Numerical Simulation of Bubble Drag Reduction and Air Layer Drag Reduction

Xiaosong Zhang

State Key Laboratory of Ocean Engineering,
School of Naval Architecture, Ocean and Civil Engineering,
Shanghai Jiao Tong University

CMHL Symposium, Shanghai, Dec. 13, 2019

OUTLINE

- **Background and Motivation**
- **Bubble Drag Reduction**
  - **Development of the bubble flow solver**
  - **Bubble drag reduction in turbulent boundary layer**
- **Air-Layer Drag Reduction**
  - **Steramwise characteristics**
  - **Air layer in a cavity**
- **Conclusion and Future works**

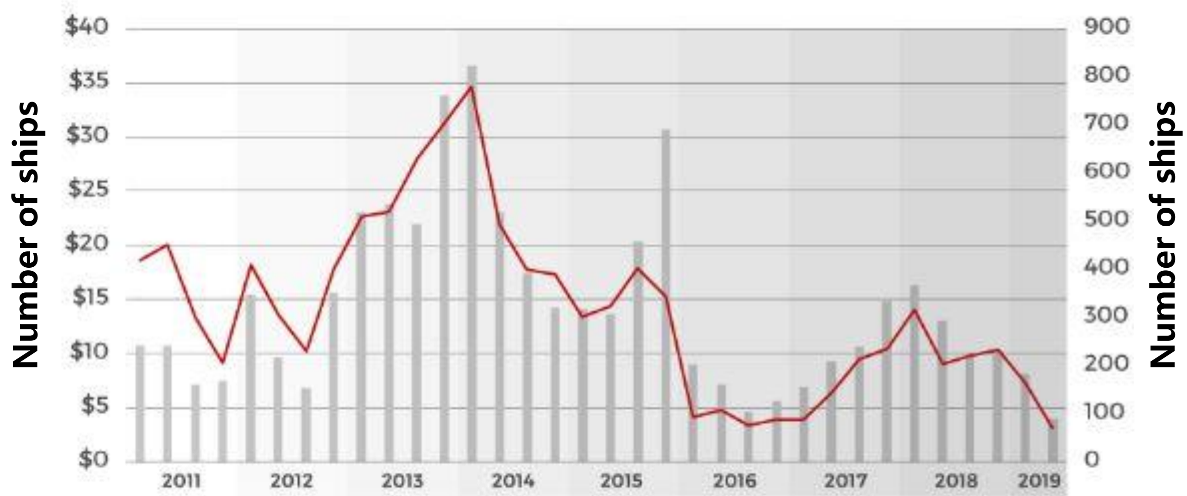
OUTLINE

- **Background and Motivation**
- **Bubble Drag Reduction**
  - Development of the bubble flow solver
  - Bubble drag reduction in turbulent boundary layer
- **Air-Layer Drag Reduction**
  - Steramwise characteristics
  - Air layer in a cavity
- **Conclusion and Future works**

Background and Motivation

Reducing the fuel consumption of ships has always been an important goal in ship design and management, especially against the background of the shipping industry recession in recent years.

Total orders of ship all over the world

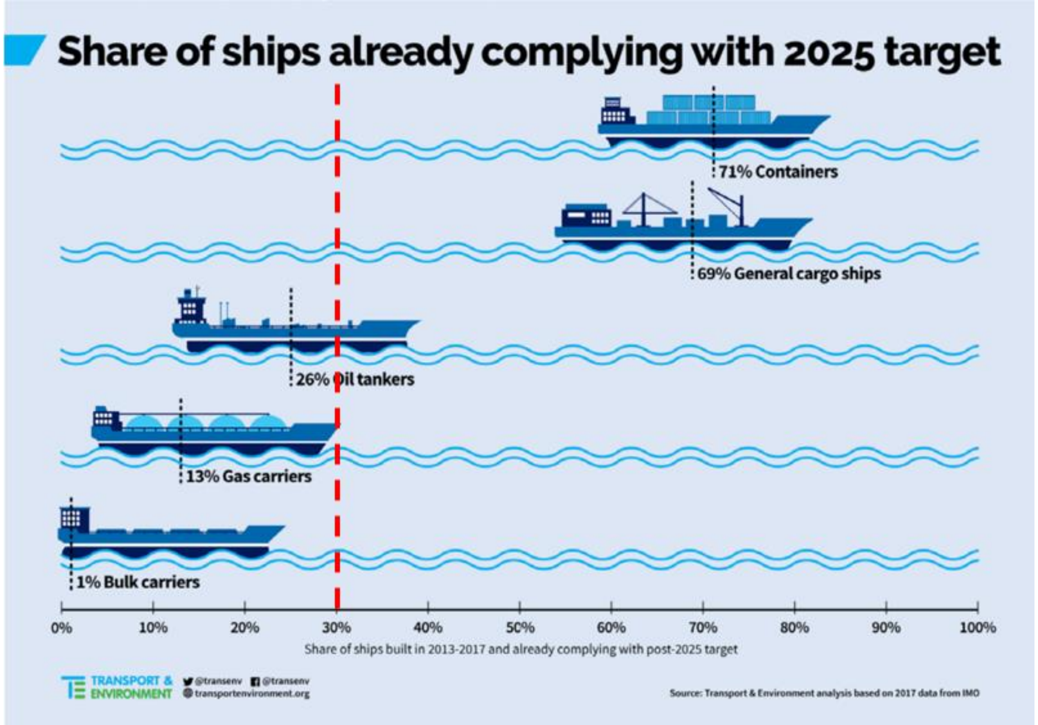


Years

Background and Motivation

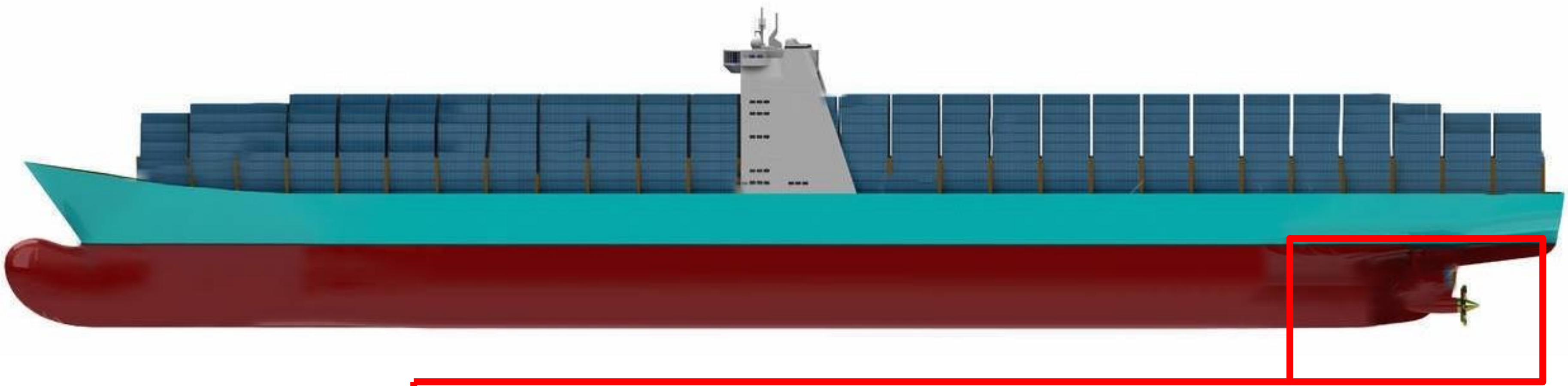
INTERNATIONAL MARITIME ORGANIZATION

Phases	Year of ship built	Energy saving to the baseline
0	2013-2015	0
1	2015-2020	10%
2	2021-2025	20%
3	2025- (Maybe 2022)	30%



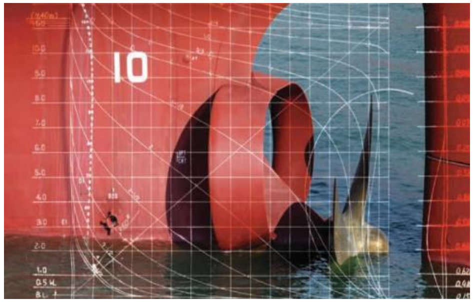
Most of the Oil tankers, Gas carries and Bulk carriers are far away from the requirement of phase-3.

Background and Motivation



Propulsion

Energy Saving Device



Wake optimization in front of propeller

Wake Equalizing Duct

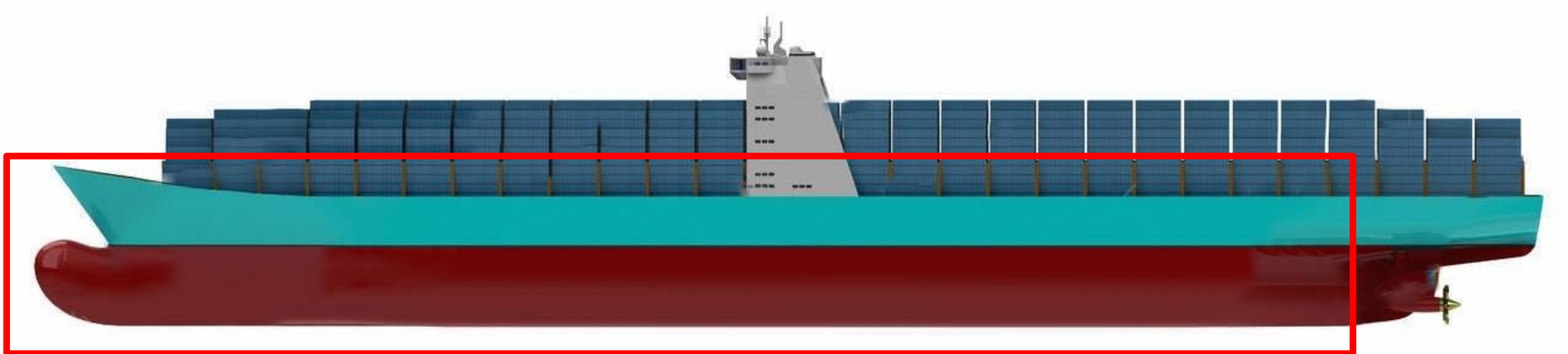
Energy recovery behind propeller

Rudder Ball

Vortex elimination

Propeller Boss Cap Fins

Background and Motivation



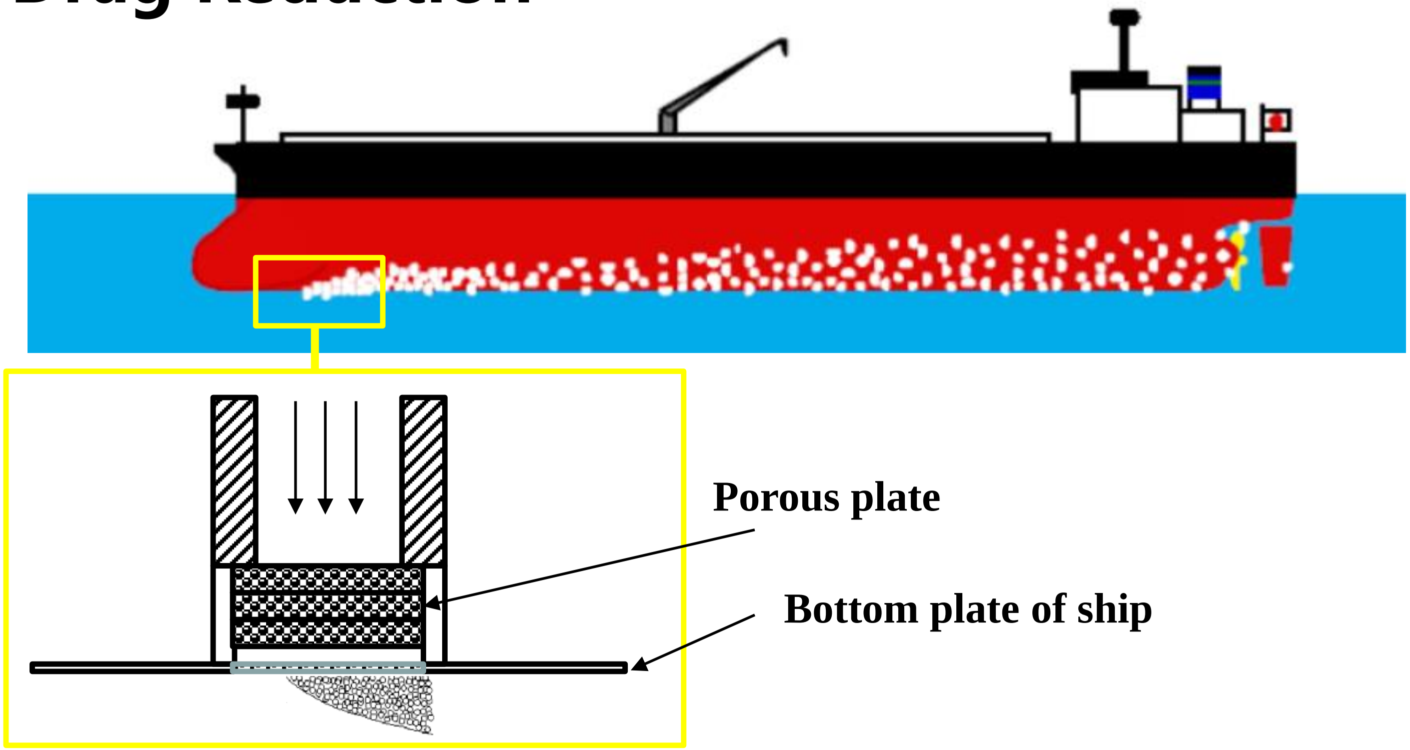
Ship Hull

Drag Reduction Techniques

- **Hull form optimization**
- **Super-hydrophobic coating**
- **Air lubrication**

Background and Motivation

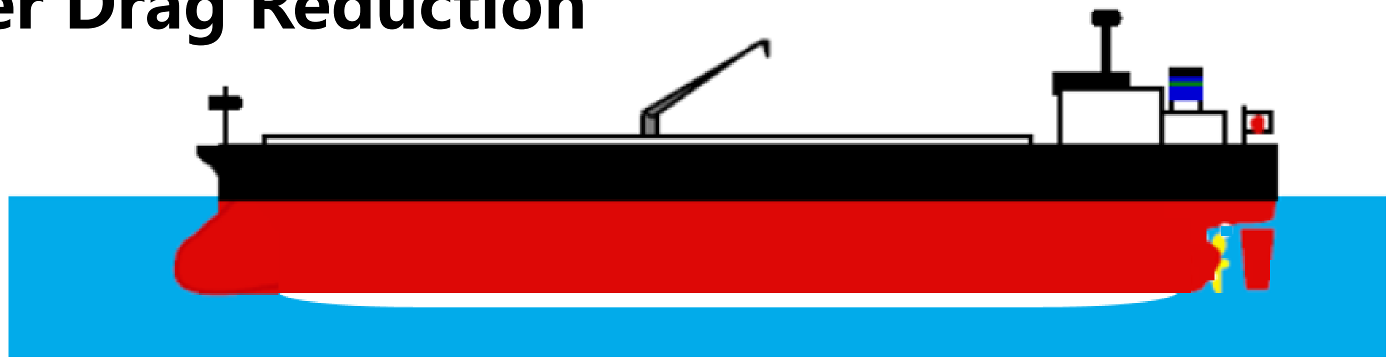
Bubble Drag Reduction

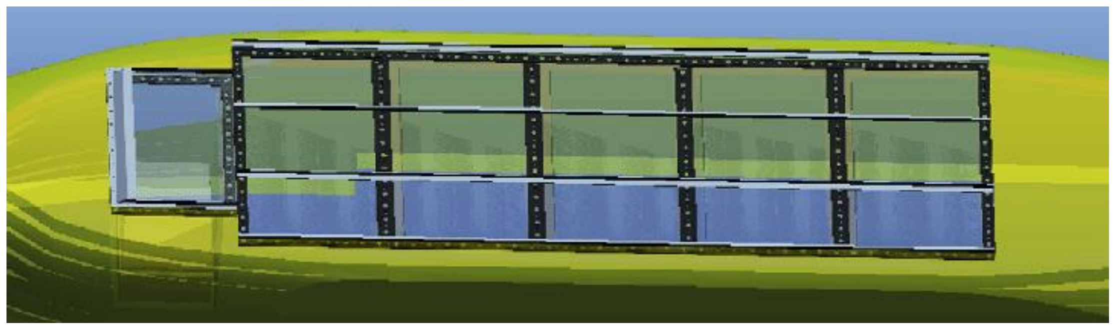


- Submillimeter microbubbles are produced through porous plates
- Microbubbles should enter the turbulent boundary layer

Background and Motivation

Air-Layer Drag Reduction





- A complete layer of air is formed to adhere to the bottom of the ship with relatively large air injection flow rate.
- Separate most of the bottom plate directly from water, reducing the wetted surface area

