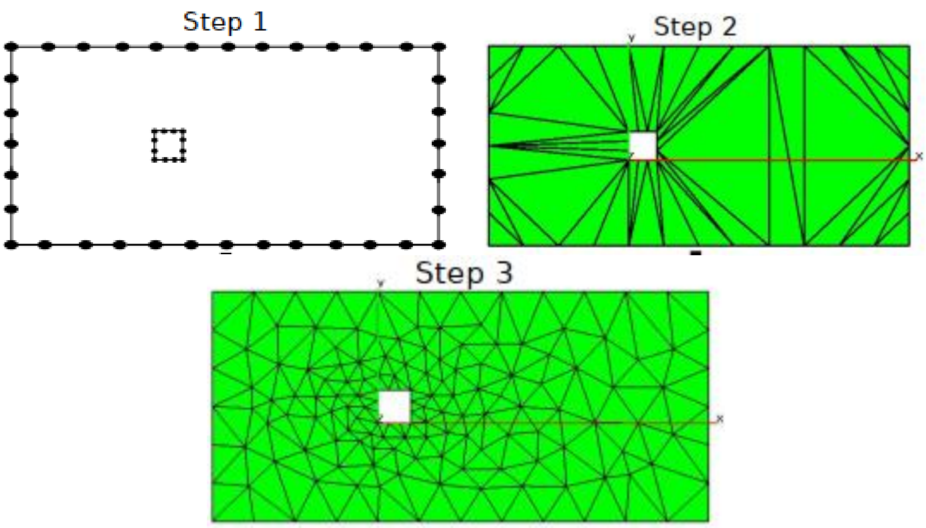

Fig.3: Volume meshing methodology.

4.2 Local Mesh Refinement

We utilize the *MESH_SIZE_SHAPE card to enable localized control over element sizing within distinct geometric zones, including boxes, spheres, and cylinders. This innovative feature empowers users to specify specific element sizes, which serve as reference dimensions during the iterative process of node addition and final volume mesh construction (Step three in the aforementioned methodology).

1. Customized Element Sizing: By selecting regions of interest, represented by geometric shapes, users can prescribe unique element sizes. This feature essentially guides the meshing algorithm to prioritize accuracy and refinement in these designated zones.

2. Enhanced Mesh Quality: As the meshing algorithm progressively incorporates nodes to achieve greater detail, the prescribed element sizes within the specified geometric zones act as crucial criteria. This mechanism ensures that the resulting mesh maintains a high level of accuracy precisely where it is most pertinent.

3. Consistency Across Interfaces: In cases where a boundary surface mesh coincides with one of these shape zones, it is advisable to maintain a harmonious element size between the shape zone and the boundary surface mesh. This alignment contributes to a seamless transition and mesh quality preservation at the interface.

4.3 Adaptive Meshing

In the realm of Fluid-Structure Interaction (FSI) simulations, our solver employs an Arbitrary Lagrangian-Eulerian (ALE) technique for mesh motion and adaptivity. This approach allows for substantial deformations in the fluid mesh. By default, the solver rebuilds the mesh solely in cases where elements become inverted, a phenomenon more prevalent in rotational scenarios like wind turbine analysis.

However, the ICFD_CONTROL_ADAPT_SIZE card offers an avenue for proactive re-meshing. This feature enables the solver to recognize instances where mesh distortion arises due to the mesh movement and helps the solver maintain a mesh size prescribed by the user. Such distortion often emerges when dealing with problems involving translational motion of bodies.

In essence, the current approach offers both default and user-triggered methods to address mesh deformation challenges in FSI simulations.

4.4 Gap Closure Treatment

In the case of body fitted meshes a complete blockage of flow through small gaps could be challenging. This is because the meshing algorithm will continue to insert elements in the gap region, preserving continuity of velocity and pressure and thus, transporting mass through the gap. In some scenarios, as shown in Fig.4, it may be desirable to completely block parts of the domain where boundaries are close enough.

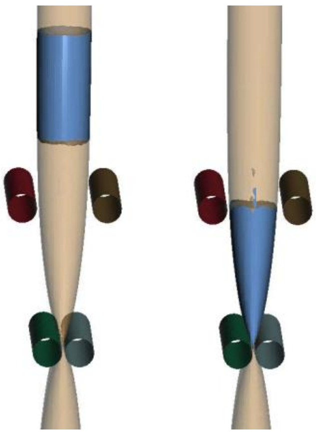

Fig 4. Fluid flow through a thin channel

LS-DYNA ICFD addresses this issue using the keyword *ICFD_CONTROL_GAP. If the distance between contacting bodies is less than a threshold value, the flow is blocked at this region. This feature relies on,

1) Gap Detection: When the sum of the distances from a node to two surfaces is less than threshold value, the node is in gap zone. While an element completely in gap zone is a gap element.

2) Gap Treatment: The identified gap nodes are put in different groups and marked as “Gap Nodes” (yellow) and “Isoloated Nodes” (blue) respectively [Fig. 5]. The Gap Nodes are applied with zero velocity condition and the Isolated Nodes undergo a reconstruction of pressure field based on the pressure field across the contact locations.

