MULTIFLUID FLOW

INTRODUCTION

We have seen how to deal with dynamic in time fluid-structure interaction problems:

- Arbitrary Lagrangian – Eulerian Methods.
- Fixed-Mesh methods.

In MultiFluid flow problems there are also several approaches which can be taken for the tracking of the interface:

- Lagrangian methods
  - Particle Finite Element Method.
- Eulerian methods
  - Volume of fluid.
  - Level set method.
  - Pressure enrichment for incompressible flows.
  - X-FEM.

INTRODUCTION

Statement of the problem

Let us consider the two-phase flow (water and air problem) for the incompressible Navier-Stokes equations:

$$\rho_1 \partial_t \boldsymbol{u} + \rho_1 \boldsymbol{u} \cdot \boldsymbol{\nabla}\mathbf{u} - \mu_1 \Delta \boldsymbol{u} + \boldsymbol{\nabla}\mathrm{p} = \rho_1 \mathbf{f} \qquad \text{in } \Omega_1$$
$$\nabla \cdot \boldsymbol{u} = 0 \qquad \text{in } \Omega_1$$

$$\rho_2 \partial_t \boldsymbol{u} + \rho_2 \boldsymbol{u} \cdot \boldsymbol{\nabla}\mathbf{u} - \mu_2 \Delta \boldsymbol{u} + \boldsymbol{\nabla}\mathrm{p} = \rho_2 \mathbf{f} \qquad \text{in } \Omega_2$$
$$\nabla \cdot \boldsymbol{u} = 0 \qquad \text{in } \Omega_2$$

Transmission conditions:

$$\boldsymbol{u}_1 = \boldsymbol{u}_2 \qquad \text{on } \Gamma$$
$$-p_1 \cdot \boldsymbol{n} + \mu_1 \mathbf{n} \cdot \nabla \mathbf{u}_1 = -p_2 \cdot \boldsymbol{n} + \mu_2 \mathbf{n} \cdot \nabla \mathbf{u}_2 \qquad \text{on } \Gamma$$

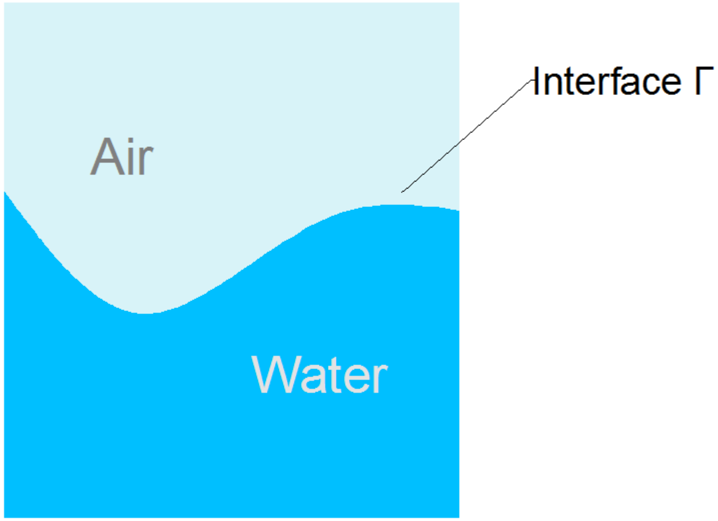
INTRODUCTION

Statement of the problem

Let us consider the two-phase flow (water and air problem) for the incompressible Navier-Stokes equations:

$$\rho_1 \partial_t \boldsymbol{u} + \rho_1 \boldsymbol{u} \cdot \boldsymbol{\nabla} \mathbf{u} - \mu_1 \Delta \boldsymbol{u} + \boldsymbol{\nabla} \mathrm{p} = \rho_1 \mathbf{f} \qquad \text{in } \Omega_1$$
$$\nabla \cdot \boldsymbol{u} = 0 \qquad \text{in } \Omega_1$$

$$\rho_2 \partial_t \boldsymbol{u} + \rho_2 \boldsymbol{u} \cdot \boldsymbol{\nabla} \mathbf{u} - \mu_2 \Delta \boldsymbol{u} + \boldsymbol{\nabla} \mathrm{p} = \rho_2 \mathbf{f} \qquad \text{in } \Omega_2$$
$$\nabla \cdot \boldsymbol{u} = 0 \qquad \text{in } \Omega_2$$



Transmission conditions:

$$\boldsymbol{u}_1 = \boldsymbol{u}_2 \qquad \text{on } \Gamma$$
$$-p_1 \cdot \boldsymbol{n} + \mu_1 \mathbf{n} \cdot \nabla \mathbf{u}_1 = -p_2 \cdot \boldsymbol{n} + \mu_2 \mathbf{n} \cdot \nabla \mathbf{u}_2 \quad \text{on } \Gamma$$