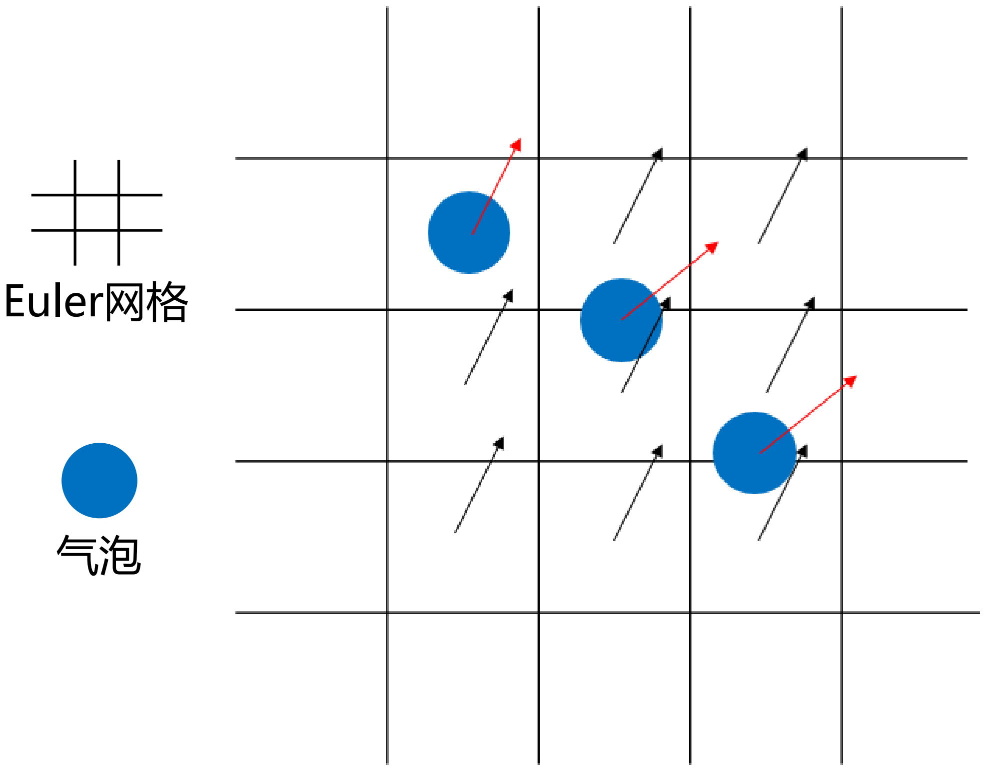
OUTLINE

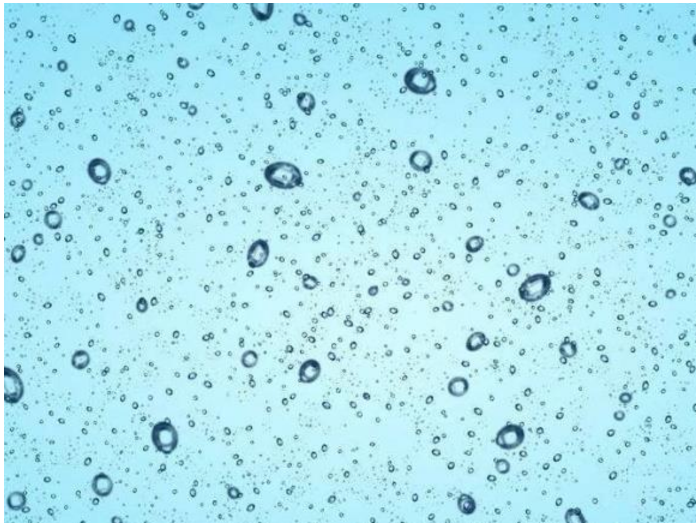
- Background and Motivation
- **Bubble Drag Reduction**
  - **Development of the bubble flow solver**
  - Bubble drag reduction in turbulent boundary layer
- Air-Layer Drag Reduction
  - Steramwise characteristics
  - Air layer in a cavity
- Conclusion and Future works

Development of the bubble flow solver

➢ Basic numerical method

Euler-Lagrange method is used to model the flow mixed with a large number of discrete bubbles.

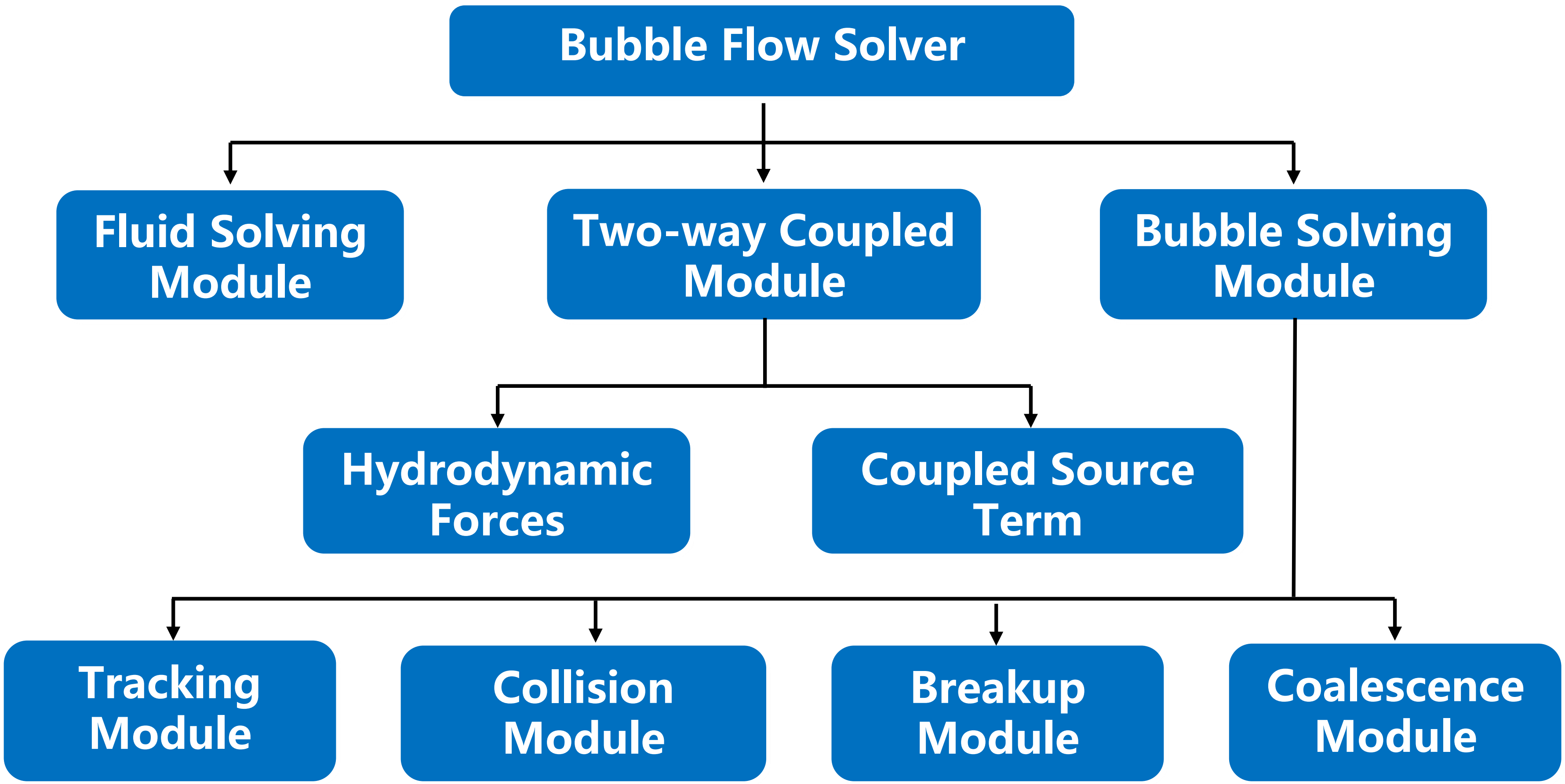




- ✓ **The liquid flow is solved on the grid based on Euler framework.**
- ✓ **The motion of each bubble is tracked individually by solving the kinematic equation based on Lagrange framework.**

Development of the bubble flow solver

➢ Main modules in the solver



Development of the bubble flow solver

➢ Governing equation for bubble motion:

$$m\frac{dv}{dt} = f_D + f_L + f_P + f_G + f_C$$

$$= \frac{3mC_D}{4d}|u-v|(u-v) + \frac{m\rho_l}{\rho_b}C_L(u-v)\times(\nabla\times u) + \frac{m\rho_l}{\rho_b}\frac{Du}{Dt} + mg\left(1-\frac{\rho_l}{\rho_b}\right) + f_C$$

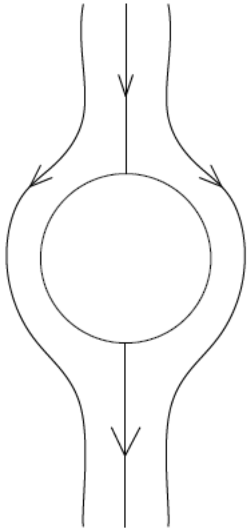
Drag | Lift | Pressure Gradient | Buoyancy | Collision force

Drag coefficient C_D and lift coefficient C_L are obtained by models

Drag coefficient:

Tomiyama drag model:

$$C_D = \max\left(\min\left(\frac{16}{\mathrm{Re}}\left(1+0.15\,\mathrm{Re}^{0.687}\right), \frac{48}{\mathrm{Re}}\right), \frac{8}{3}\frac{Eo}{Eo+4}\right)$$



Development of the bubble flow solver

➢ **Governing equation for bubble motion:**

$$m\frac{dv}{dt} = f_D + f_L + f_P + f_G + f_C$$

$$= \underbrace{\frac{3mC_D}{4d}|u-v|(u-v)}_{\text{Drag}} + \underbrace{\frac{m\rho_l}{\rho_b}C_L(u-v)\times(\nabla\times u)}_{\text{Lift}} + \underbrace{\frac{m\rho_l}{\rho_b}\frac{Du}{Dt}}_{\text{Pressure Gradient}} + \underbrace{mg\left(1-\frac{\rho_l}{\rho_b}\right)}_{\text{Buoyancy}} + \underbrace{f_C}_{\text{Collision force}}$$

Drag coefficient C_D and lift coefficient C_L are obtained by models

- **Lift coefficient:**

Tomiyama lift model:

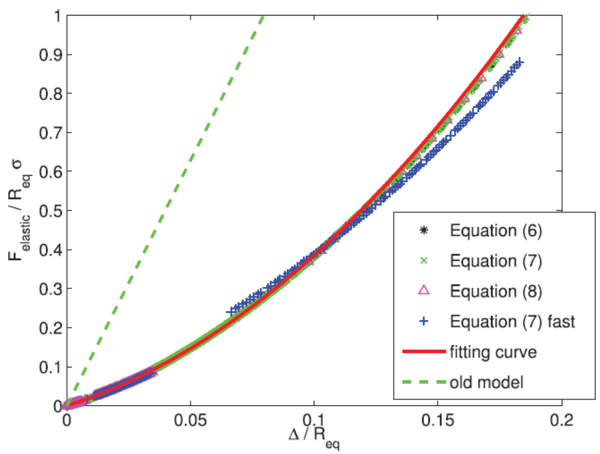
$$C_L = \begin{cases} \min\left[0.288\tanh(0.121\mathrm{Re}),\, f(Eo_d)\right] & Eo_d < 4 \\ f(Eo_d) & 4 \le Eo_d \le 10.7 \end{cases}$$

$$f(Eo_d) = 0.00105Eo_d^3 - 0.0159Eo_d^2 - 0.0204Eo_d + 0.474$$

Development of the bubble flow solver

➢ Collision modeling:

Bubble collision is modeled by a elastic soft sphere model. A non-linear collide force model is adopted.



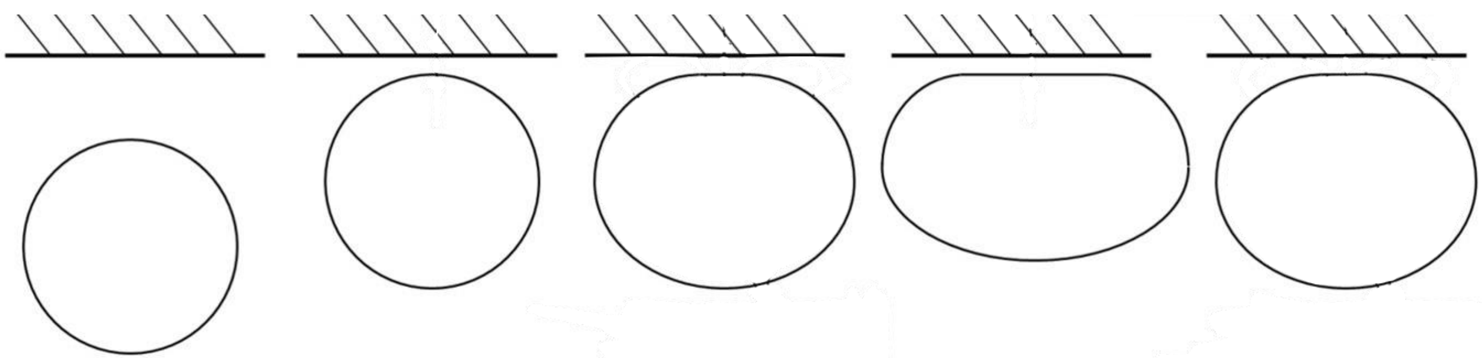
Heitkam S , et al. A simple collision model for small bubbles[J]. Journal of Physics: Condensed Matter, 2017, 29(12):124005.

Elastic force

$$F_{elastic} = 18.5\sigma\left(\frac{\Delta}{R_{eq}}\right)^2 + 2.0\Delta\sigma$$

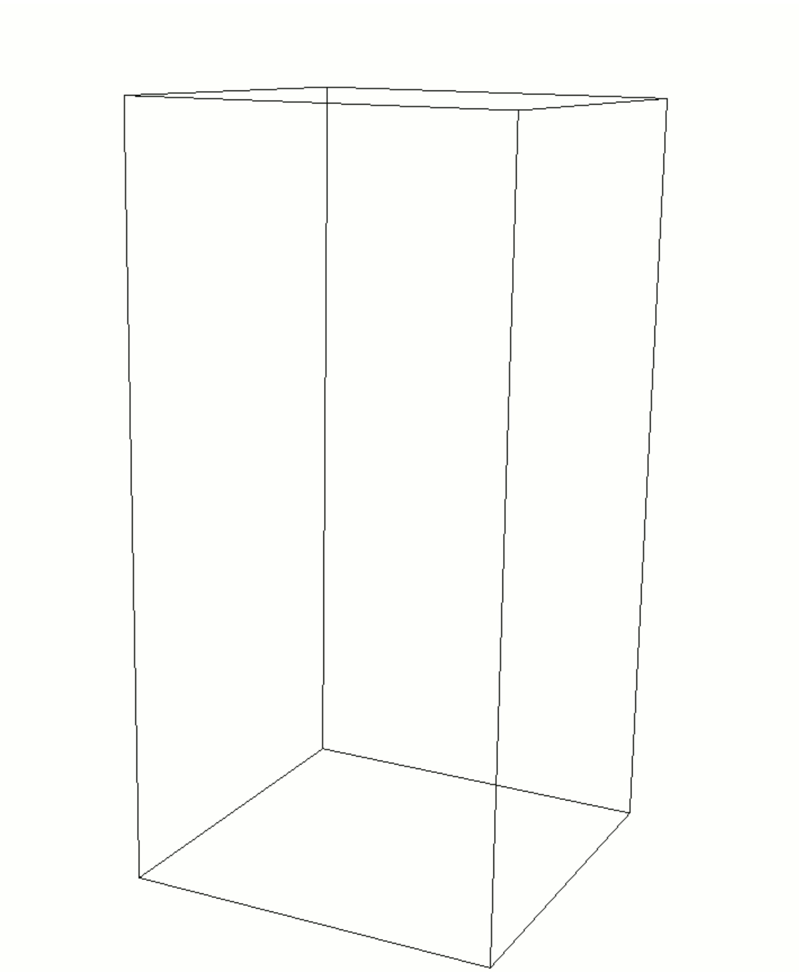
Viscous force

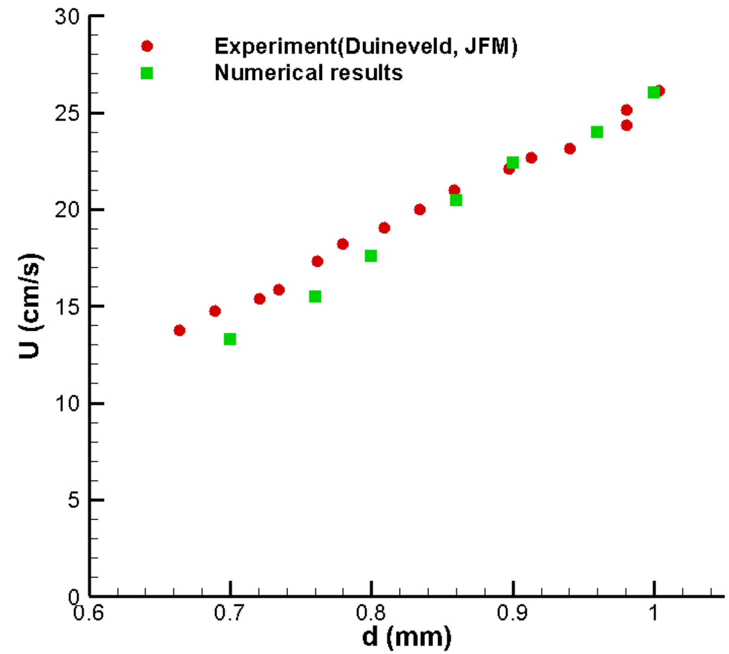
$$F_{viscous} = uC_{bc}\frac{12\mu_l}{2\pi}0.34\left(\frac{\Delta}{R_{eq}} + 0.0002\right)^{-0.5} \times \left(4.0\sqrt{\frac{R_{eq}^3}{h_0}} + 3.0R_a\frac{R_{eq}}{h_0}\right)$$



Development of the bubble flow solver

➢ Model Validation——Microbubble Rise Up

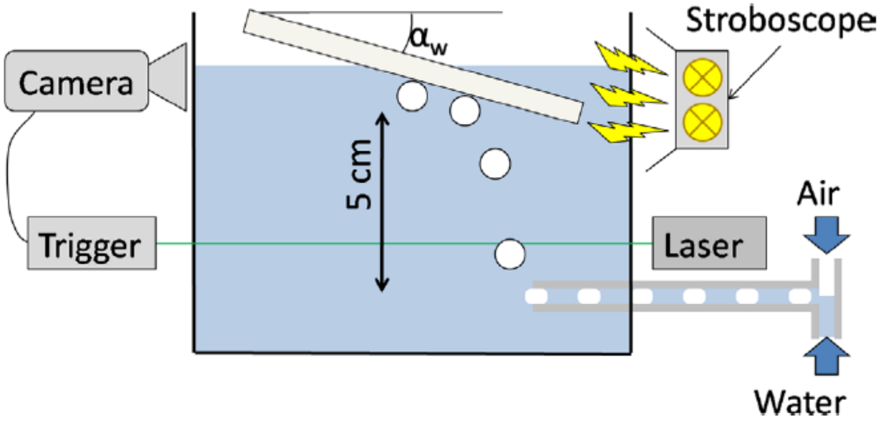


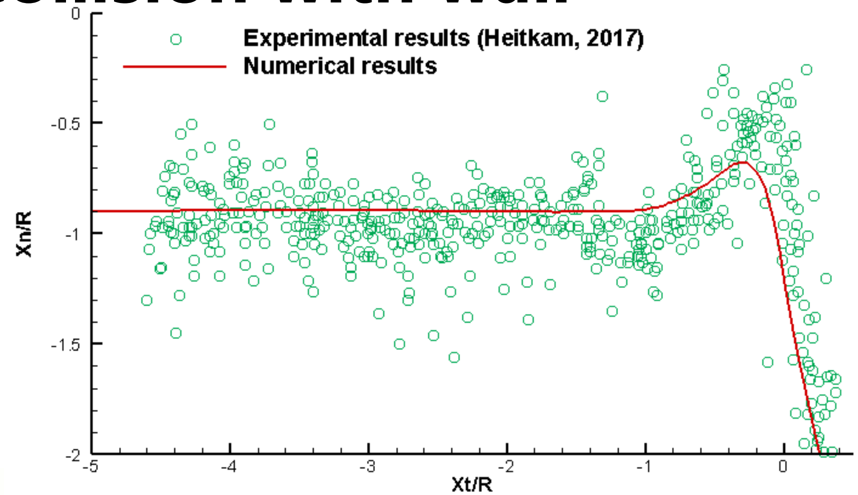


The rising velocity of single microbubble is in good agreement with the experimental results, which proves the accuracy of the computational hydrodynamic forces on the microbubble.

