Numerical Heat Transfer

Chapter 13 Application examples of fluent for flow and heat transfer problem

Instructor Chen, Li; Tao, Wen-Quan

CFD-NHT-EHT Center
Key Laboratory of Thermo-Fluid Science & Engineering
Xi’an Jiaotong University

Xi’an, 2018-12.-19

数值传热学

第 13 章 求解流动换热问题的Fluent软件应用举例

主讲陈 黎，陶文铨

西安交通大学能源与动力工程学院
热流科学与工程教育部重点实验室

2018年12月19日，西安

13. A3 Multiphase flow using VOF

采用流体体积法研究多相流

Focus: in this example, first the background of multiphase flow is introduced, and then Volume of Fluid method is discussed in detail.

13. A3 Numerical simulation of multiphase flow using volume of fraction(VOF) method

- **Problem description：** The computational domain is a 2D channel. Air with velocity of 5 m/s flows into the channel from the left inlet; and water with velocity of u=0.1m/s enters the channel from a micro-pore at the bottom.

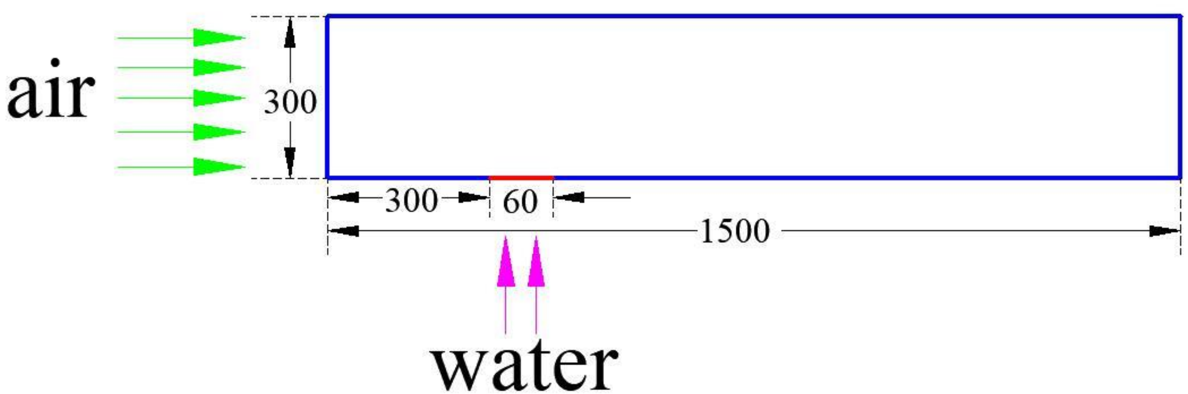
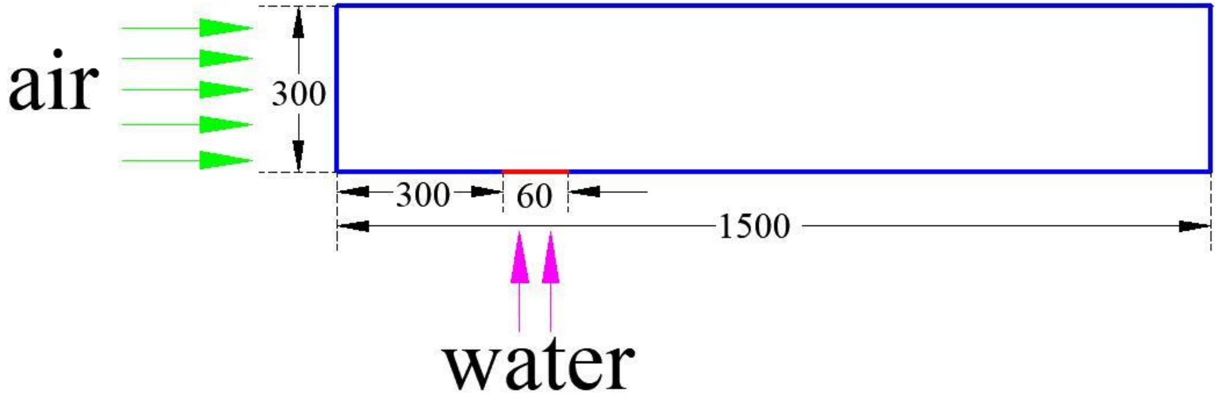


Fig.1 Computational domain and geometry sizes (μm)

- **Find:** water dynamic behavior, pressure drop, and saturation in the channel;

The boundary conditions are as follows:



	Fluid flow
Air inlet	Velocity inlet
Water inlet	Velocity inlet
Outlet	Outflow
Bottom	Wall, 140º
Up	Wall, 60º

1. Background of Multiphase flow

Multiphase fluid flows are widely encountered in natural, scientific and engineering systems

A phase refers to gas, liquid or solid state of matter. A multiphase flow is the flow of a mixture of phases, such as gas (bubbles) in a liquid, or liquid (droplets) in a gas, and so on.

Same component （单组分多相流)

Liquid water and water vapor system

H_2O

Multiple components （多组分多相流）

Liquid water and air system, H_2O, N_2, O_2…

Crown

2. Fundamental definitions

Surface tension: refers to the tensile force exists at the phase interface separating two fluids, due to a mutual attraction between molecules near the interface

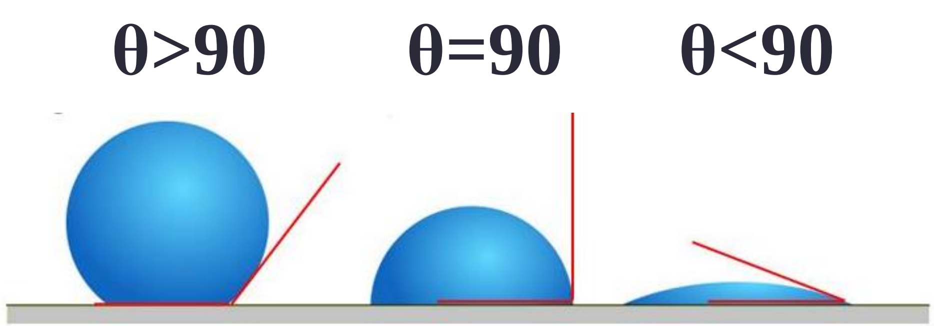
unit: N/m Typical value: water-air: 0.0725 N/m

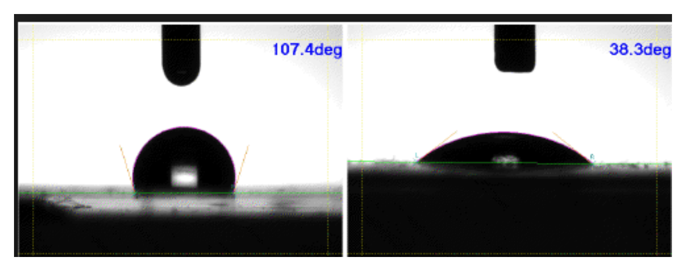
Water striders stay on top of water (水面上的水黾)

Lotus effect (荷叶效应)

Contact angle

measurement of the surface wettability. The angle of the triple-phase line. Hydrophilic surface（亲水）with angle less than 90, liquid tends to spread. Hydrophobic surface（疏水）with angle higher than 90, liquid tends to form droplet. Neutral surface with angle as 90.