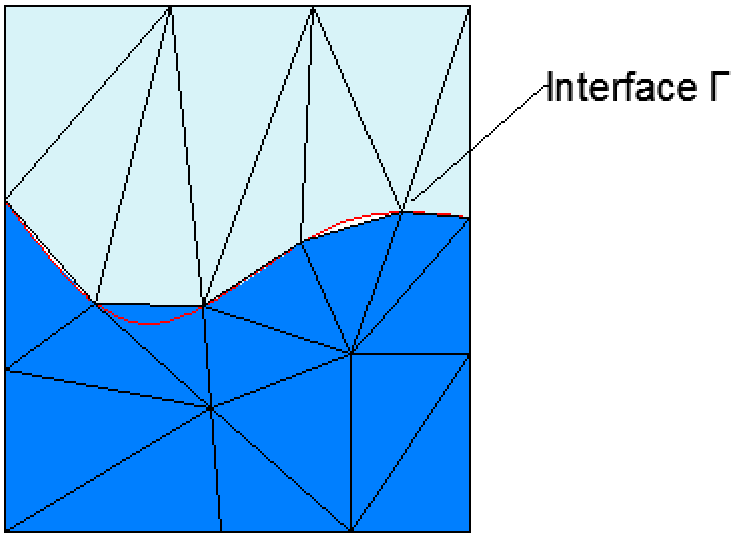
LAGRANGIAN METHODS

Lagrangian methods for the tracking of the interface

- In a Lagrangian tracking of the interface, we use an Arbitrary Lagrangian-Eulerian method.

- The nodes representing the interface are treated in a Lagrangian way.

- If the flow is not too complex this is a good solution:
  - Sharp tracking of the interface
  - No need of additional artifacts
  - Velocity gradients can be discontinuous at the interface
  - If the pressure interpolation space is discontinuous, then pressure can be discontinuous at the interface.

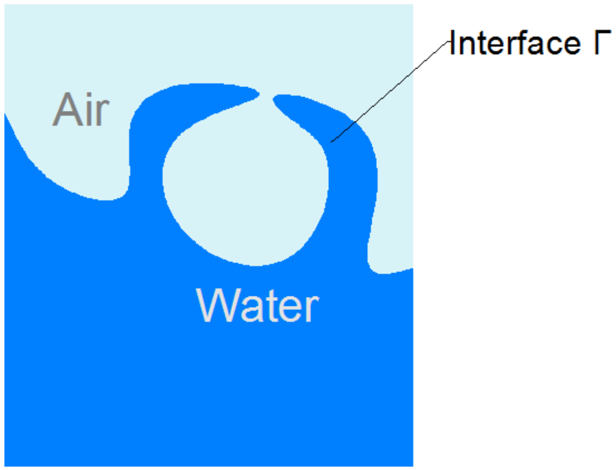


LAGRANGIAN METHODS

Lagrangian methods for the tracking of the interface

However in more complex flows it can lead to:

- Mesh distortion.
- Element folding.
- How to deal with a volume of water closing around a volume of air?

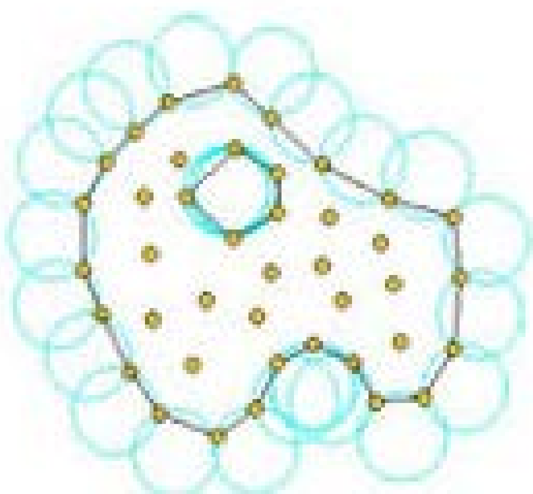


LAGRANGIAN METHODS

The Particle Finite Element Method (P-FEM)

The Particle Finite Element Method consists precisely in adopting a purely Lagrangian approach for the simulation of both fluids

All the nodes of the mesh are Lagrangian => each node corresponds to a particle. When the mesh gets too distorted: remeshing (but keeping the same nodes).



Identifying the boundary nodes in the PFEM method. An air bubble is captured inside the water domain. Remeshing will respect this boundary.

LAGRANGIAN METHODS

The Particle Finite Element Method (P-FEM)

At each time step, the PFEM procedure is the following:

1. Identify the surface boundary between the two fluids.
2. Discretize both subdomains with a finite element mesh (remeshing, only the connectivities change).
3. Solve the Lagrangian equations of motion for the two-phase problem. Both monolithic or partitioned approaches are possible.
4. Move the mesh nodes to the position corresponding to the next time step.
5. Go back to step 1 for the next time step.

The advantage is that we can use a Lagrangian method for the simulation of fluids.

On the other hand, we need to remesh at each time step and implement an algorithm for identifying the interface (Alpha – Shape Method).