Development of the bubble flow solver

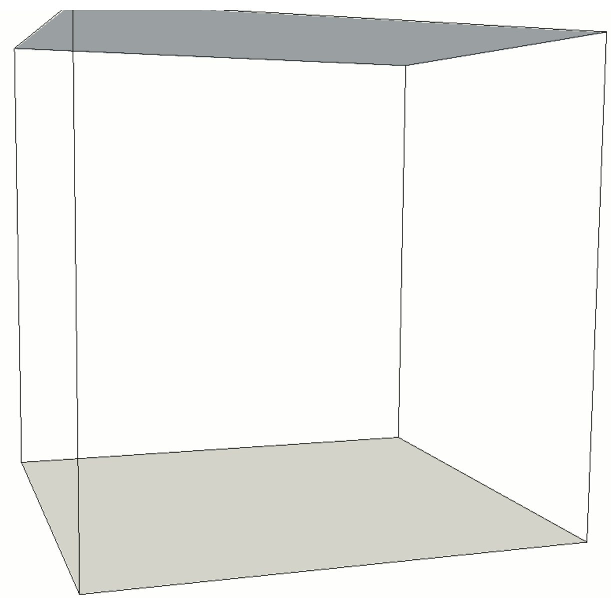
➢ **Model validation——Collision with wall**





The accuracy of collision force calculation is validated by deformation and trajectory of microbubble colliding with a plate obliquely.

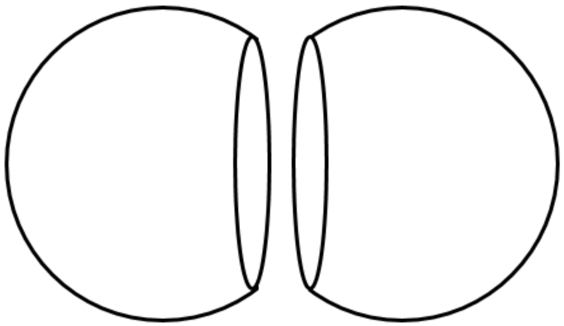
The numerical results are in good agreement with the experimental data.



Development of the bubble flow solver

➢ Coalescence & Breakup

Film drainage model:



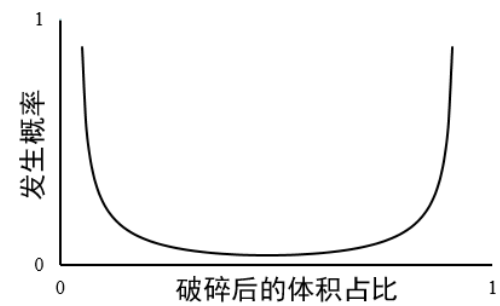
If two bubbles contact long enough to drain the liquid film between them, then coalescence happen

Critical We number criteria:

$$We_{crit} = \frac{\rho_l \delta u(d)^2 d}{\sigma}$$

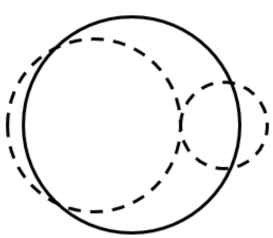
Daughter bubble size distribution:

$$f(\gamma) = \frac{1}{\pi\sqrt{\gamma(1-\gamma)}}$$



Conservation:

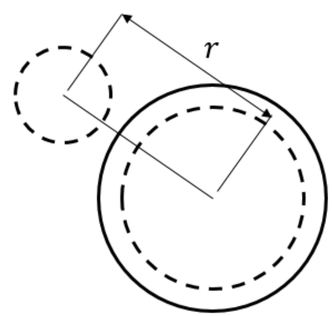
$$Position_c = \frac{d_a Position_a + d_b Position_b}{d_a + d_b}$$



$$d_c = \left(d_a^3 + d_b^3\right)^{1/3}$$

$$U_c = \frac{\left(d_a^3 U_a + d_b^3 U_b\right)}{\left(d_a^3 + d_b^3\right)}$$

Position:

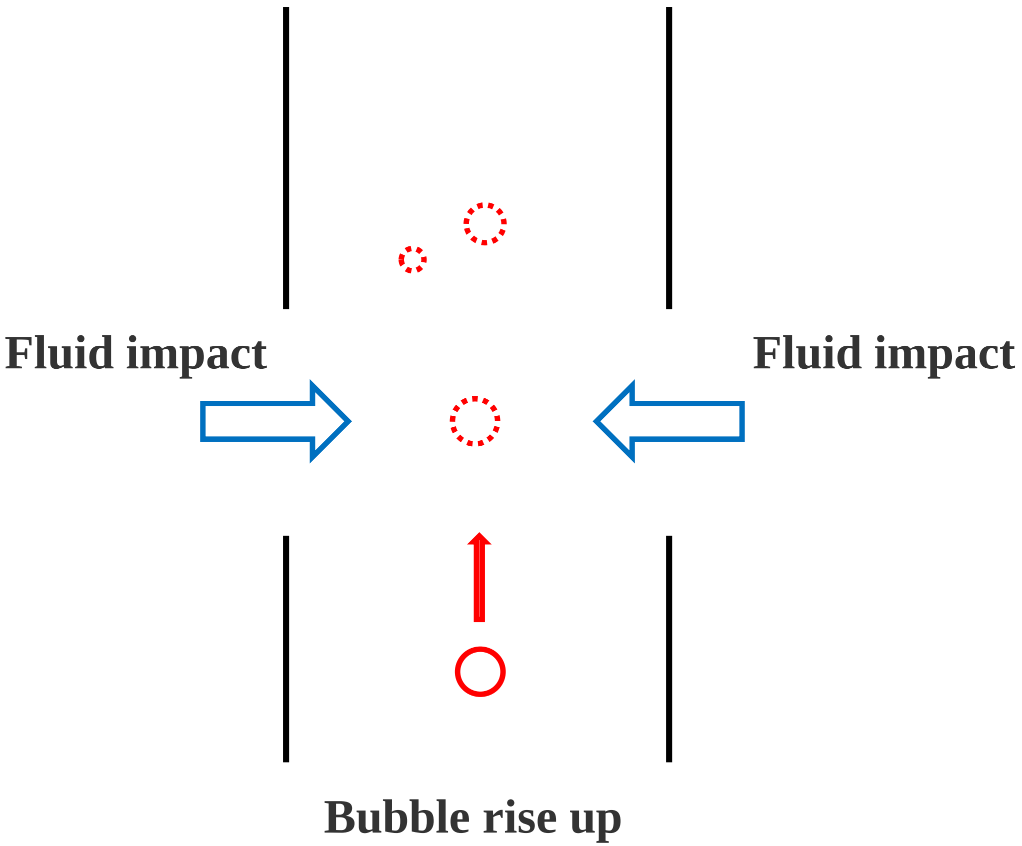


$$\Delta x = r cos\alpha cos\beta$$
$$\Delta y = r cos\alpha sin\beta$$
$$\Delta z = r sin\alpha$$
$$r = 0.6(d_1 + d_2)$$
$$\alpha = random(-\pi, \pi)$$
$$\beta = random(0, 2\pi)$$

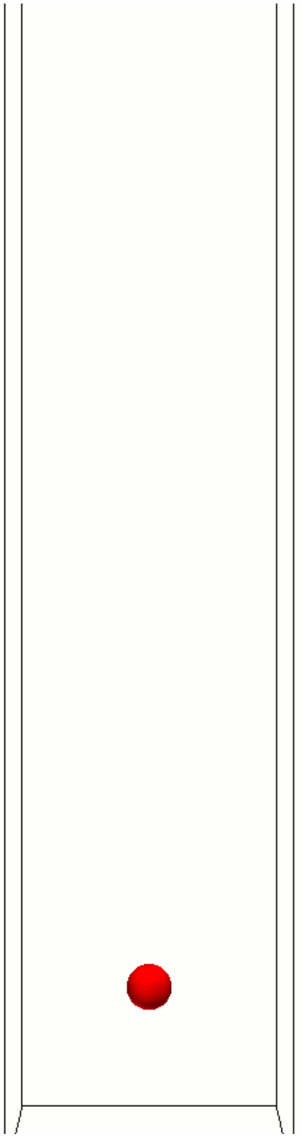
Development of the bubble flow solver

➢ **Bubble breakup:**

Case design：



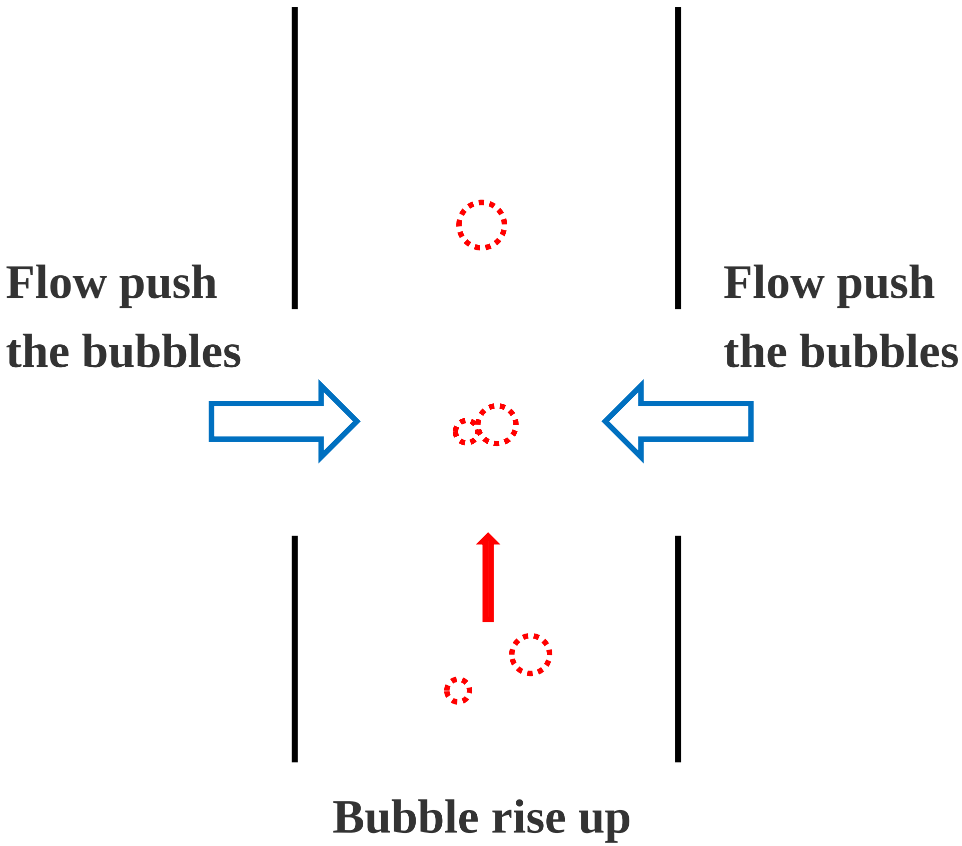
Numerical result：



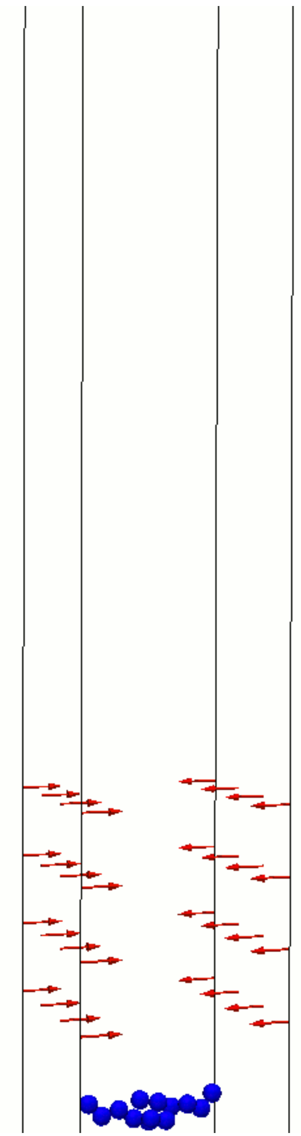
Development of the bubble flow solver

➢ **Bubble coalescence:**

Case design：



Numerical result：



Development of the bubble flow solver

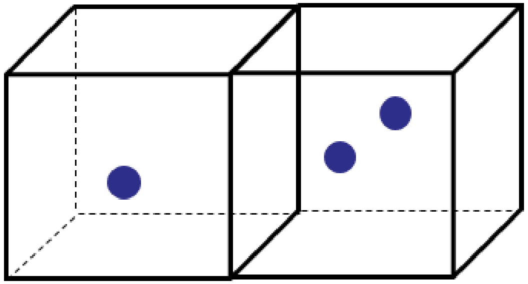
➢ Two-way coupling:

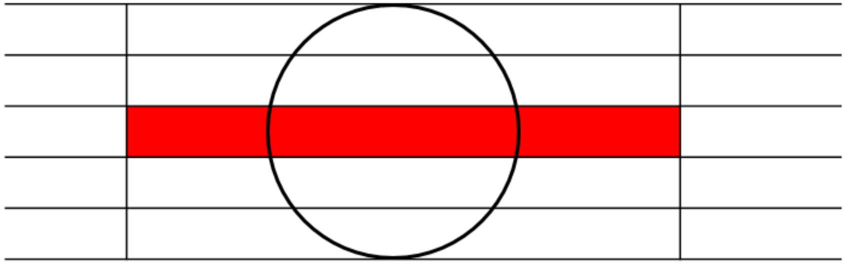
Governing equations for the liquid phase solving:

$$\frac{\partial \alpha_f}{\partial t} + \nabla \cdot (\alpha_f u) = 0$$

$$\frac{\partial \rho_f \alpha_f u}{\partial t} + \nabla \cdot (\rho_f \alpha_f u u) = -\nabla p + \nu \Delta u + \rho_f \alpha_f g - F_{pf}$$

where F_{pf} is the coupled force from bubble to liquid, α_f is liquid volume fraction in cell. The calculation of these two variable is the key problem in two-way coupled algorithm.