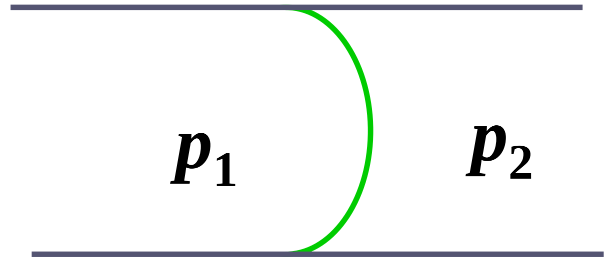




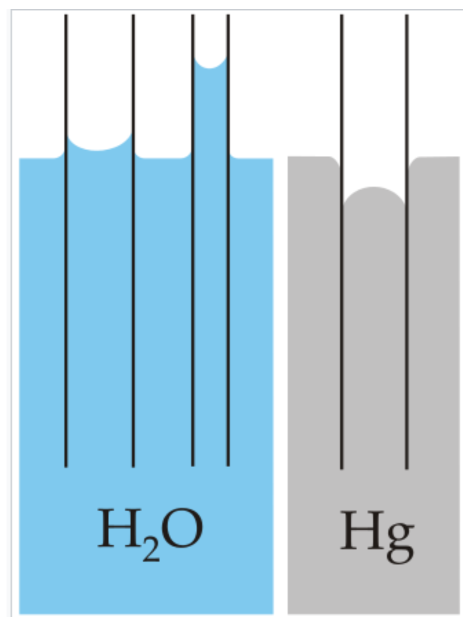
Capillary pressure

pressure difference across a phase interface, related to the surface tension force

$$P_c = P_1 - P_2 = \frac{\sigma\cos\theta}{r}$$



Because of the capillary pressure, a liquid can flow in narrow spaces without the assistance of, or even in opposition to, external forces like gravity.



$$h = \frac{2\gamma\cos\theta}{\rho g r}$$

mercury

2. Different methods for multiphase flow

Macroscopic

Volume of Fluid (VOF) 流体体积法

Level Set (LS) 水平集法

VOSET by NHT group

Phase-field 相场方法

Front tracking 前沿跟踪方法

Mesoscopic

Lattice Boltzmann Method, Smooth Particle Hydrodynamics

（格子Boltzmann 方法， 光滑粒子方法）

Microscopic

Molecular dynamics （分子动力学）

3. Volume of Fluid (VOF)

Proposed by Hirt and Nichols in 1981.

JOURNAL OF COMPUTATIONAL PHYSICS 39, 201–225 (1981)

Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*

Volume of fluid (VOF) method for the dynamics of free boundaries
CW Hirt, BD Nichols - Journal of computational physics, 1981 - Elsevier
Several methods have been previously used to approximate free boundaries in finite-difference numerical simulations. A simple, but powerful, method is described that is based on the concept of a fractional volume of fluid (VOF). This method is shown to be more flexible ...

Several methods have been previously used to approximate free boundaries in finite-difference numerical simulations. A simple, but powerful, method is described that is based on

In fact, VOF is one of the most popular methods for multiphase flow. It has been successfully adopted for a wide range of problems, and is till being improved and enhanced.

Volume of fraction（体积分数): the basic variable in VOF

The volume fraction of each fluid in a computational cell

$$C_m = \frac{V_m}{V_{\mathrm{cell}}} \qquad \sum C_m = 1$$

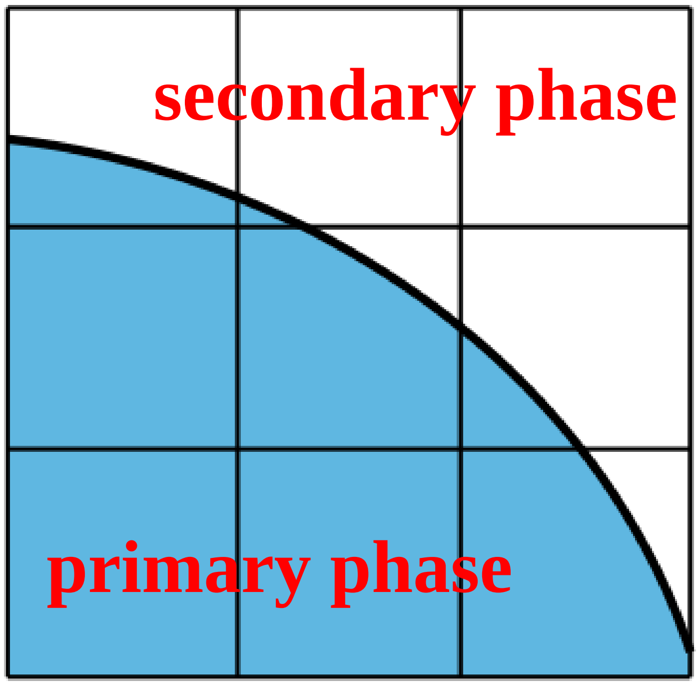
For two-phase flow: primary phase(主相) and secondary phase (次相)。

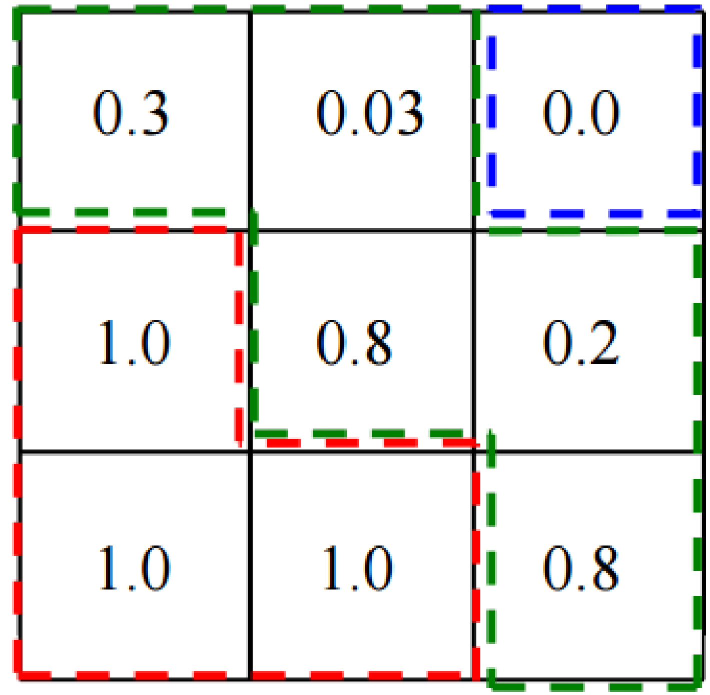
$C_1 = 1$ **The cell is filled with the primary phase**

$C_1 = 0$ **The cell is free of primary phase**

$C_1 \in (0,1)$ **The cell is partially filled with primary phase**

Schematic of 2D two-phase flow system





$C_1 = 1$ **The cell is filled with the primary phase**

$C_1 = 0$ **The cell is free of primary phase**

$C_1 \in (0,1)$ **The cell is partially filled with primary phase**

Governing equation of *C*

1. The change of *C* is due to the flow in/out of the corresponding phase into a cell.

2. *C* is evolved according to local velocity obtained from solving the N-S equations.

$$\frac{\partial C_m}{\partial t} + \mathbf{u}\cdot\nabla C_m = 0$$

Unsteady term Convection term

Convection-diffusion type equation

The two phases are not soluble (互溶), so there is no diffusion term. When there is chemical reaction or phase change, source term is not zero.