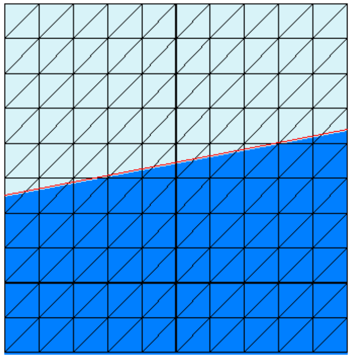
EULERIAN METHODS

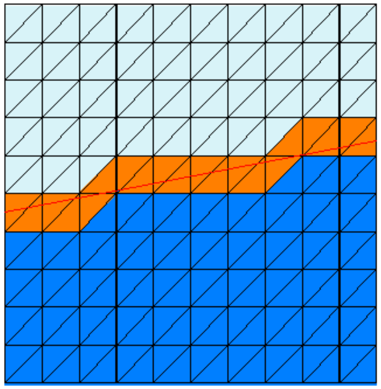
Volume of Fuid Method

Eulerian methods use a completely different approach for the tracking of the interface between the two fluids.
The Volume of Fluid method is based on a scalar fraction function C which extends over the whole computational domain.

The function is piecewise constant for each element, and discontinuous at the element interfaces.

For a cell full of air: $C = 0$ For a cell full of water: $C = 1$
At cells which are cut by the interface: $0 < C < 1$.





EULERIAN METHODS

Volume of Fuid Method

The fraction function needs to be transported with the fluid velocity $\boldsymbol{u}$.

We solve a purely advection equation over the whole computational domain $\Omega = \Omega_1 \cup \Omega_2$:

$$\partial_t C + \boldsymbol{u} \cdot \nabla C = 0$$

The main problem for solving this equation is that C is discontinuous.

Many times it is solved discretely by advecting points at the end of the interface.

Mass conservation can be easily enforced through small corrections in the advected function.

Problem: The interface is not sharp: A fluid at rest will not necessarily remain at rest.

EULERIAN METHODS

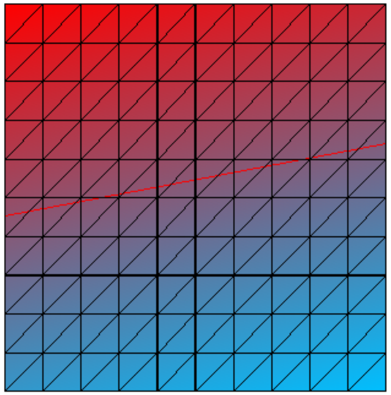
The Level Set Method

The objective of the level set method is to improve the behavior of the VOF method by allowing a sharp tracking of the interface.

- The idea is similar to the VOF metod, but the level set function is not discontinuous.

- Instead, ψ is a continuous function which denotes the signed distance to the interface:

$$\psi \begin{cases} < 0 & \boldsymbol{x} \in \Omega_1 \\ = 0 & \boldsymbol{x} \in \Gamma \\ > 0 & \boldsymbol{x} \in \Omega_2 \end{cases}$$



EULERIAN METHODS

The Level Set Method

At each time step we need to advect the level set function by using the velocity of the fluid as advection velocity.

$$\partial_t \psi + \boldsymbol{u} \cdot \nabla \psi = 0$$

We can easily solve this equation by using finite elements, finite differences … Stabilization is required.

Integration:
We subdivide the elements which are cut by the interface.

- We integrate the air equations in the Gauss points in Ω_1.
- We integrate the water equations in the Gauss points in Ω_2.

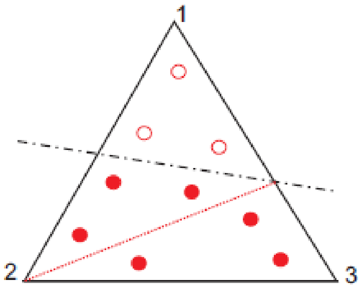


Figure 4.3: Splitting of elements

The interface is now sharp.

EULERIAN METHODS

Initial computation of the distance function

In the initial configuration we need to initialize the distance function ψ.

First approach:

- We consider the interface to be represented by straight segments (surfaces in 2D) where $\psi = 0$ inside each element.
- For all points of the mesh
    - For segments of the interface

Check if this is the minimum distance from the point to the level-set interface

This can be very costly $\mathcal{O}(n^2)$

Octrees and binary searches can be used to improve the performance of the initialization algorithm.

EULERIAN METHODS

Improved methods for the initialization of the level set function

Our objective is to obtain a level-set function which is as smooth as possible (signed distance).

We can iteratively solve the pseudo-transient problem:

$$\begin{aligned} \partial_t \psi + |\nabla \psi| = 1 \quad & \text{on } \Omega \\ \psi = 0 \quad & \text{on } \Gamma \end{aligned}$$

When reaching steady state we will have the effective signed distance and:

$$|\nabla \psi| = 1$$

It is convenient to reinitialize the level set function every several time steps of the simulation in order to keep it smooth (small gradients)