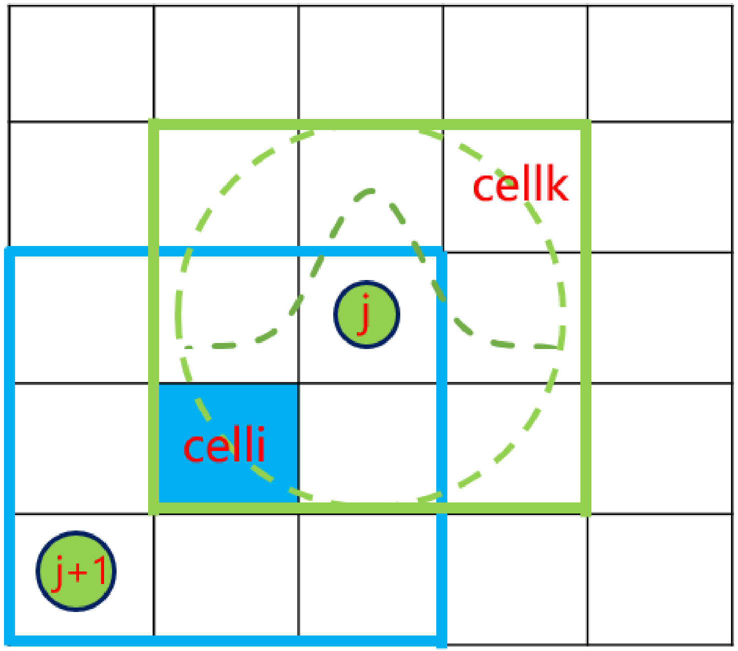
Traditionally, the void fraction was defined in each computational cell as the ratio of the total volume of bubbles in the cell by the cell volume:

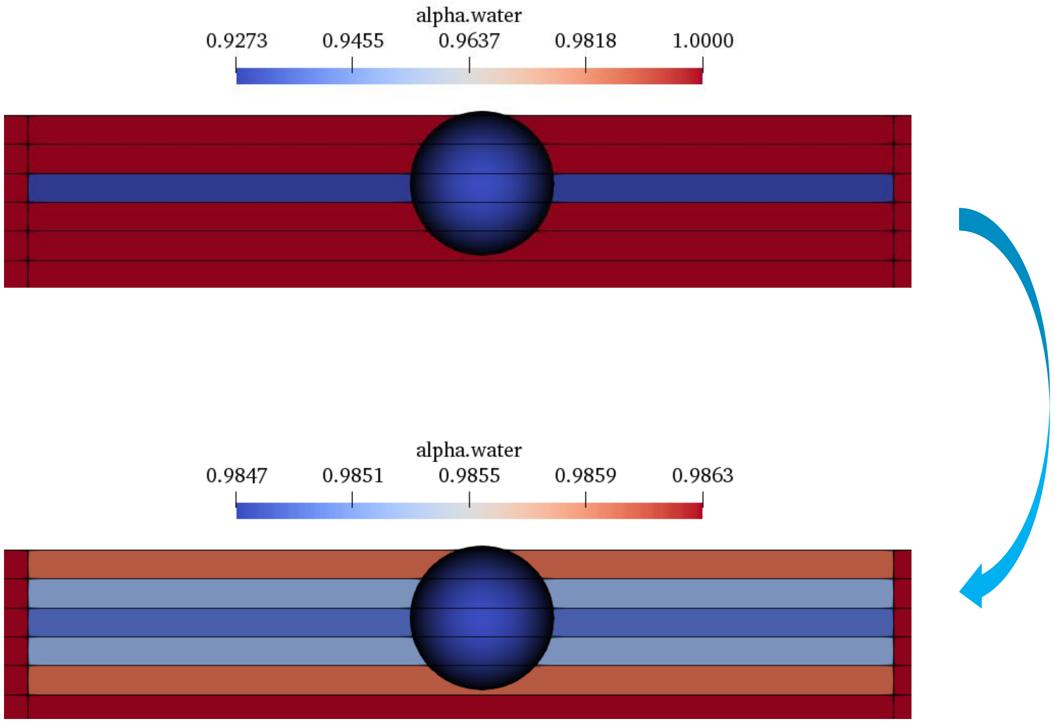
$$\alpha_f = 1 - \frac{\sum_{i=1}^{N} \frac{1}{6}\pi d^3}{\Delta V}$$

However, this algorithm is correct only when the bubble diameter is smaller than the grid size.

Development of the bubble flow solver

- **Two-way coupling:**

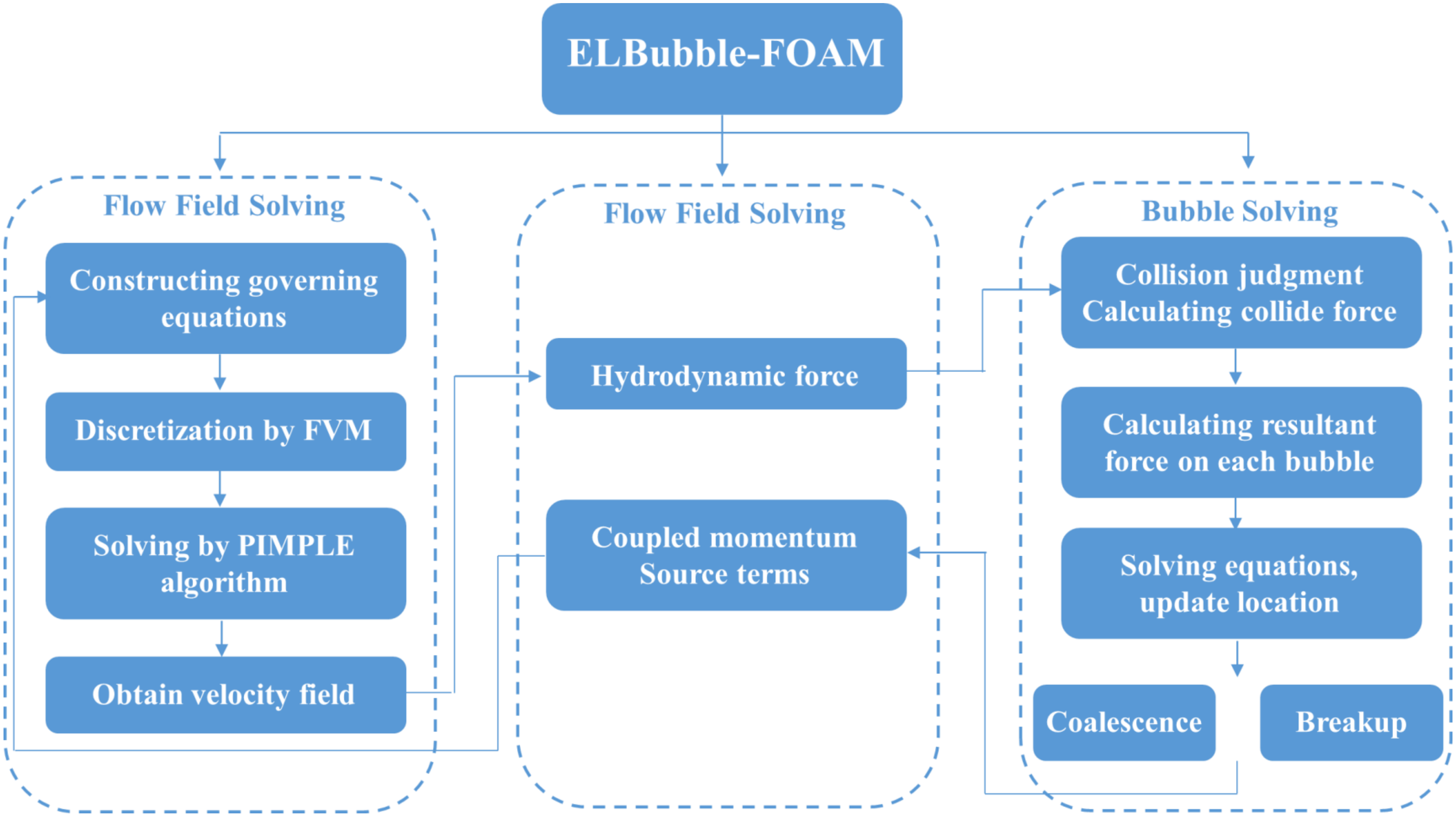




In order to improve the authenticity and stability of the code, a Gaussian bubble volume distribution scheme is embedded in the code.

Development of the bubble flow solver

➢ **Framework of the whole solver:**

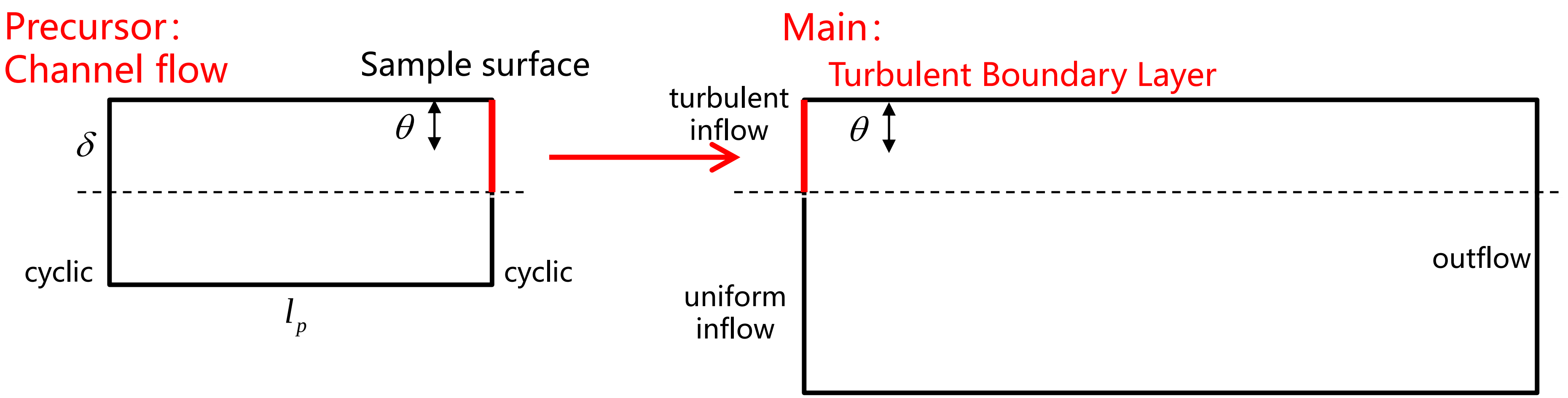


OUTLINE

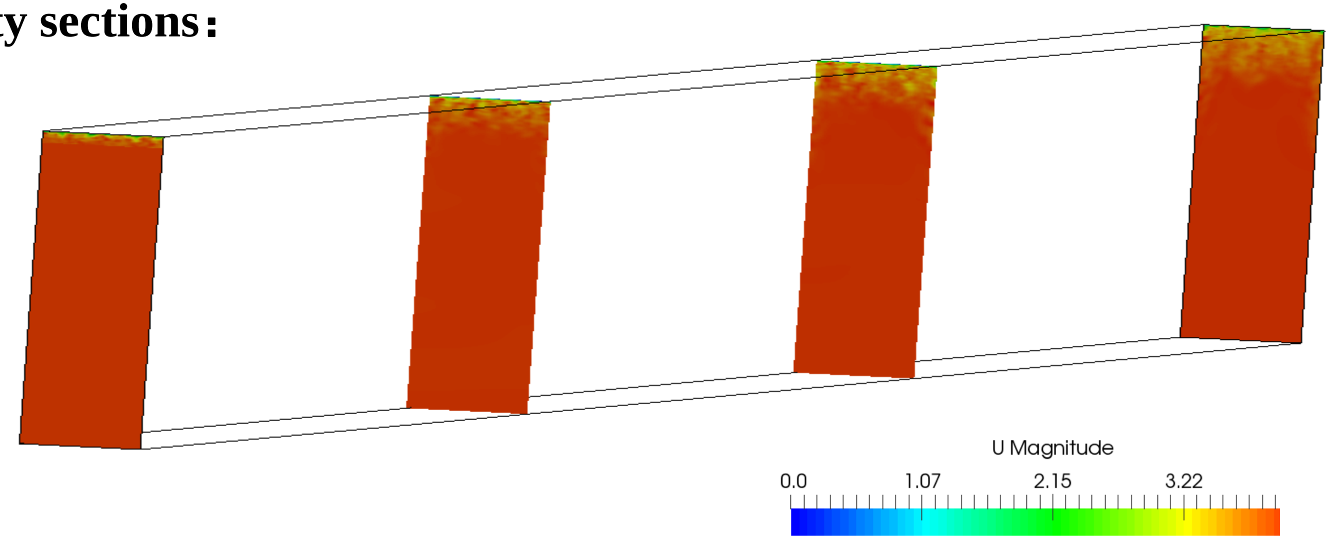
- Background and Motivation
- Bubble Drag Reduction
  - Development of the bubble flow solver
  - **Bubble drag reduction in turbulent boundary layer**
- Air-Layer Drag Reduction
  - Steramwise characteristics
  - Air layer in a cavity
- Conclusion and Future works

BDR in turbulent boundary layer

➢ **Turbulent boundary layer generation:**

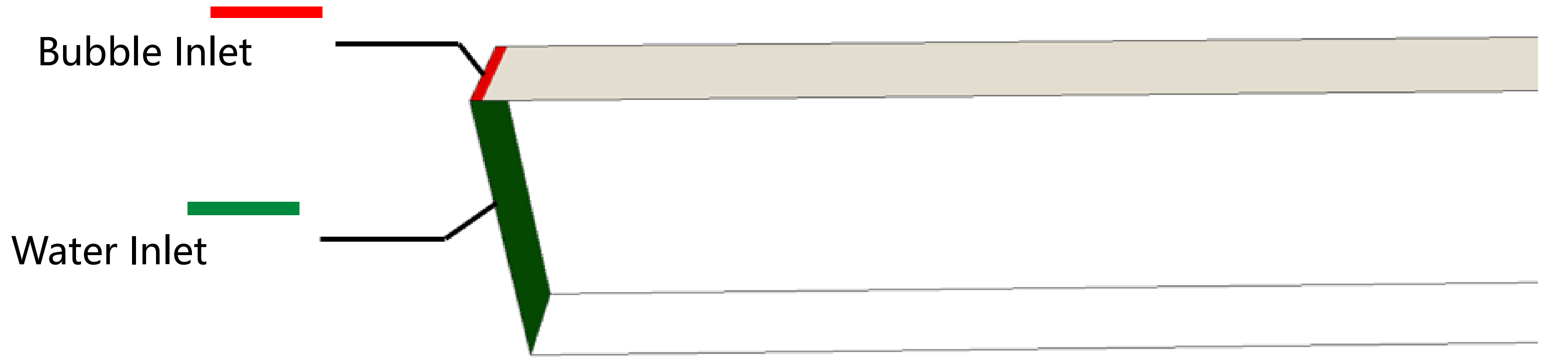


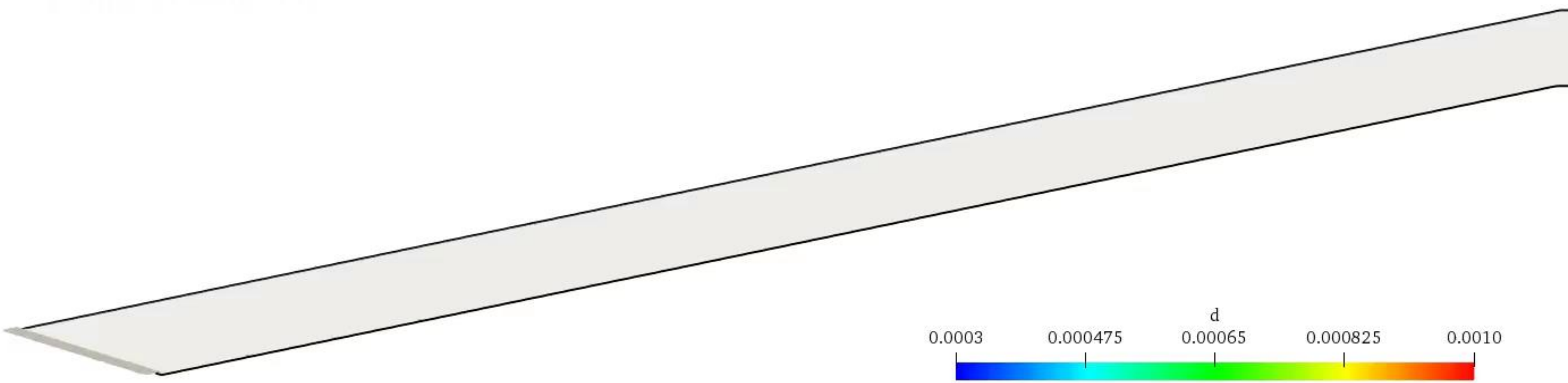
Velocity sections:



BDR in turbulent boundary layer

- **Bubble injection:**

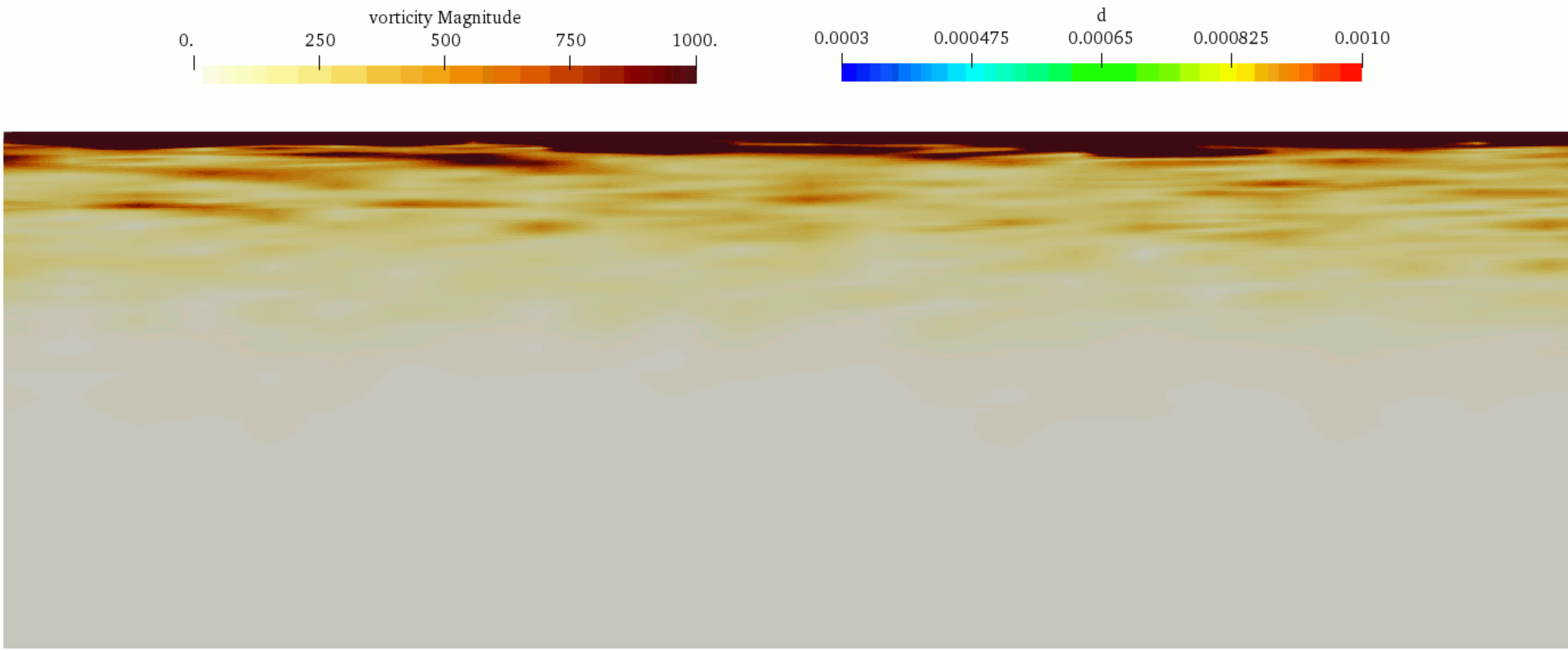




BDR in turbulent boundary layer

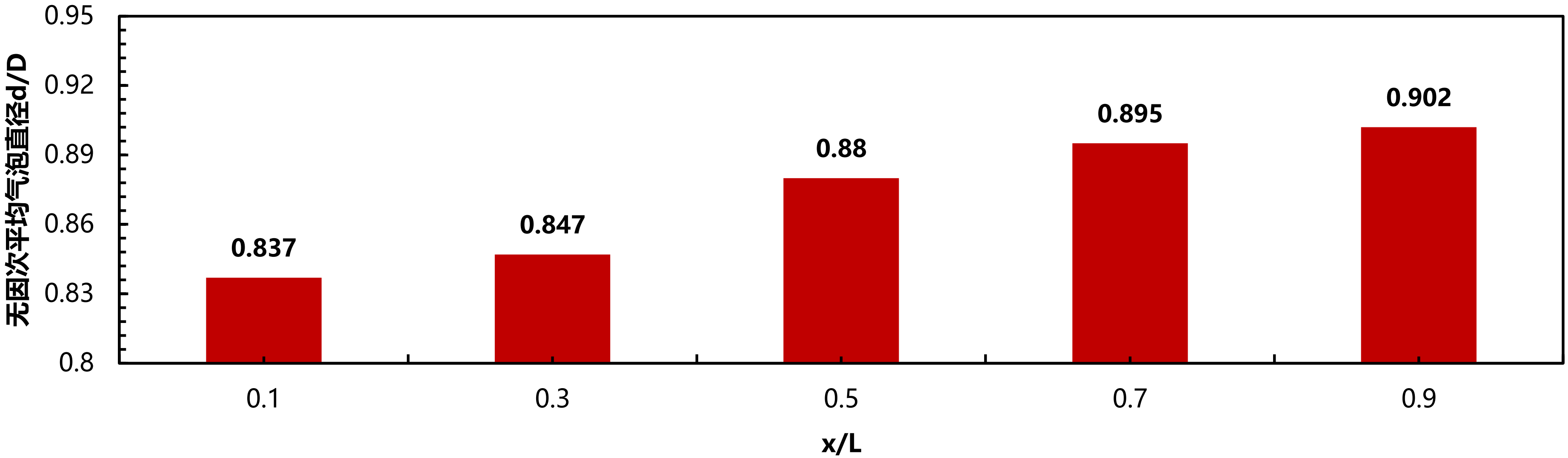
➢ Bubble injection:

Bubbles are all 1mm when injected into flow field. Under the action of turbulence, bubbles rotate, oscillate, breakup and coalesce to form a new size distribution.



BDR in turbulent boundary layer

➢ **Analysis of bubble size distribution :**



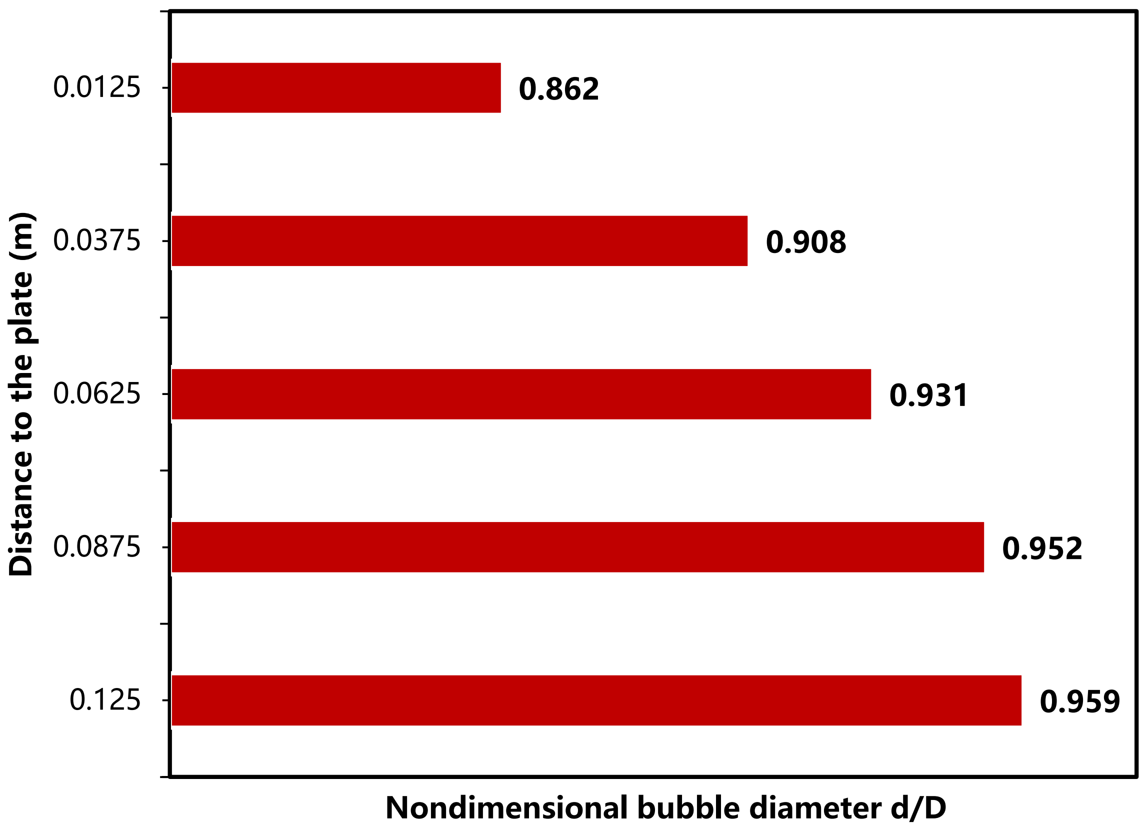
Five ranges are taken to calculate the bubble diameters along the stream-wise direction. In the region close to the injector, breakup is dominant and the average bubble diameter is smaller. Besides, coalescence is more frequent in the downstream, so the average bubble diameter increases along the downstream.

BDR in turbulent boundary layer

➢ **Analysis of bubble distribution:**

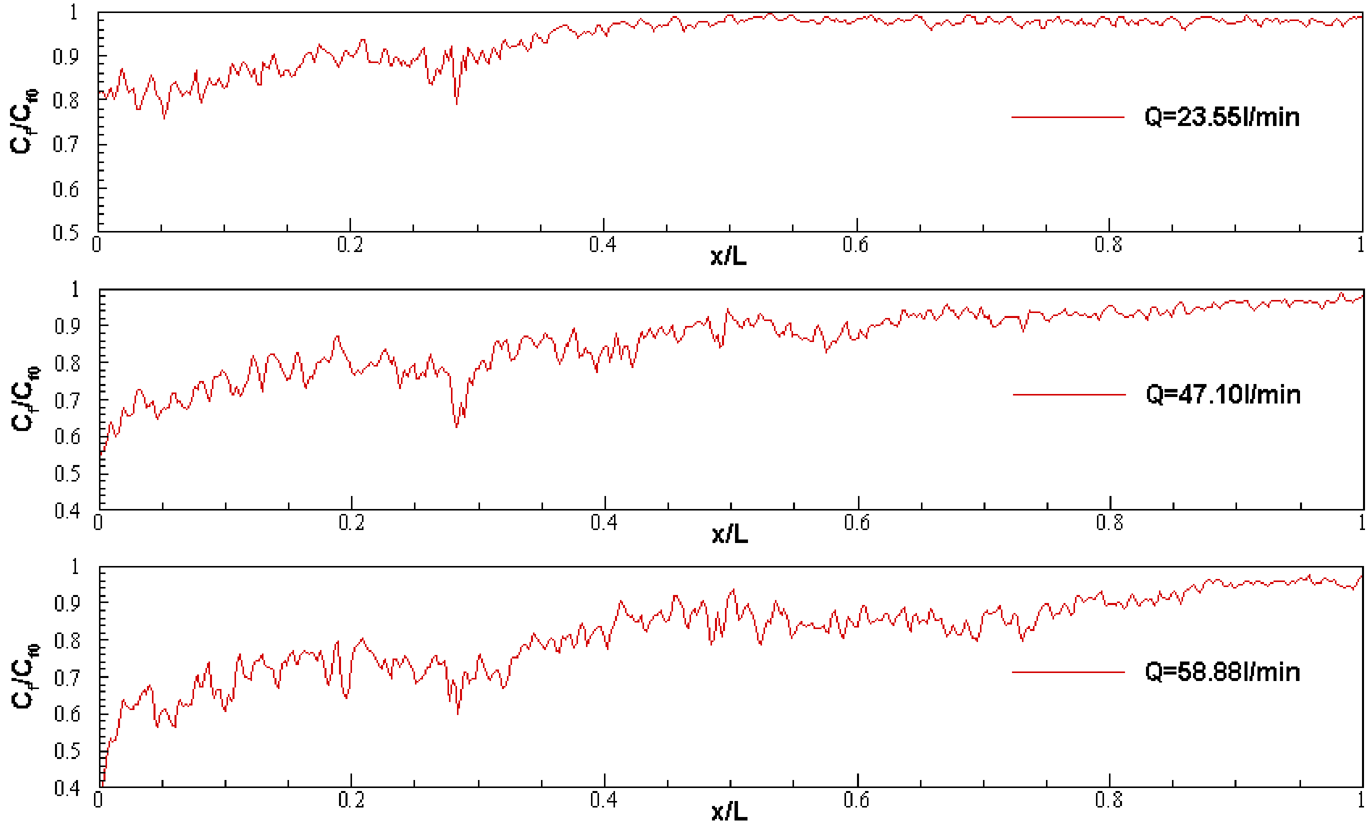
The distribution of bubble diameter in the wall-normal direction is analyzed

The closer to the wall, the smaller the average bubble diameter, which indicate that smaller bubbles are more likely to enter the interior of the turbulent boundary layer.



BDR in turbulent boundary layer

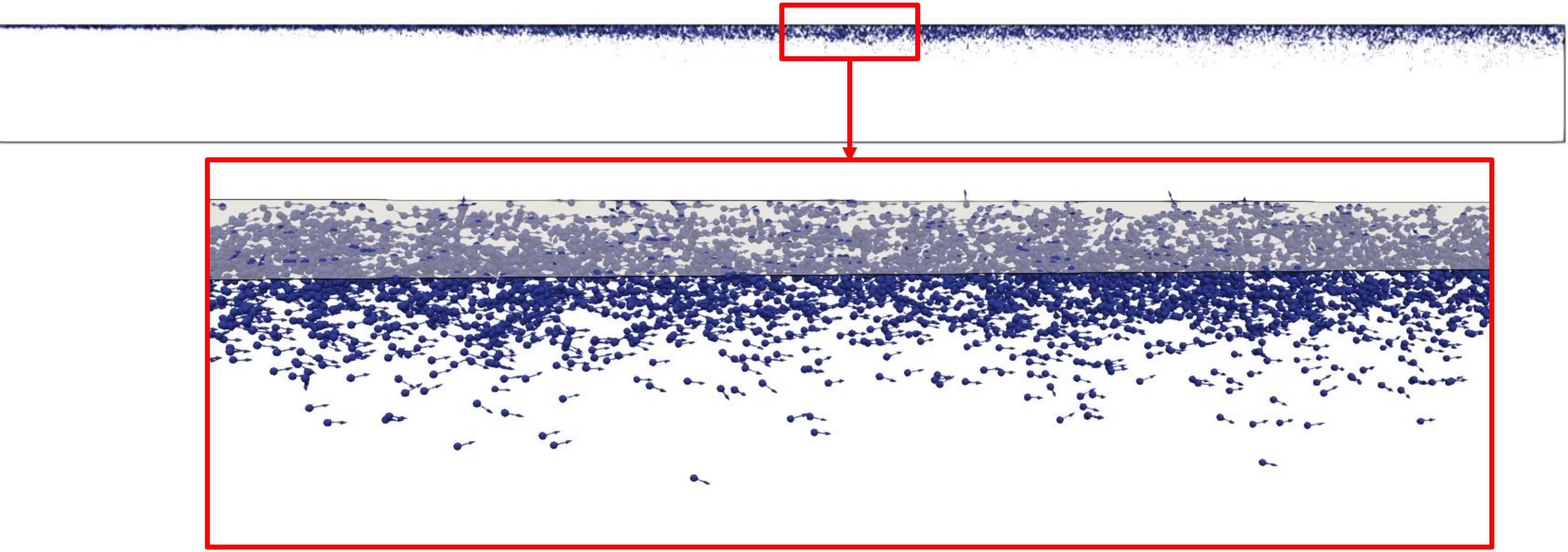
➢ **Drag reduction along the flow direction:**



Best DR effects are obtained near the injector and decreases continuously downstream.

BDR in turbulent boundary layer

➢ **Bubble migration away from the plate:**

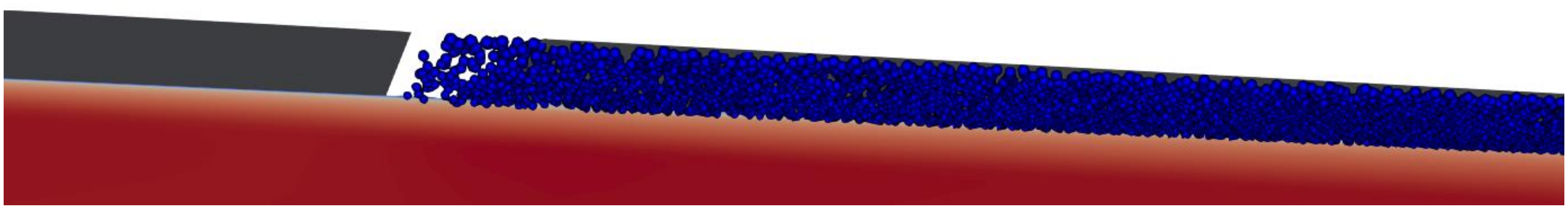


Bubbles have a significant velocity component in the wall-normal direction, which drives the bubbles away from the plate.

BDR in turbulent boundary layer

➢ The influence of turbulence:

In order to verify whether turbulence is the deterministic condition of the bubble migration, a laminar boundary layer simulation with bubble injection is carried out.



In laminar flow, bubbles quickly attach to the plate and move forward in a state of balance between buoyancy and elastic forces in the wall-normal direction

BDR in turbulent boundary layer

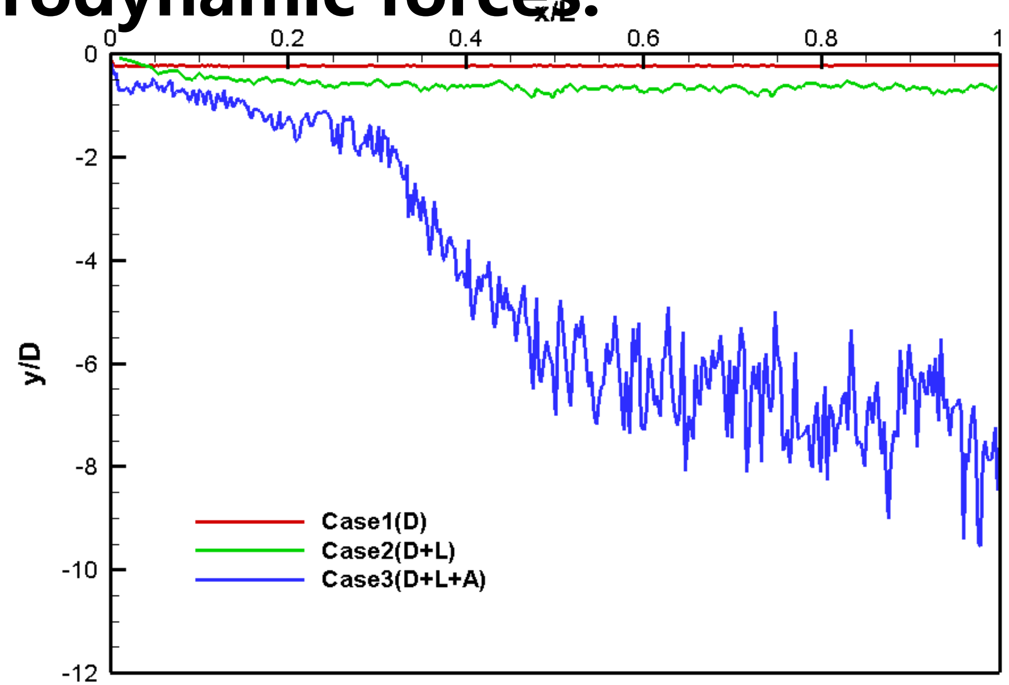
➢ Effect of different hydrodynamic forces:

Three cases are set:

Case1: Only drag

Case2: Drag + Lift

Case2: Drag + Lift +
Fluid acceleration force



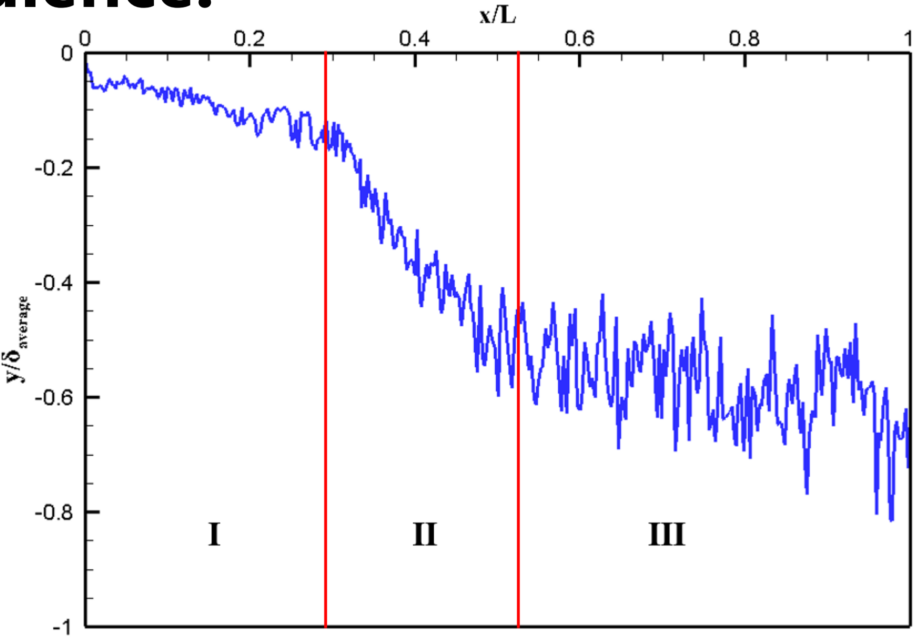
Averaged bubble trajectories with different hydrodynamic forces