The governing equations for multiphase phase flow using VOF

Surface tension force

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

$$\frac{\partial(\rho\mathbf{u})}{\partial t}+\nabla\cdot(\rho\mathbf{u}\mathbf{u})=-\nabla p+\nabla\cdot[\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^{\mathrm{T}})]+\rho\mathbf{g}+\boxed{\mathbf{F}}$$

u from NS equation

$$\frac{\partial C_m}{\partial t}+\mathbf{u}\cdot\nabla C_m=0$$

$$\mathbf{F}=2\sigma k\frac{\rho\nabla C_{\mathrm{l}}}{(\rho_{\mathrm{l}}+\rho_{\mathrm{g}})}$$

CSF model

$$\rho=C_{\mathrm{l}}\rho_{\mathrm{l}}+C_{\mathrm{g}}\rho_{\mathrm{g}}\qquad \mu=C_{\mathrm{l}}\mu_{\mathrm{l}}+C_{\mathrm{g}}\mu_{\mathrm{g}}$$

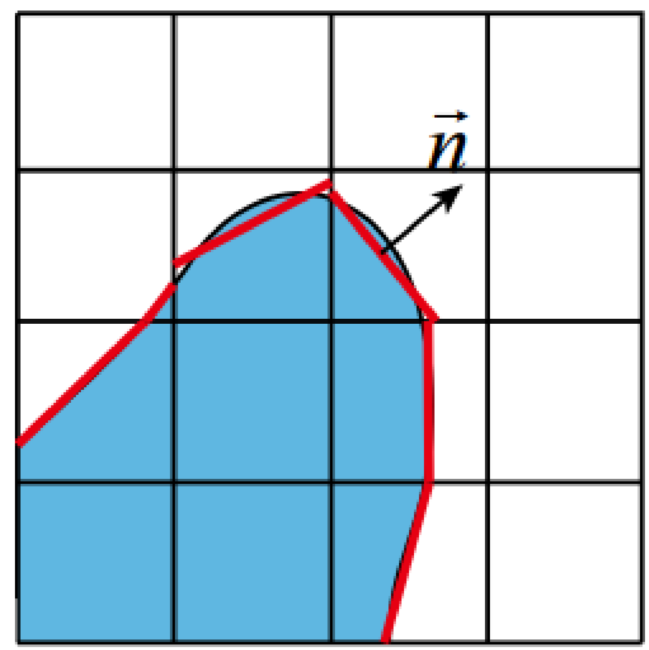
Two-way coupled with each other.

Continuum surface force (CSF) model

The form of volumetric force is required in NS equation. However, surface tension force is a kind of surface force, rather than volumetric force.

CSF transfer the surface tension force to volumetric force

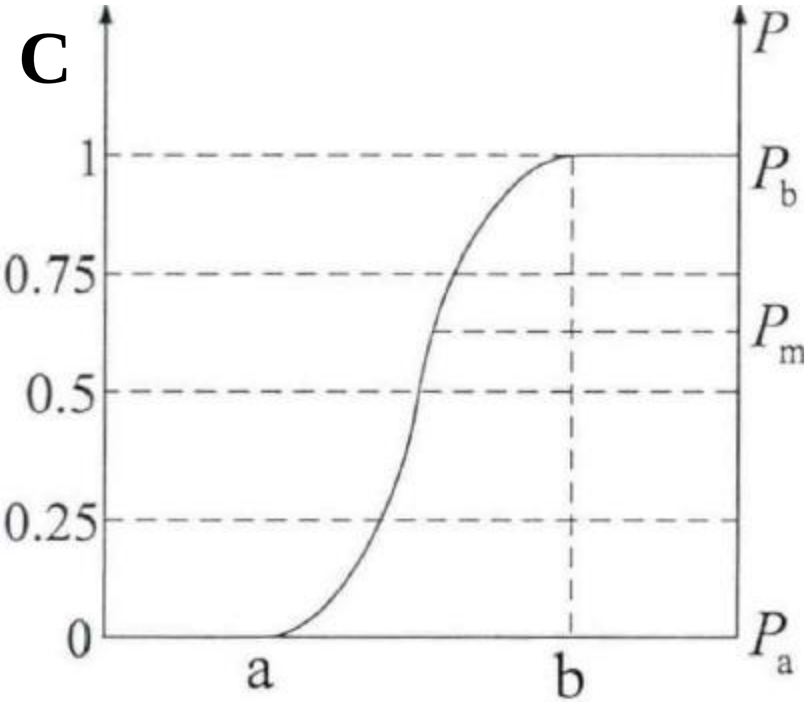
✓ **Smooth C**

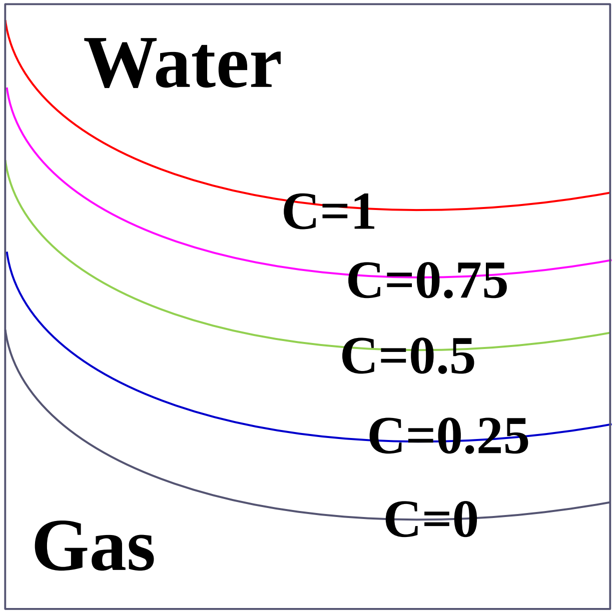


- **VOF in fact is a sharp-interface model.**
- **The thickness of the interface is zero.**
- **C is sharply changed from 1 to 0 across the interface.**

In microscopic, however, the interface is not sharp, it has a finite thickness, for example, of a few nanometers. **Therefore, transition from phase 1 to phase 2 is smooth.**

The purpose of smoothing *C* is to make *C* changes gradually from 1 to 0.

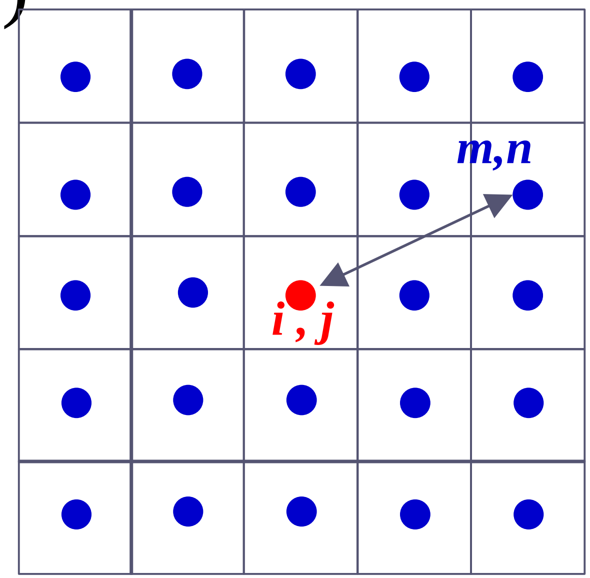




The following function is adopted to smooth C

$$\tilde{C}_{i,j} = \sum_{m,n} C_{m,n} K\left(\left|\mathbf{r}_{i,j} - \mathbf{r}_{m,n}\right|, \varepsilon\right)$$