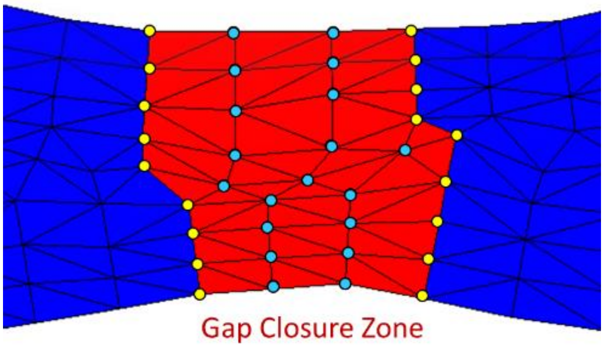

Fig.5: Gap zone painted in red.

4.5 DEM Coupling

The ICFD solver offers a unique solution for situations where achieving a perfectly sealed mesh (eg. complex contact) in Fluid-Structure Interaction (FSI) simulations becomes challenging. This innovative feature involves coupling the structural mesh with the fluid mesh through the utilization of Discrete Element Method (DEM) particles at the interface. This approach becomes particularly relevant in scenarios where the structure is subjected to high-speed motion, substantial deformation, and intricate contact interactions.

The activation of the feature is made possible through the utilization of the keyword *ICFD_CONTROL_DEM_COUPLING. Using this the users are allowed to choose,

1) The type of coupling (one-way/two-way)
2) The type of DEM particles needed (active/inactive)
3) The size of the particles
4) The type of force transfer between the fluid and the structure (drag-based/pressure-based)

Within the Discrete Element Method (DEM), discretization occurs through the implementation of interacting spheres. These spheres engage through imaginary springs upon contact, generating forces that causes the particle movement. Each sphere possesses distinct attributes such as mass, radii, and spatial coordinates. When DEM is integrated with ICFD, each sphere becomes associated with a corresponding host finite element within the fluid mesh. This coupling imparts a force to the sphere, thereby influencing its behavior within the simulation.

This approach presents a promising avenue for effectively addressing FSI scenarios that require interface representation beyond the confines of traditional meshing techniques.

5 Best Practices

The realm of fluid-structure interaction simulations can be quite intimidating due to the complexities in solution methodologies and model development. This discussion zeroes in on some best practices that can be followed while starting to build the model. These best practices are the compass that can steer the users towards accurate and efficient simulations. Whether you're new to this or a seasoned engineer, these tips will help you make the most of LS-DYNA ICFD, unlocking valuable insights from your FSI simulations.

5.1 Meshing at the FSI Interface

When building a model for Fluid-Structure Interaction (FSI) simulations, there are several considerations that differ from creating a mesh for typical structural or fluid simulations. FSI simulations involve the coupling of fluid and structural domains, introducing additional complexities and challenges.

Interface Meshing in FSI simulations require a well-meshed interface between the fluid and structural domains to accurately capture the interaction between the two as the interface serves as the region where vital information (force and displacement) is exchanged. It always is a good practice to not have any gaps between the structural mesh and its fluid counterpart, which can be visually verified during the model build process. If a mesh with gaps is used in the simulation, the user may encounter “ICFD:: WARNING: Free FSI node detected” message which may adversely affect the results.

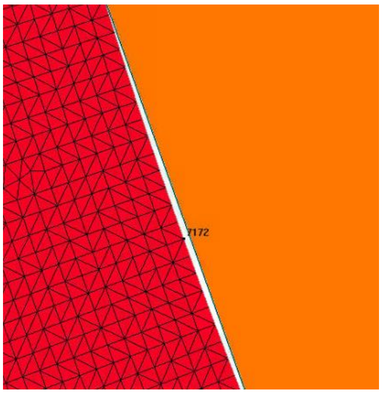

Fig.6: Inappropriate meshing at Interface.

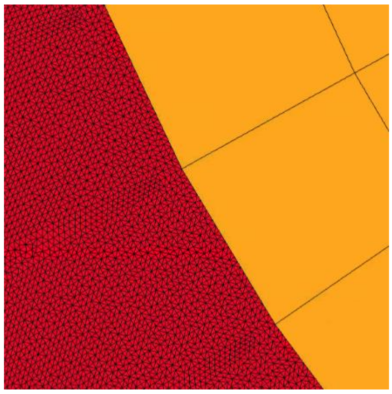

Fig.7: Appropriate meshing at Interface.

By default, LS-DYNA connects FSI nodes if the distance between the fluid mesh and the structural mesh is less than *IDC*min(h, H)*, where IDC is a factor equal to 0.25, *h* and *H* are fluid and structural mesh sizes. This can be honored by either having the same mesh size on both the sides of the interface or by increasing the factor “*IDC”* using *ICFD_CONTROL_FSI.