- **Drag does not cause the bubbles to migrate away from the plate;**
- **Lift pushes bubbles away from the plate, but not obviously.**
- **Fluid acceleration force is the dominant factor.**

BDR in turbulent boundary layer

➢ **The influence of turbulence:**

The averaged bubble trajectory considering all liquid forces is isolated for further analysis. The bubble movement can be clearly divided into three stages:



- **Stage Ⅰ: Bubbles are in the inner layer of TBL; Samll velocity; High void fraction and excellent drag reduction effect.**
- **Stage Ⅱ: Transition stage; Migrate obviously.**
- **Stage Ⅲ: Bubbles oscillate in the out layer of TBL; Poor drag reduction effect.**

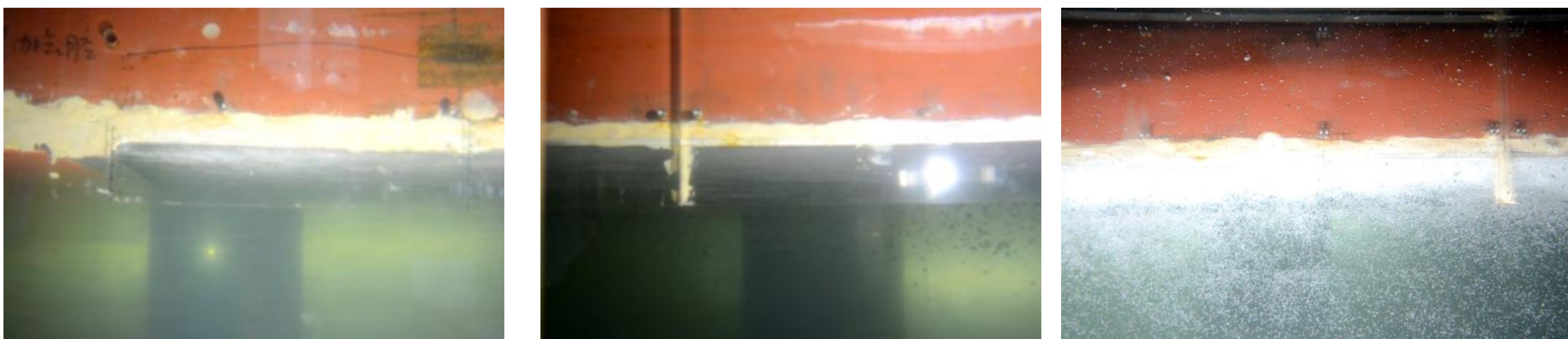
OUTLINE

- Background and Motivation
- Bubble Drag Reduction
  - Development of the bubble flow solver
  - Bubble drag reduction in turbulent boundary layer
- Air-Layer Drag Reduction
  - **Steramwise characteristics**
  - Air layer in a cavity
- Conclusion and Future works

Air-Layer Drag Reduction

➢ **Typical test phenomenon:**

Picture of flow condition on the plate (a) Bubble；(b) Transition；(c) Air layer



Side view of unsteady air layer

Air-Layer Drag Reduction

➤ Numerical method:

We try to use VOF method to model the air-layer two phase flow.

The solver interFoam in OpenFOAM is adopted.

Artificial compressive term in α-equation is used for interface sharpening.

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha U) + \nabla \cdot (\alpha(1-\alpha)c|U|\frac{\nabla \alpha}{|\nabla \alpha|}) = 0$$

$$\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U U) - \nabla \cdot \tau = -\nabla p_{rgh} - gh\nabla p + \sigma k \nabla \alpha$$

$$\nabla \cdot U = 0$$

Air-Layer Drag Reduction

➢ LES model:

Eddy in the flow field is filtered according to the scale. The large-scale vortex structure is directly solved, while the small-scale one is approximated by sub-grid model. The filtered governing equations:

$$\nabla \cdot \overline{U} = 0$$

$$\frac{\partial \rho \overline{U}}{\partial t} + \nabla \cdot (\rho \overline{U}\,\overline{U}) = -\nabla \overline{p_{rgh}} - gh\nabla \overline{p} + \nabla \cdot (\overline{\sigma_{ij}} + \overline{\tau_{ij}}) + \sigma k \nabla \overline{\alpha}$$

The sub-grid scale stress tensor is required to close.

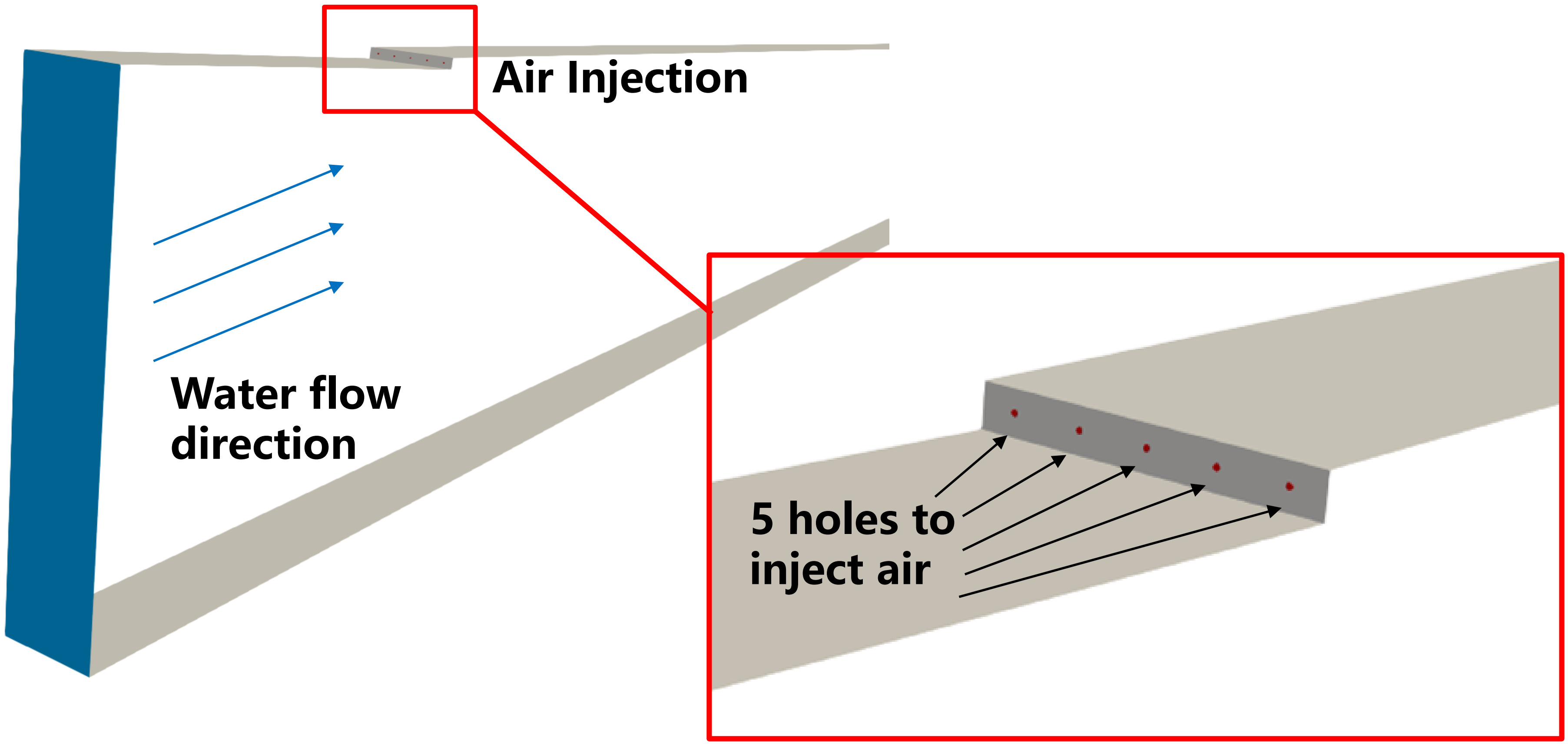
$$\tau_{ij} = \overline{u_i u_j} - \overline{u_i}\,\overline{u_j}$$

SGS models: Smagorinsky model; dynamic Smagorinsky model; WALE model; kEqn model…

Air-Layer Drag Reduction

➢ Computational domain:

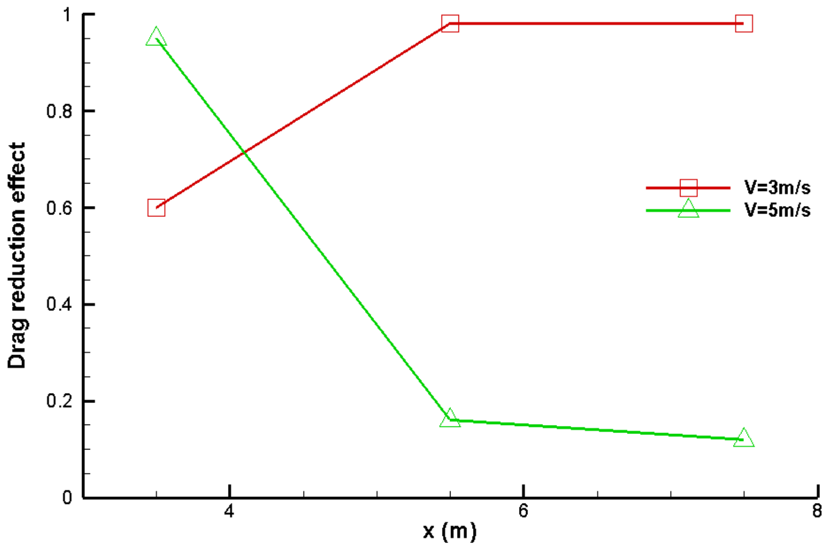
Computational domain is assigned over the whole length of flat plate in experiment. Width across 5 holes was chosen on the span direction.



Air-Layer Drag Reduction

➢ Conditions:

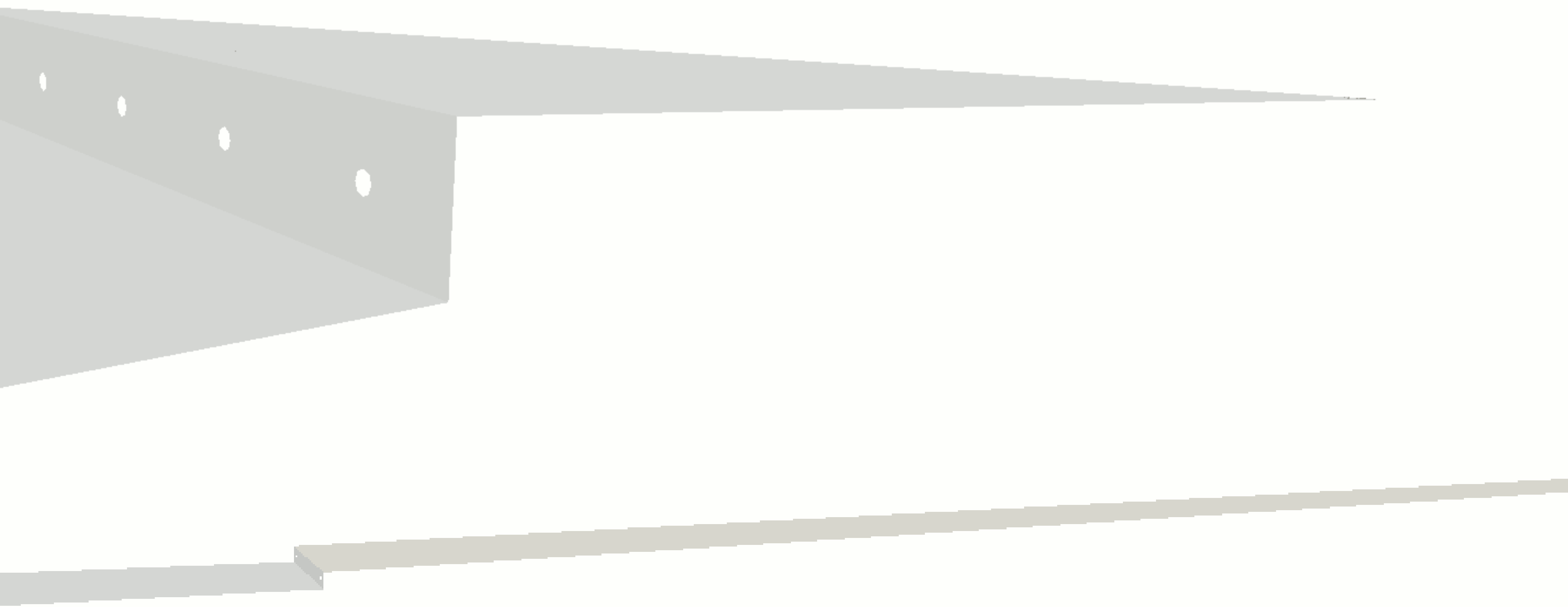
Conditions are chosen that all three flow states are included. The key problem is to predict the length of stable air layer accurately.



The change of the flow state along the downstream direction

Air-Layer Drag Reduction

➢ **LES results:**

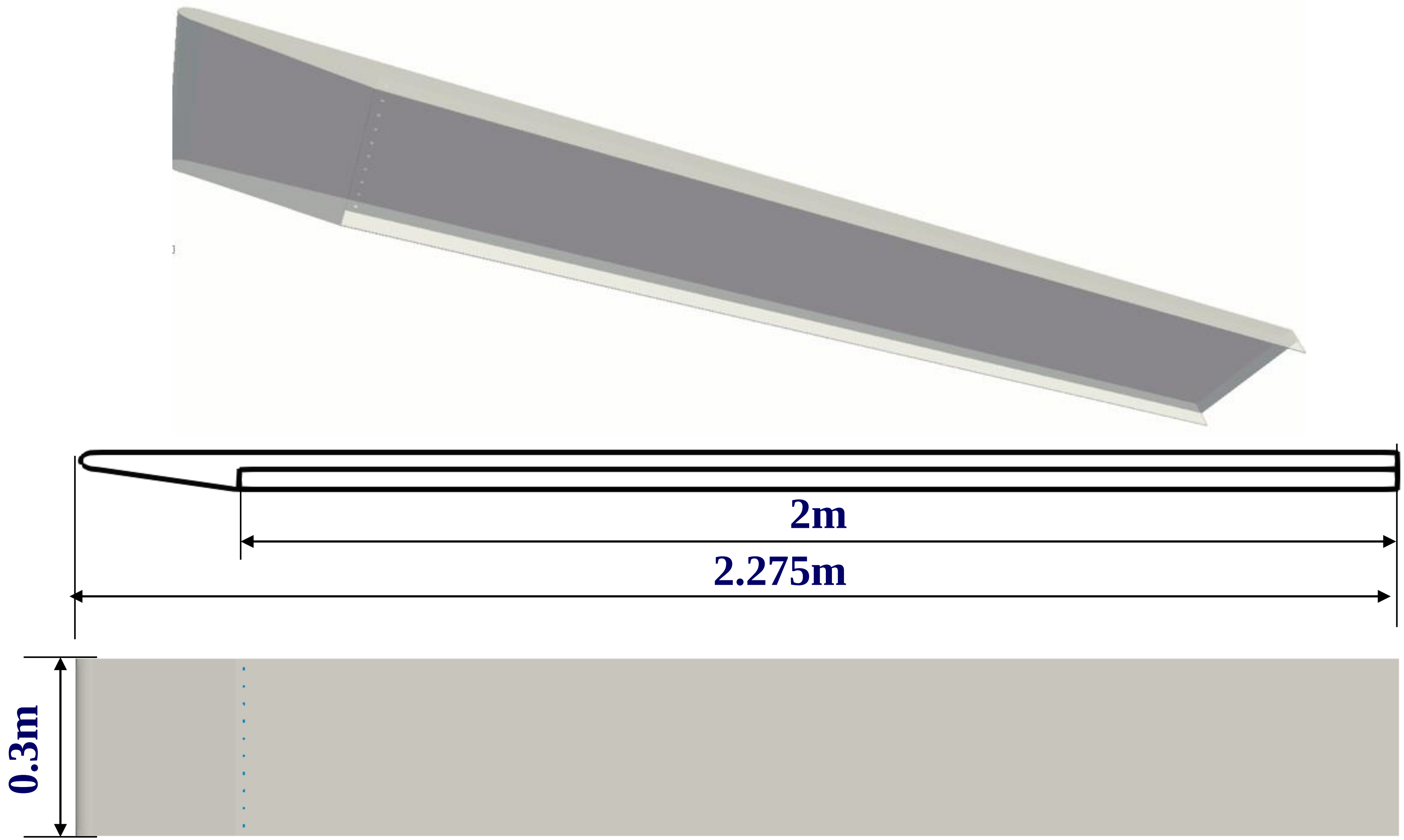


OUTLINE

- Background and Motivation
- Bubble Drag Reduction
  - Development of the bubble flow solver
  - Bubble drag reduction in turbulent boundary layer
- Air-Layer Drag Reduction
  - Steramwise characteristics
  - **Air layer in a cavity**
- Conclusion and Future works