Smoothed one



ε

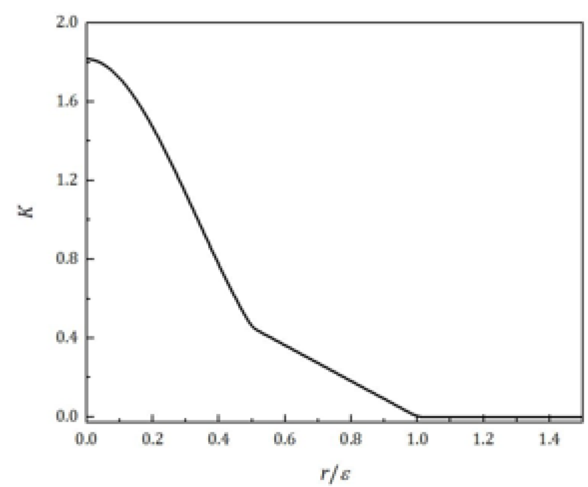
Control the thickness of the interface! 3Δ

$\left|\mathbf{r}_{i,j} - \mathbf{r}_{m,n}\right|$

Distance between two points (i,j) and (m,n)

K **Smooth integration kernel**

$$K(r,\varepsilon) = \begin{cases} (40/7\pi)\left(1 - 6(r/\varepsilon)^2 + 6(r/\varepsilon)^3\right) & (r/\varepsilon < 1/2) \\ (80/7\pi)(1 - r/\varepsilon) & (1/2 \le r/\varepsilon < 1) \\ 0 & (r/\varepsilon > 1) \end{cases}$$



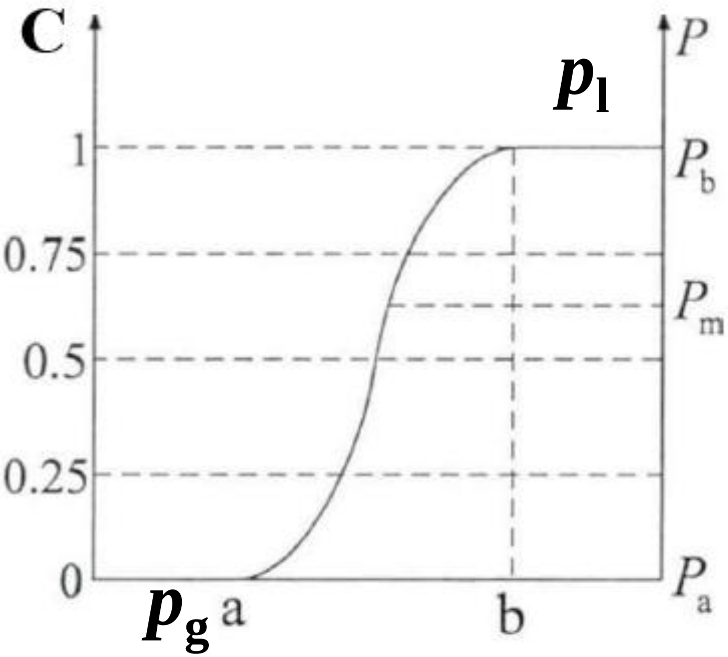
Smoothed C, namely $\tilde{C}$, is adopted to calculate force

$$\mathbf{n} = \nabla \tilde{C}$$

interface mean curvature

$$k = \frac{1}{r} = \nabla \cdot \left(\frac{\nabla \tilde{C}_{\mathrm{l}}}{|\nabla \tilde{C}_{\mathrm{l}}|}\right)$$

pressure in the transition region is



$$P_x = P_{\mathrm{g}} + \sigma k (C_x - C_g) = P_{\mathrm{g}} + \sigma k C_x$$

$$\mathbf{F} \sim \nabla(P_x - P_{\mathrm{g}}) = \nabla(\sigma k (C_x - C_g))$$
$$= \sigma k \nabla C$$

$$\mathbf{F} = 2\sigma k \frac{\rho \nabla C_{\mathrm{l}}}{(\rho_{\mathrm{l}} + \rho_{\mathrm{g}})}$$

Suppose local *k* is constant.

How to solve the VOF equation?

$$\frac{\partial C_m}{\partial t} + \mathbf{u} \cdot \nabla C_m = 0$$

Conservation form

$$\frac{\partial C_m}{\partial t} + \nabla(\mathbf{u} C_m) = 0$$

1. This is a convection-diffusion equation without diffusion term, and can be solved using schemes (such as QUICK) introduced in NHT.

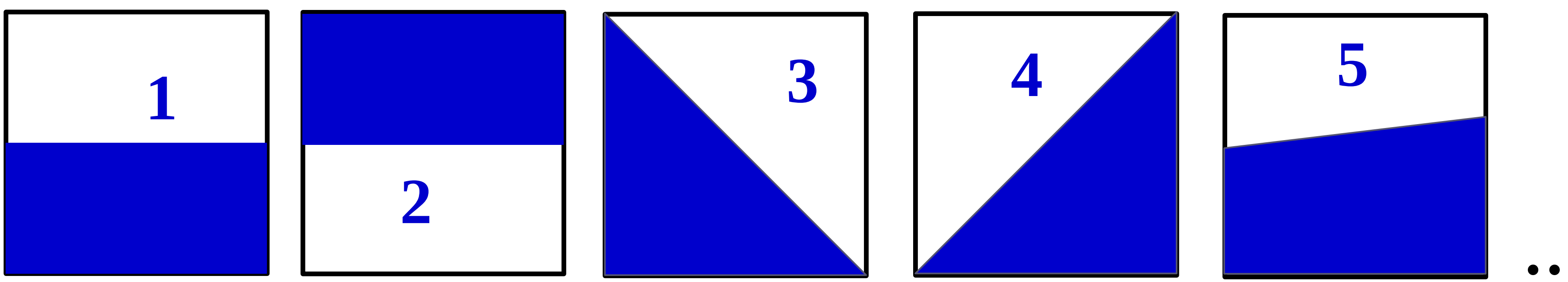
However, because *C* is not a continuous function. Such method may result in false diffusion, leading to gradually increasing thickness of the interface.