5.2 Level Set

Multiphase fluids involve dynamic combinations of two or more fluids, with understanding their behavior often proving convoluted. This complexity deepens when incorporating Fluid-Structure Interaction (FSI). In addressing these challenges, LS-DYNA employs a level-set approach for multiphase fluid simulations. Preprocessing offers *MESH_INTERF and *INITIAL_LEVEL_SET keywords, each with merits and demerits. However, prioritizing *INITIAL_LEVEL_SET is encouraged due to its efficiency and time-saving benefits in model development. Notably, *INITIAL_LEVEL_SET excels when modeling multi-fluid domains as standard shapes (box, sphere, cylinder) require only coordinates, unlike *MESH_INTERF, which demands separate surface entities for each fluid.

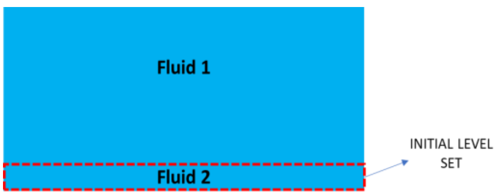

Fig.8: Level set approach

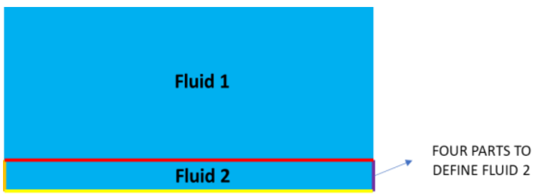

Fig.9: Mesh interface approach

In Fig. 8, the *INITIAL_LEVEL_SET is employed by the user to define the boundary of "Fluid-2." This process only involves providing the coordinates of the red box. However, if the user had to achieve the same outcome using *MESH_INTERF, it would have demanded the creation of four separate parts as depicted in Figure 9. This approach would be more time intensive.

5.3 Mesh Projection

Projecting the fluid mesh onto the structural mesh is pivotal in Fluid-Structure Interaction (FSI) simulations as it ensures accurate representation of interface interactions and seamless information transfer between the fluid and structural domains. When intense rotation, or high-speed translation is expected on the structure, the users are recommended to set XPROJ=1 in *ICFD_CONTROL_FSI.

5.4 Contacts in FSI simulations

Fluid-Structure Interaction (FSI) simulations require a minimum gap between contacting bodies to allow for fluid volume mesh generation. Attempting to run a simulation with zero gaps will trigger a "Volume Mesh Error," halting the simulation. This gap is regulated by the movement of structural components in the FSI model. To maintain a non-zero gap, consider the below approaches,

1. For shell elements, assign non-zero values to SST/MST parameters in the *CONTACT keyword to introduce a gap.
2. For solid elements, employ SLDTHK in *CONTACT optional card B to establish an artificial contact thickness.

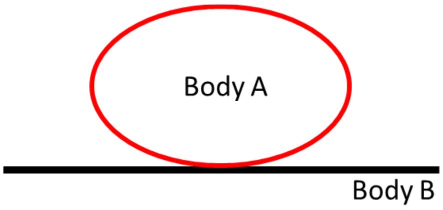

Fig 10: Zero gap between Body A and Body B.

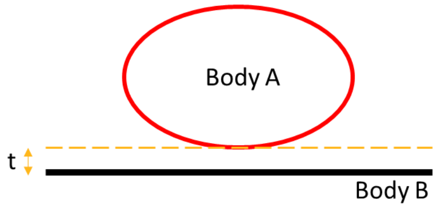

Fig 11: A gap of thickness t between Body A and Body B.

The outlined approach becomes necessary only when the thickness of the volume elements significantly exceeds the size of the gap. Caution is advised when adjusting these parameters, as higher values may lead to results deviating from real-world behavior.

6 Summary

This paper offers a comprehensive overview of the LS-DYNA ICFD (Incompressible Computational Fluid Dynamics) solver, focusing on practical guidelines for its effective application in tackling intricate Fluid-Structure Interaction (FSI) challenges. The LS-DYNA ICFD solver stands as a potent tool for simulating interactions between fluids and solid structures, facilitating precise predictions of real-world phenomena. The paper delves into the foundational principles of the ICFD solver, highlighting its distinctive attributes. It also furnishes insights into some of the best practices that can be utilized, to quickly overcome any challenges with the model. This work is dedicated to offering a clear, concise, and easy-to-comprehend guide to engineers and researchers, regardless of their expertise.

7 Literature

[1] White, F. M. (2008). Fluid Mechanics (7th ed.). McGraw-Hill Education.
[2] Cengel, Y. A., & Cimbala, J. M. (2014). Fluid Mechanics: Fundamentals and Applications. McGraw-Hill Education.
[3] Fox, R. W., McDonald, A. T., & Pritchard, P. J. (2004). Introduction to Fluid Mechanics (7th ed.). John Wiley & Sons.
[4] Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of Computational Physics, 39(1), 201-225.
[5] Osher, S., & Sethian, J. A. (1988). Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. Journal of Computational Physics, 79(1), 12-49.
[6] Bathe, K. J., & Zhang, H. (2005). Finite element developments for general fluid flows with structural interactions. Computers & Structures, 83(17-18), 1415-1428.
[7] ICFD Theory Manual, Incompressible fluid solver in LS-DYNA.
[8] Facundo Del Pin, Inaki Calichoury, Rodrigo R.Paz. Review and Advances of Coupling Methods for the ICFD solver in LS-Dyna.