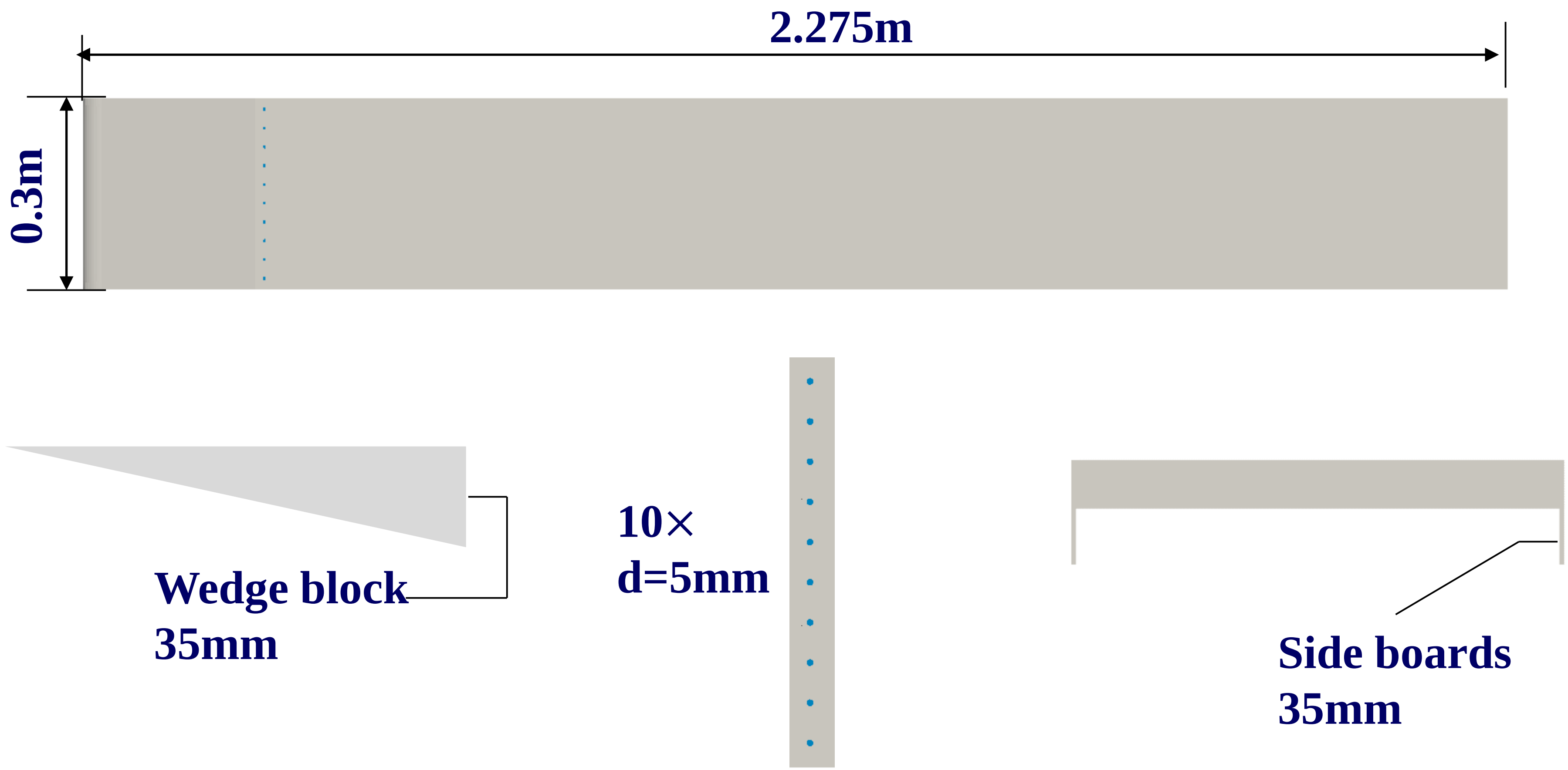
Air-Layer Drag Reduction

- **Whole geometry:**



Air-Layer Drag Reduction

➢ **Whole geometry:**

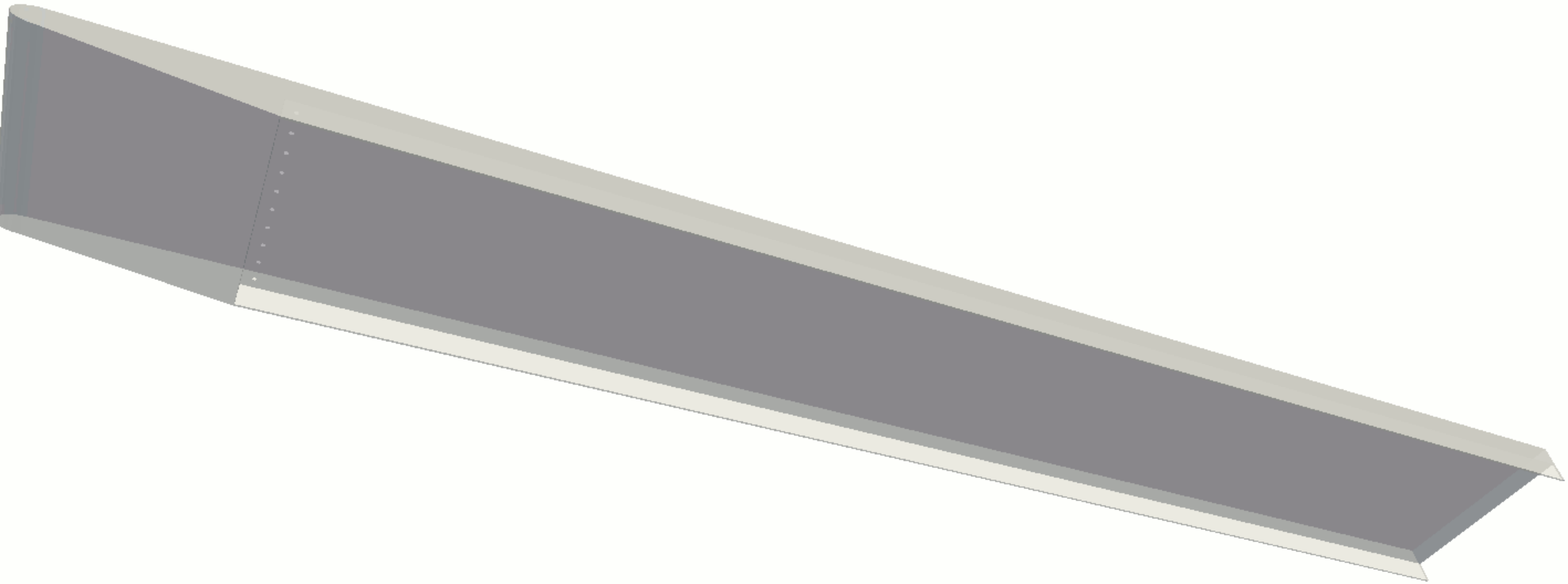


Air-Layer Drag Reduction

- **Air layer in a cavity:**

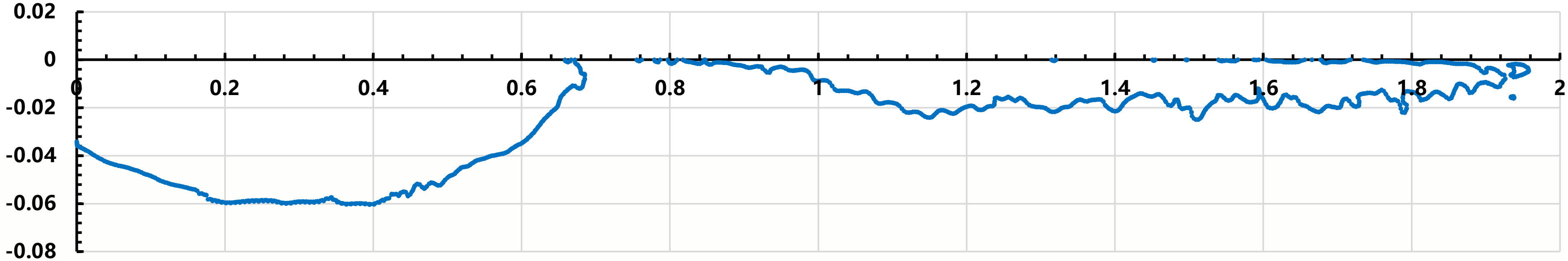
$Q=0.01m^3/s=10L/s$

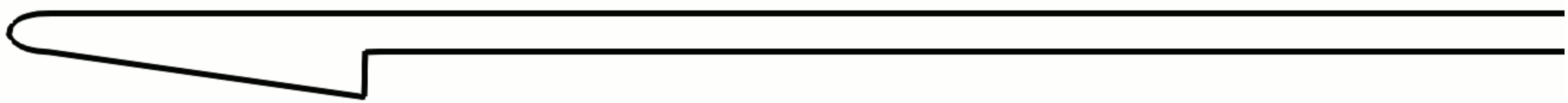
$U=2m/s$



Air-Layer Drag Reduction

➢ **Air layer shape in downstream direction:**



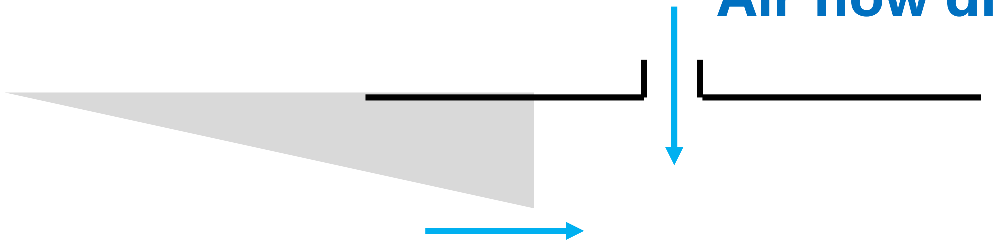


Air-water interface

Streamline

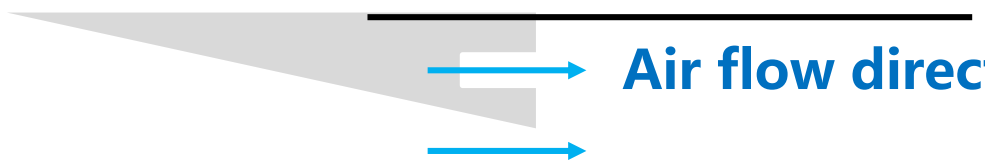
Air-Layer Drag Reduction

➢ **Effect of air injection direction:**



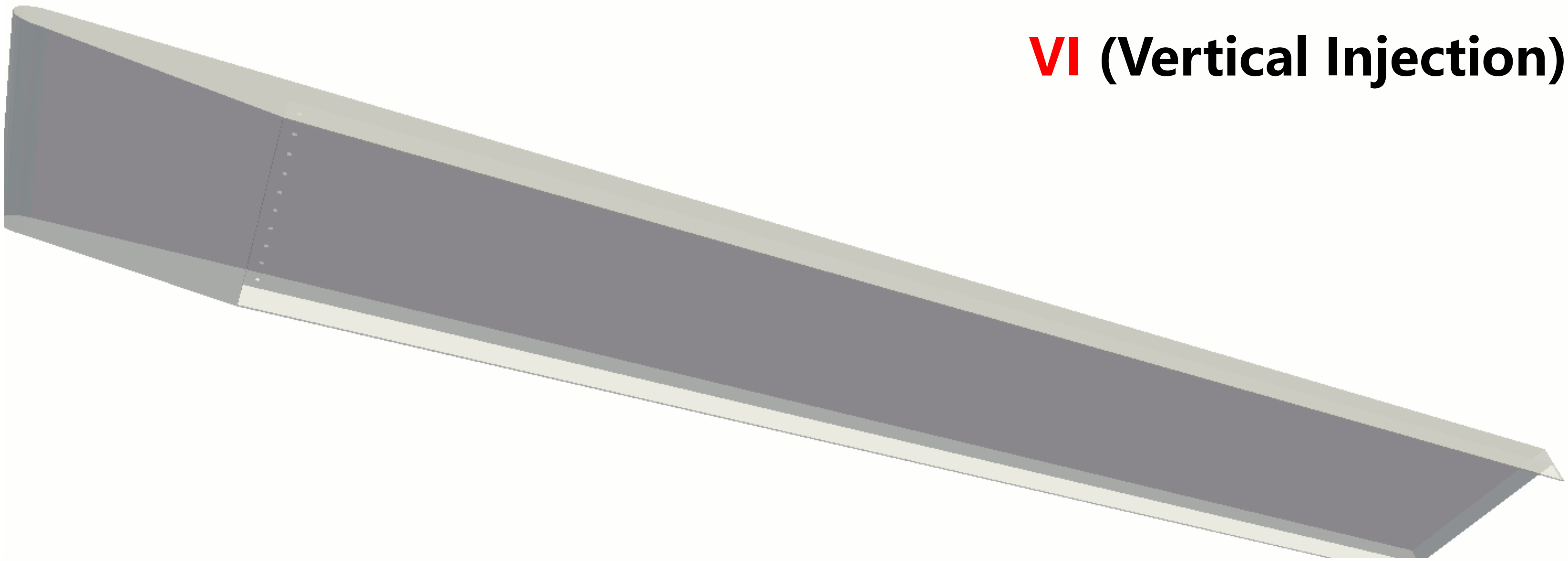
VI (Vertical Injection)

Whether the disordered flow and the large bubble are caused by the different direction of air flow and water flow?

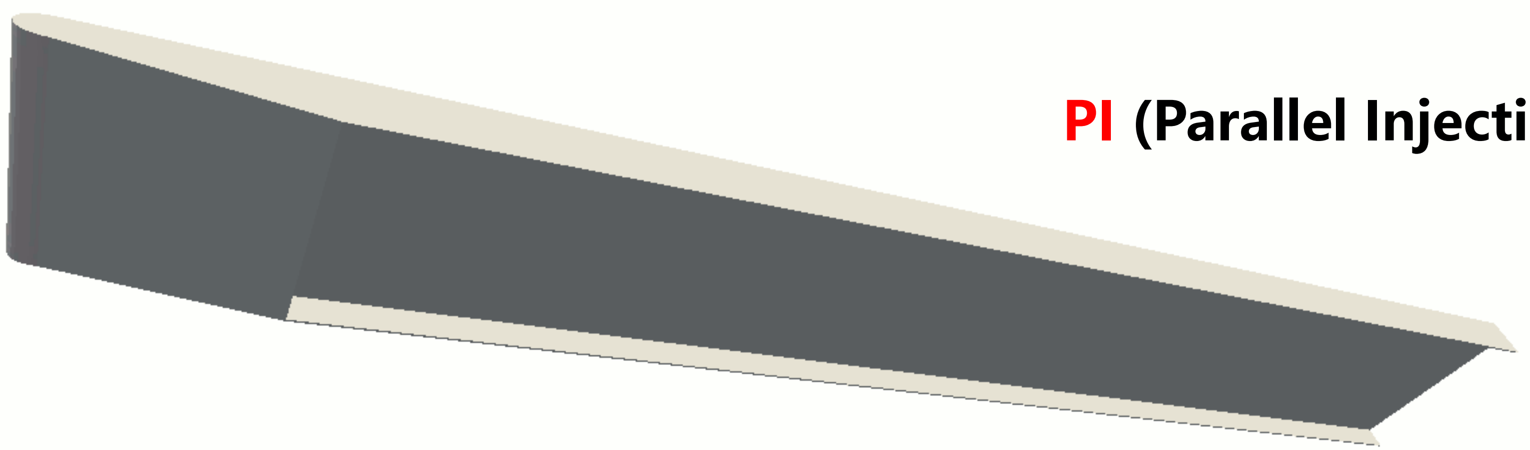


PI (Parallel Injection)

Air-Layer Drag Reduction



VI (Vertical Injection)

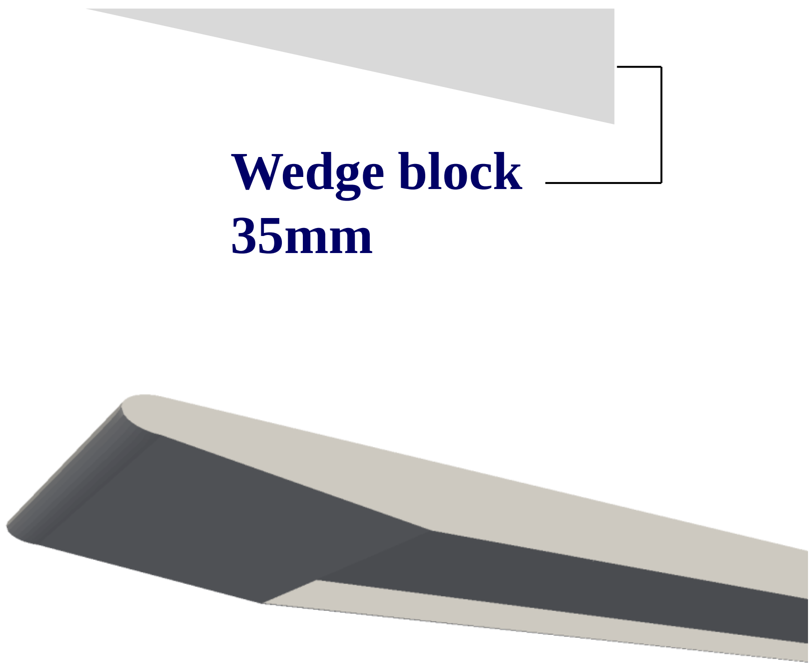


PI (Parallel Injection)