2. Reconstruction method

Step 1. Interface reconstruction

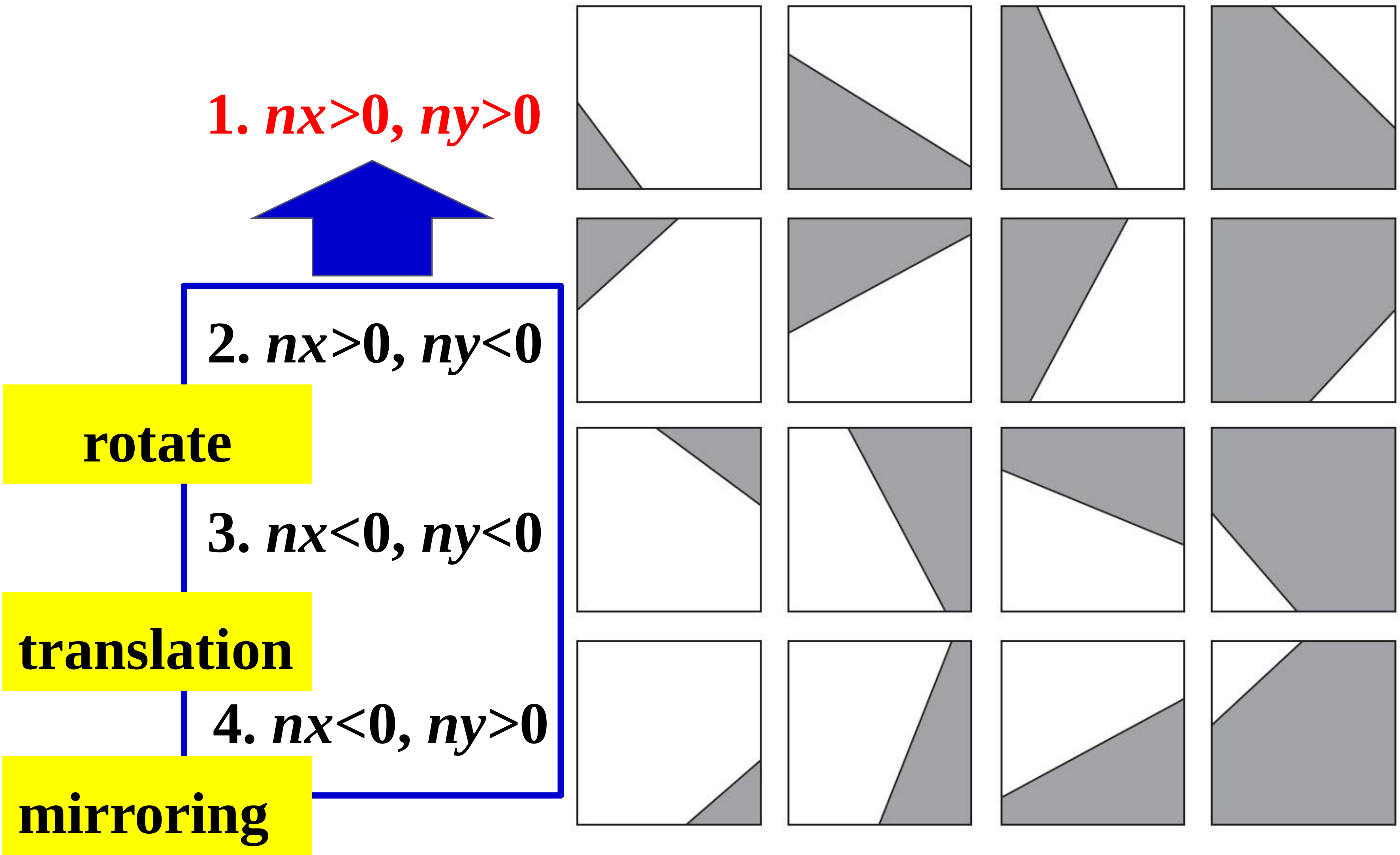
For a value of *C* in a computational cell, the pattern of interface should be determined first.

For example, for *C*=0.5, the interface may be as follows.



Then which one is the right interface?

There are totally 16 kinds of interface pattern, depending on local *C* and normal direction (*nx*, *ny*)



Normal direction of the interface

$$n_{i,j}^{x}=(C_{i+1,j+1}+2C_{i+1,j}+C_{i+1,j-1}-C_{i-1,j+1}-2C_{i-1,j}-C_{i-1,j-1})/\delta x$$

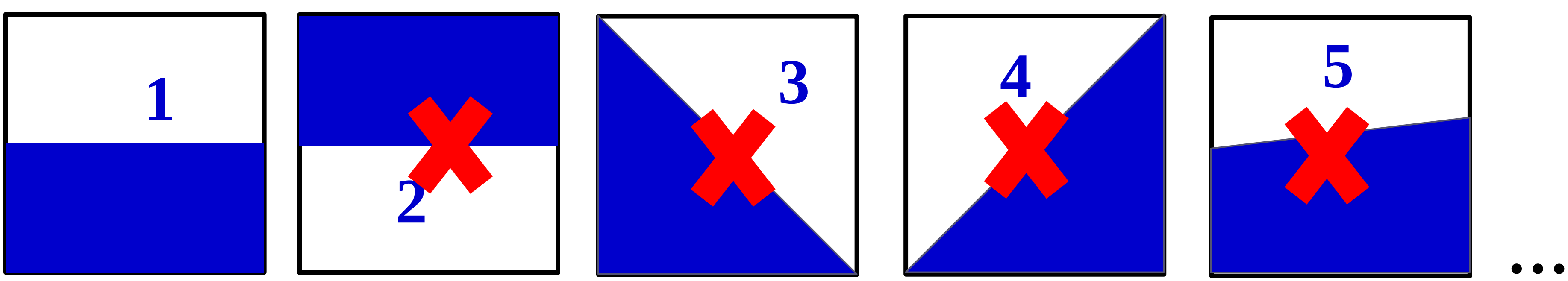
$$n_{i,j}^{y}=(C_{i+1,j+1}+2C_{i,j+1}+C_{i-1,j+1}-C_{i+1,j-1}-2C_{i,j-1}-C_{i-1,j-1})/\delta y$$

Interface normal direction

Volume of fraction

Interface is reconstructed!

For example, for C=0.5, nx=0, ny=1



Normal direction of the interface

$$n_{i,j}^{x}=(\tilde{C}_{i+1,j+1}+2\tilde{C}_{i+1,j}+\tilde{C}_{i+1,j-1}-\tilde{C}_{i-1,j+1}-2\tilde{C}_{i-1,j}-\tilde{C}_{i-1,j-1})/\delta x$$

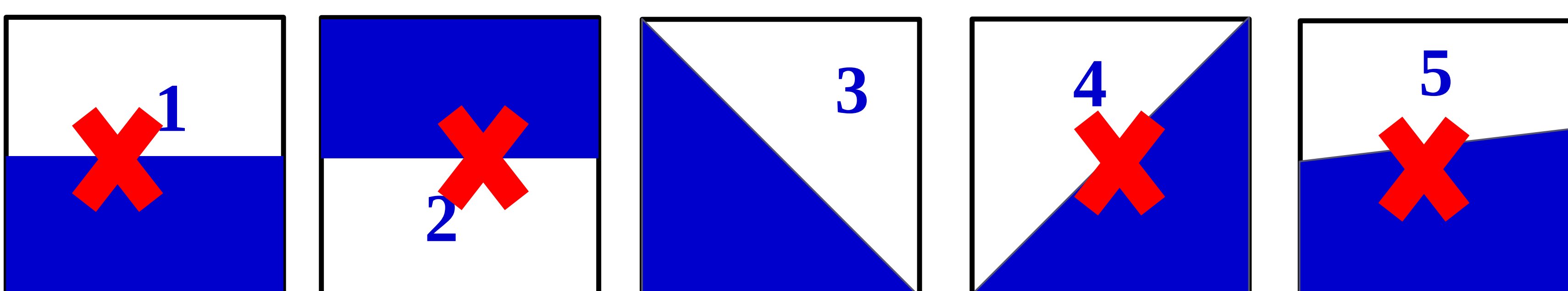
$$n_{i,j}^{y}=(\tilde{C}_{i+1,j+1}+2\tilde{C}_{i,j+1}+\tilde{C}_{i-1,j+1}-\tilde{C}_{i+1,j-1}-2\tilde{C}_{i,j-1}-\tilde{C}_{i-1,j-1})/\delta y$$

Interface normal direction

Volume of fraction

Interface is reconstructed!

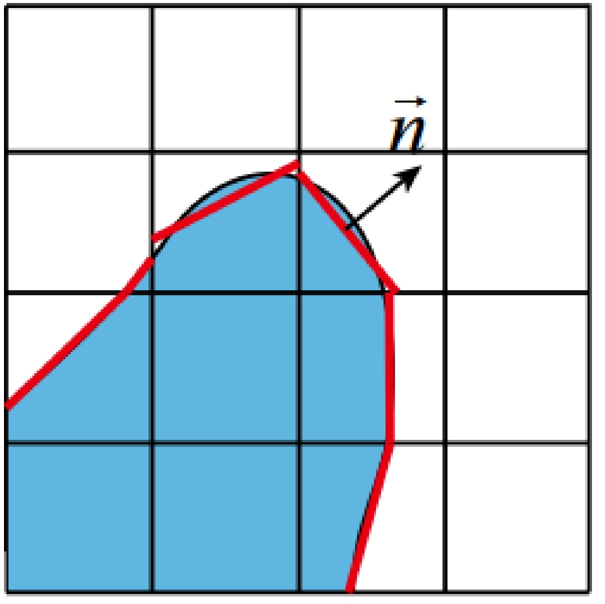
For example, for C=0.5, nx= $1/\sqrt{2}$ **, ny=** $1/\sqrt{2}$



By the reconstruction scheme, the phase interface is determined in each computational cell.

Piecewise linear interface calculation (PLIC)

The smooth interface is approximately described by a set of lines.

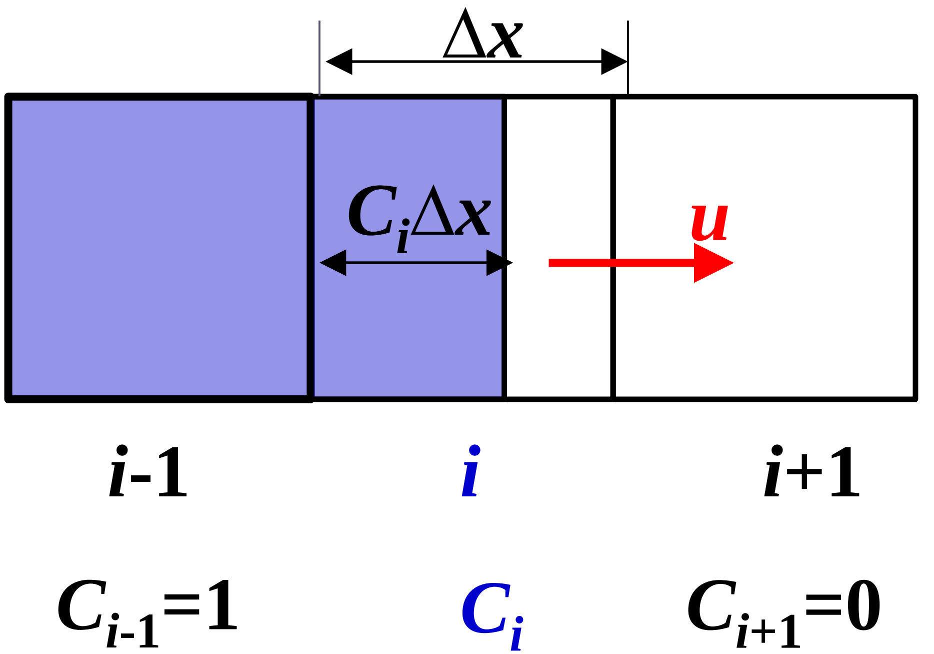


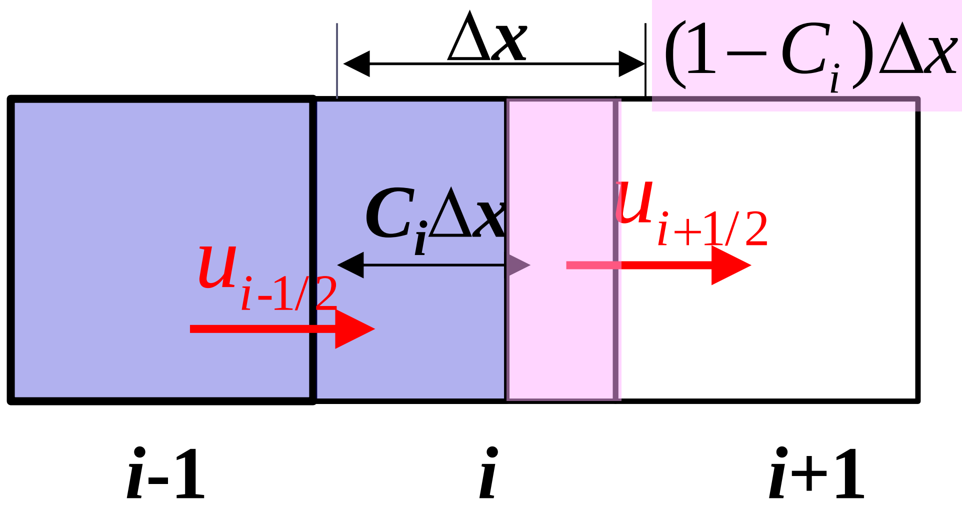
D.L. Youngs, Time-dependent multi-material flow with large fluid distortion, Numerical methods for Fluid Dynamics, 1982, 24(2), 273-285

Step 2. Interface advection in a given velocity field

Based on the reconstructed phase interface, calculate the variation of *C* according to local velocity filed, and then update the *C* for next time step.

Taking 1D interface as example.





The volume that flows from *i*-1 to *i* is

$$F_{i\text{-}1/2} = u_{i\text{-}1/2}\Delta t$$

The volume that flows from *i* to *i*+1 is

$$F_{i+1/2} = \begin{cases} 0 & u_{i+1/2}\Delta t < (1-C_i)\Delta x \\ u_{i+1/2}\Delta t - (1-C_i)\Delta x & u_{i+1/2}\Delta t > (1-C_i)\Delta x \end{cases}$$

Total volume is

$$C_i^{t+\Delta t} = C_i^t + (F_{i-1/2} - F_{i+1/2}) / \Delta x$$

CFL condition

$$C_i^{t+\Delta t} = C_i^t + (F_{i-1/2} - F_{i+1/2}) / \Delta x$$

Courant number $u\Delta t / \Delta x$

Because C **should be smaller than 1, thus** $u\Delta t / \Delta x$ **also should be smaller than 1. This is the CFL condition.**

Therefore, during the numerical simulation, the time step should be sufficiently low that the CFL condition is satisfied, or the Courant number <1.

The governing equations for multiphase phase flow using VOF

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

$$\mathbf{F}=2\sigma k\frac{\rho\nabla C_{\mathrm{l}}}{(\rho_{\mathrm{l}}+\rho_{\mathrm{g}})}$$

CSF model

$$\frac{\partial(\rho\mathbf{u})}{\partial t}+\nabla\cdot(\rho\mathbf{u}\mathbf{u})=-\nabla p+\nabla\cdot[\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^{\mathrm{T}})]+\rho\mathbf{g}+\boxed{\mathbf{F}}$$

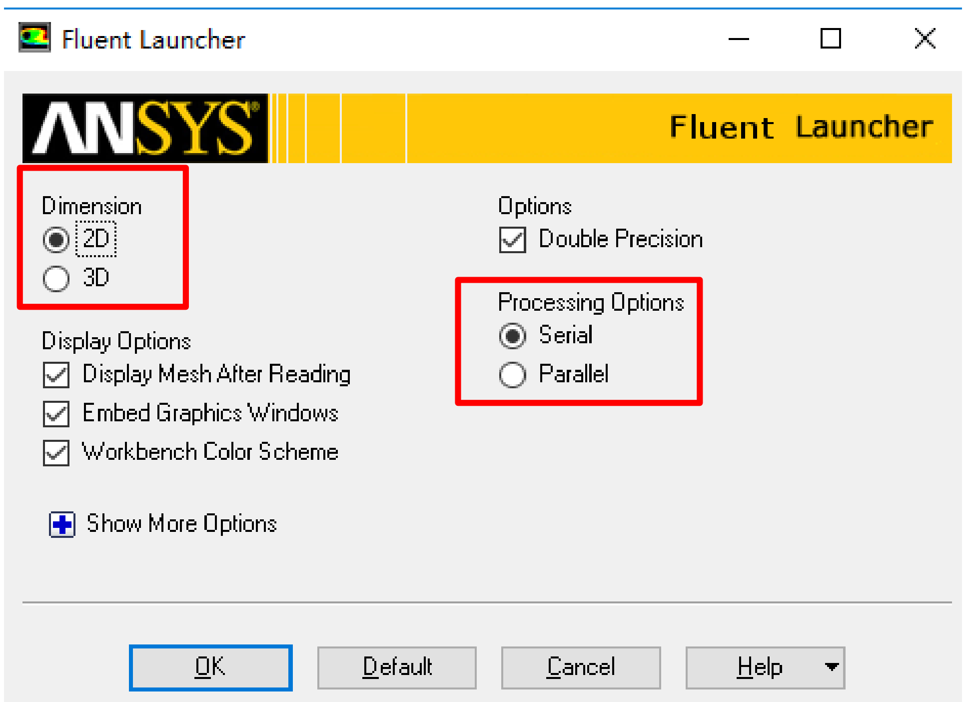
$$\frac{\partial C_m}{\partial t}+\mathbf{u}\cdot\nabla C_m=0$$