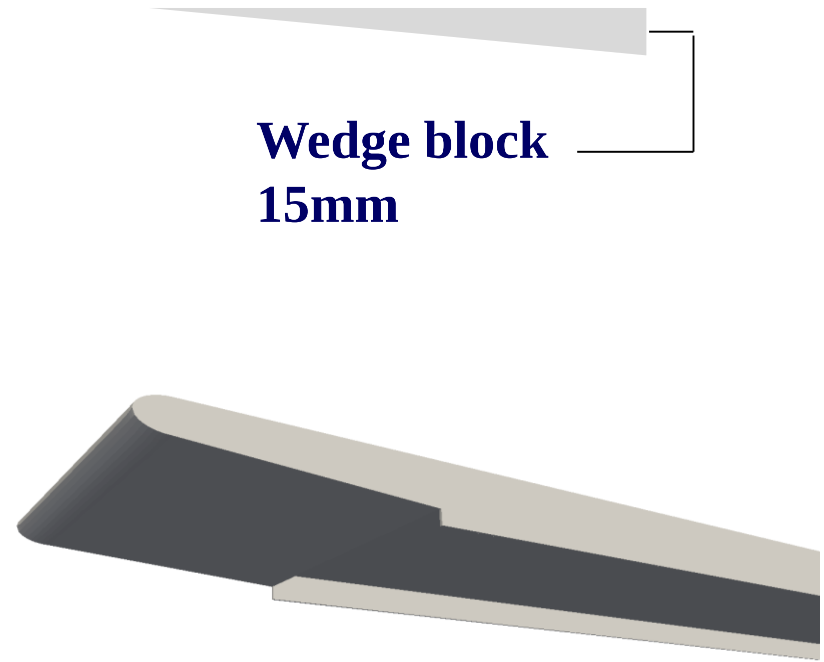
Air-Layer Drag Reduction

- **Effect of wedge block height:**

Original:



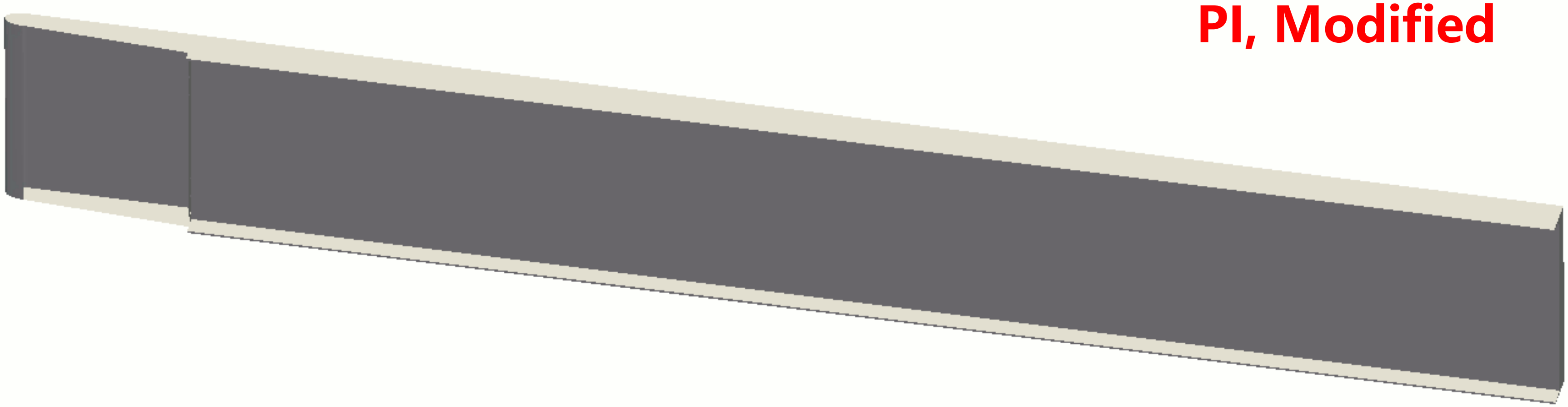
Modified:



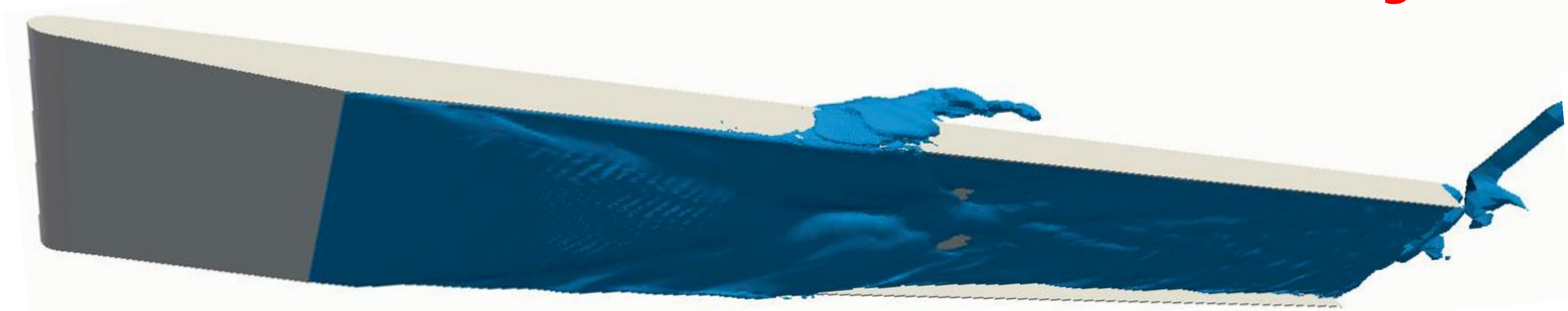
Air-Layer Drag Reduction

➢ **Effect of wedge block height:**

PI, Modified



PI, Original

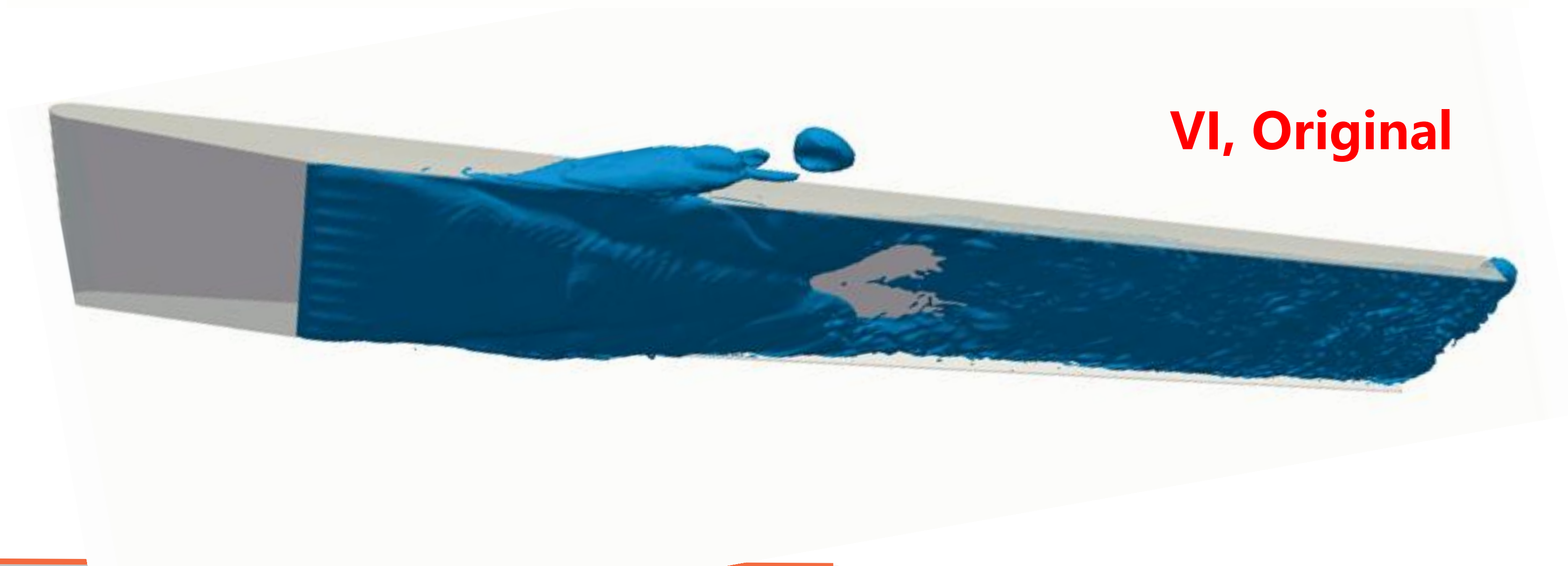


Air-Layer Drag Reduction

- **Effect of wedge block height:**

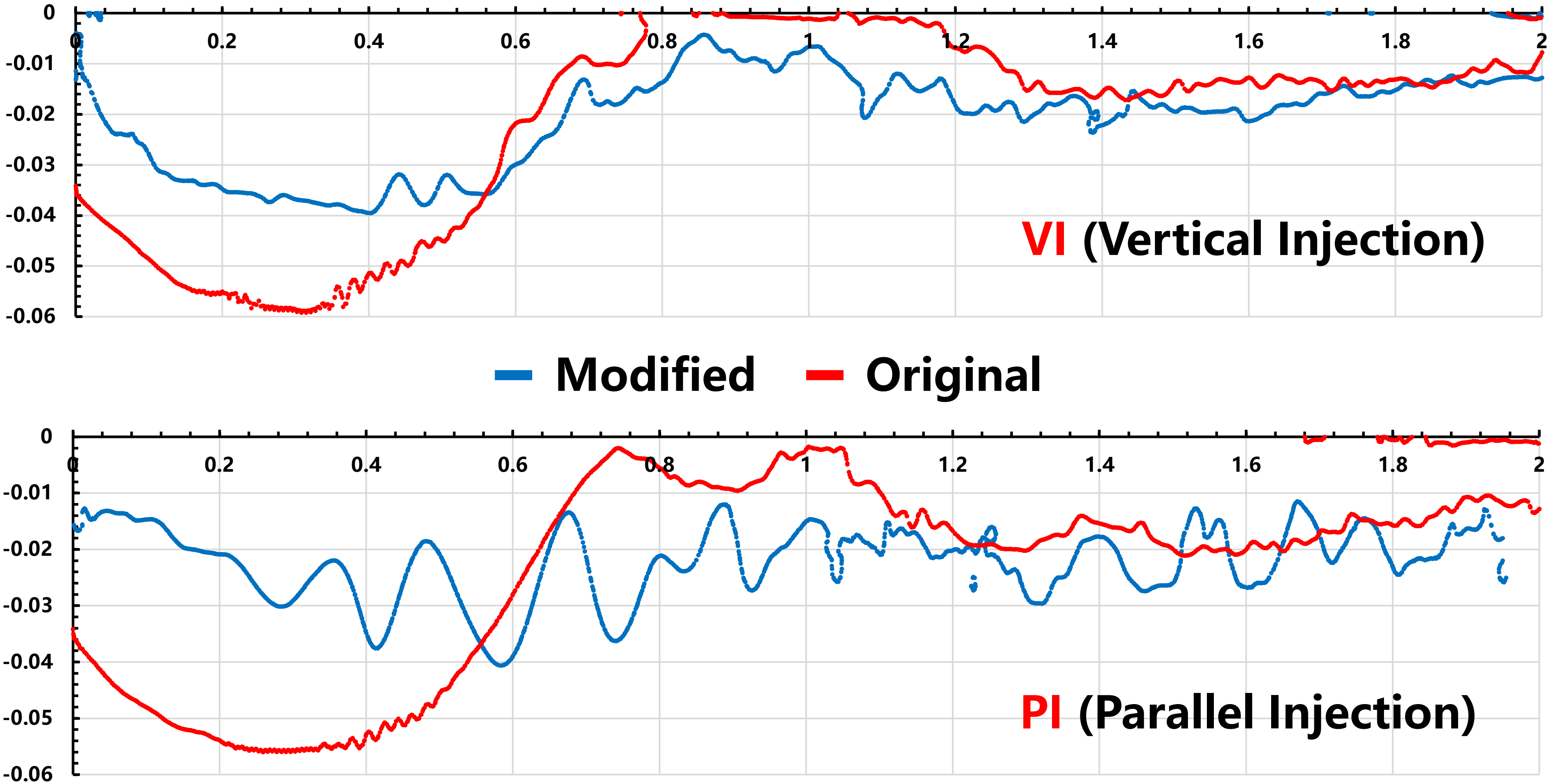
VI, Modified

VI, Original



Air-Layer Drag Reduction

- **Effect of wedge block height:**



OUTLINE

- Background and Motivation
- Bubble Drag Reduction
  - Development of the bubble flow solver
  - Bubble drag reduction in turbulent boundary layer
- Air-Layer Drag Reduction
  - The effect of turbulence modeling
  - Air layer in a cavity
- **Conclusion and Future works**

Conclusion

Bubble Drag Reduction

- ✓ We simulate the bubble drag reduction successfully by using a two-way coupled Euler-Lagrange method. A bubble flow solver is developed that can predict various bubble kinematic behavior such as collision, breakup and coalescence.
- ✓ Bubble drag reduction effect and bubble size distribution in a turbulent boundary layer are predicted well. The bubble migration caused by the acceleration force of turbulent fluid is considered to be the main reason for the failure of bubble drag reduction in the downstream. And Bubble trajectories can be divided into three stages.

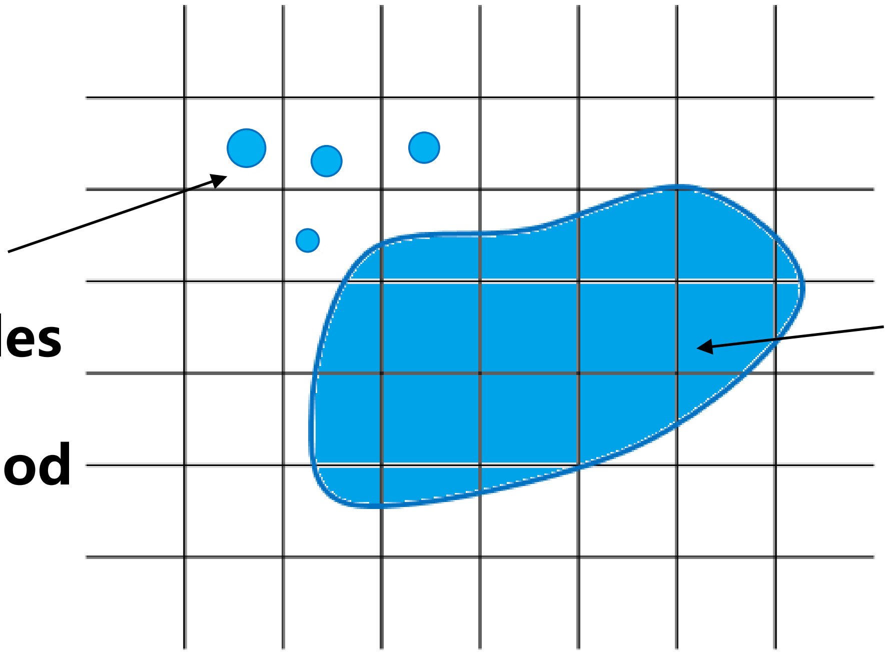
Conclusion

Air Layer Drag Reduction

- ✓ **Turbulence modeling plays an important role in the prediction of air layer evolution. LES model performs better than RANS model in the simulation of an unsteady air layer.**
- ✓ **Air layer drag reduction in a cavity is simulated and the effect of two key parameters is studied. The adoption of parallel injection is a little better to form a complete air layer. While obvious improvement can be found by reducing the height of wedge block.**

Future works

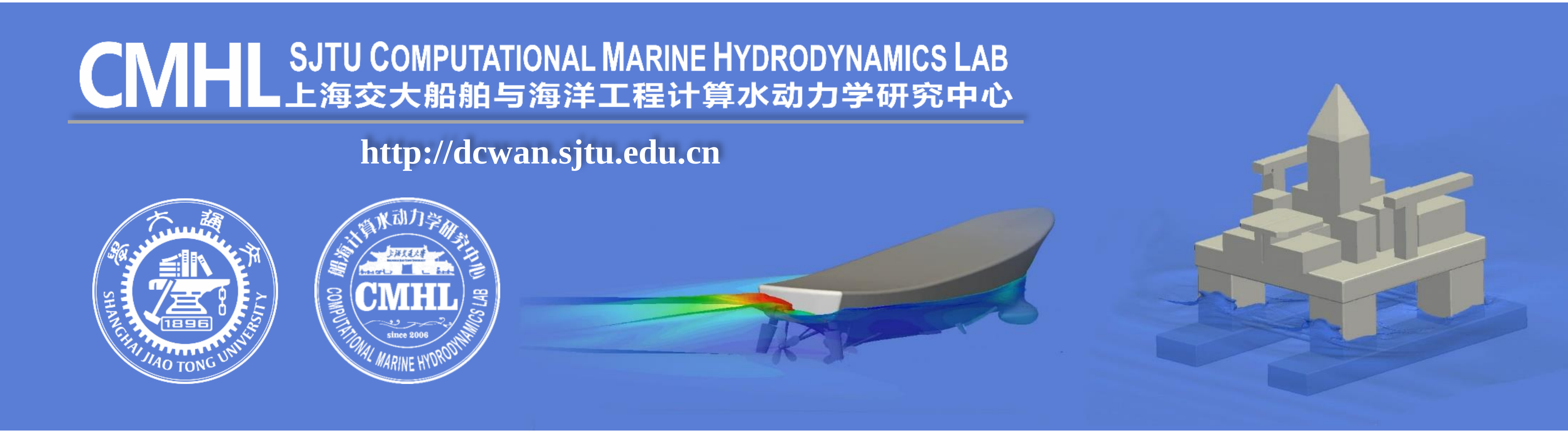
Sub-grid bubbles are tracked by Lagrange method



Large air-water interface is captured by VOF method

Future works will be focused on the development of a muti-scale two phase flow solver.

Thanks!


CMHL
SJTU COMPUTATIONAL MARINE HYDRODYNAMICS LAB
上海交大船舶与海洋工程计算水动力学研究中心
http://dcwan.sjtu.edu.cn
SHANGHAI JIAO TONG UNIVERSITY
1896
CMHL
since 2006
COMPUTATIONAL MARINE HYDRODYNAMICS LAB