$$\rho=C_{\mathrm{l}}\rho_{\mathrm{l}}+C_{\mathrm{g}}\rho_{\mathrm{g}}\qquad\mu=C_{\mathrm{l}}\mu_{\mathrm{l}}+C_{\mathrm{g}}\mu_{\mathrm{g}}$$

Two-way coupled with each other.

2 Process of simulation

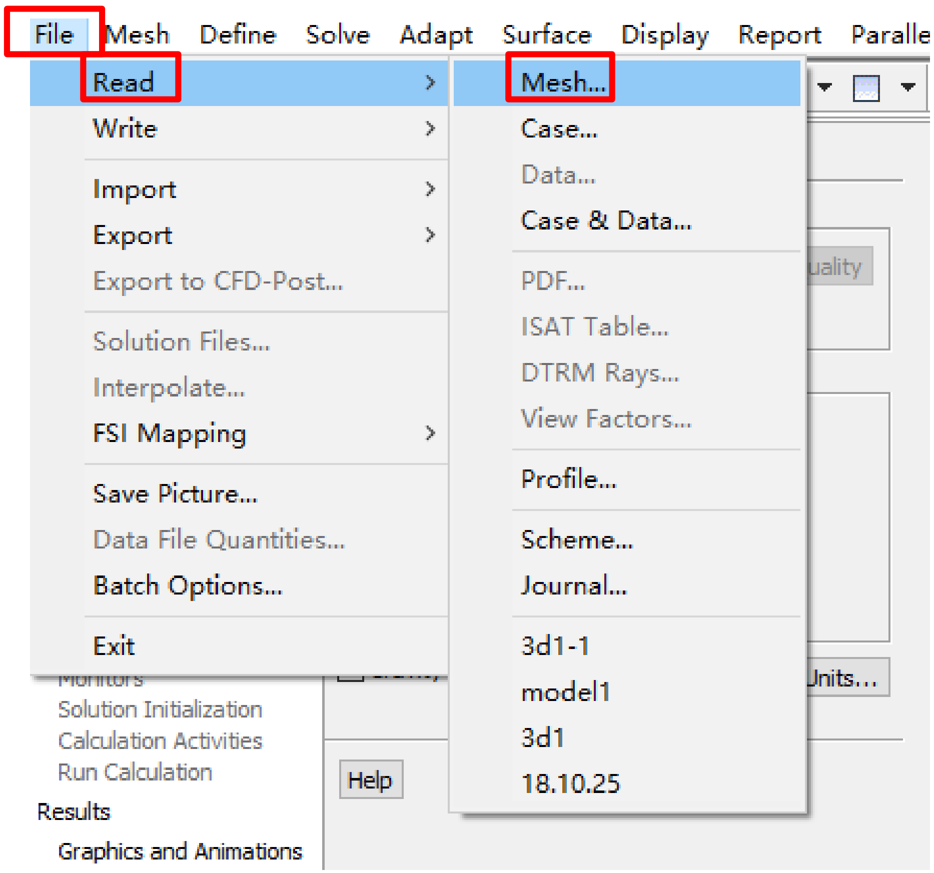
2.1 Launch ANSYS Fluent



- Choose 2-Dimension
- Choose Display Options
- Choose Double Precision
- Choose Serial Processing

2.2 Read the mesh

File → Read→Mesh

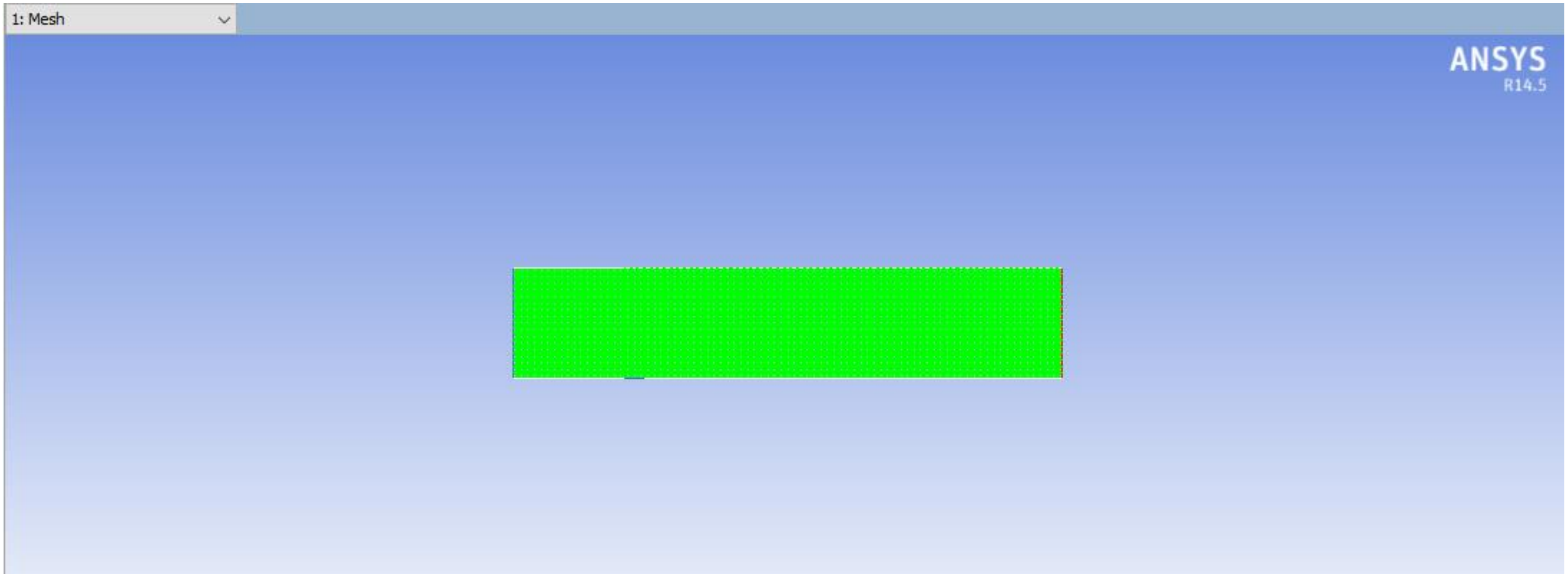



```
Building...
     mesh
     materials,
     interface,
     domains,
     zones,
        water
        gdl
        wall
        air-in
        air-out
        int_fluid
        fluid
Done.

Preparing mesh for display...
Done.
```

2.3 Check the mesh

General→Mesh→Check

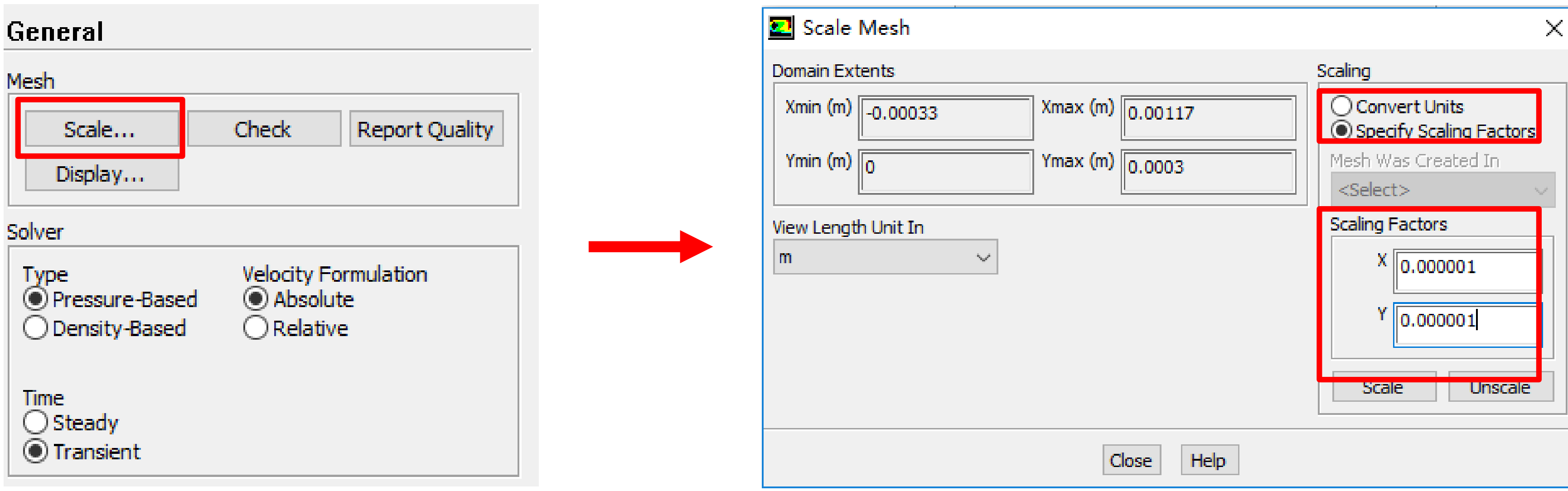



```
Mesh Check

 Domain Extents:
   x-coordinate: min (m) = -3.300000e+02, max (m) = 1.170000e+03
   y-coordinate: min (m) = 0.000000e+00, max (m) = 3.000000e+02
 Volume statistics:
   minimum volume (m3): 2.500000e+01
   maximum volume (m3): 2.500000e+01
     total volume (m3): 4.500000e+05
 Face area statistics:
   minimum face area (m2): 5.000000e+00
   maximum face area (m2): 5.000000e+00
 Checking mesh.........................
Done.
```

2.4 Scale the domain size

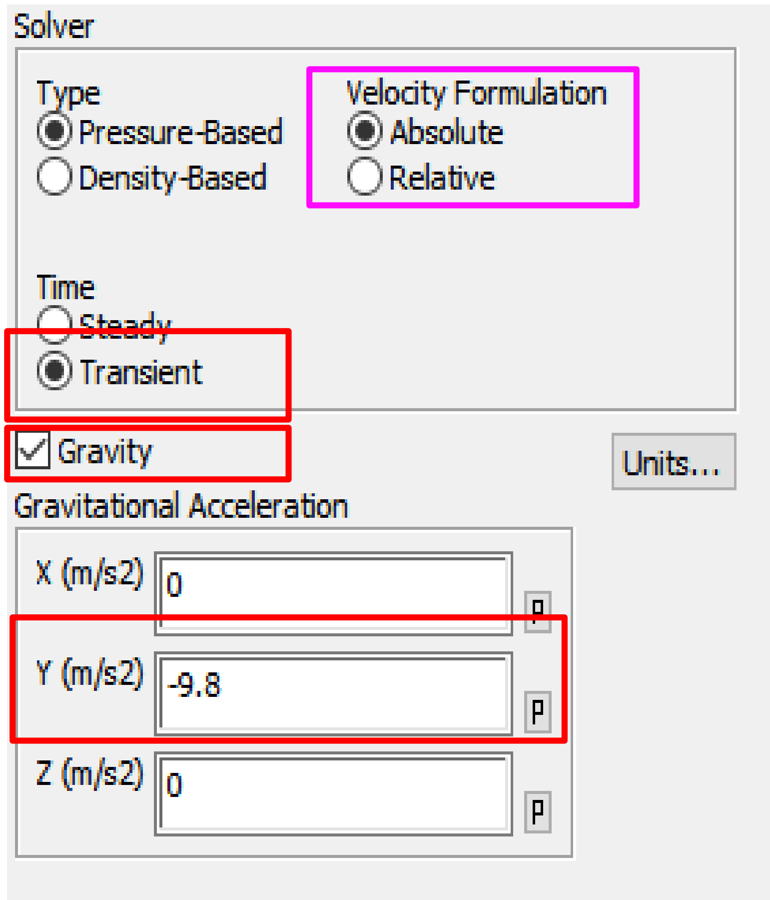
General→Mesh→Scale



- Choose **Specify Scaling Factor**
- Write **0.000001** in the Scaling Factor box to convert the unite from **m** to **μm**.

2.5 Choose the solver

General→Solver



- **Choose Transient**

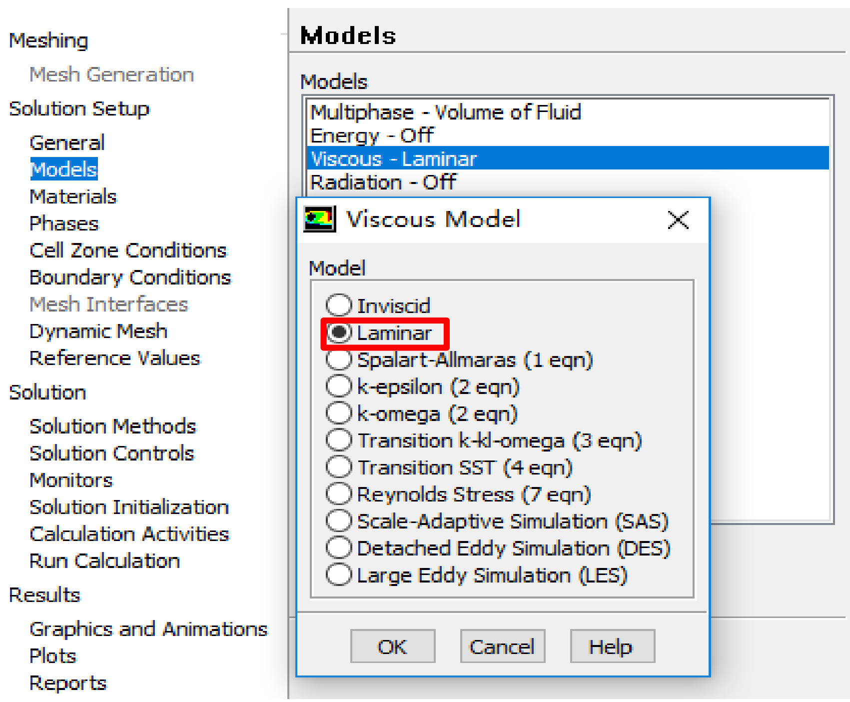
  The dynamic behaviors of water is to be studied.

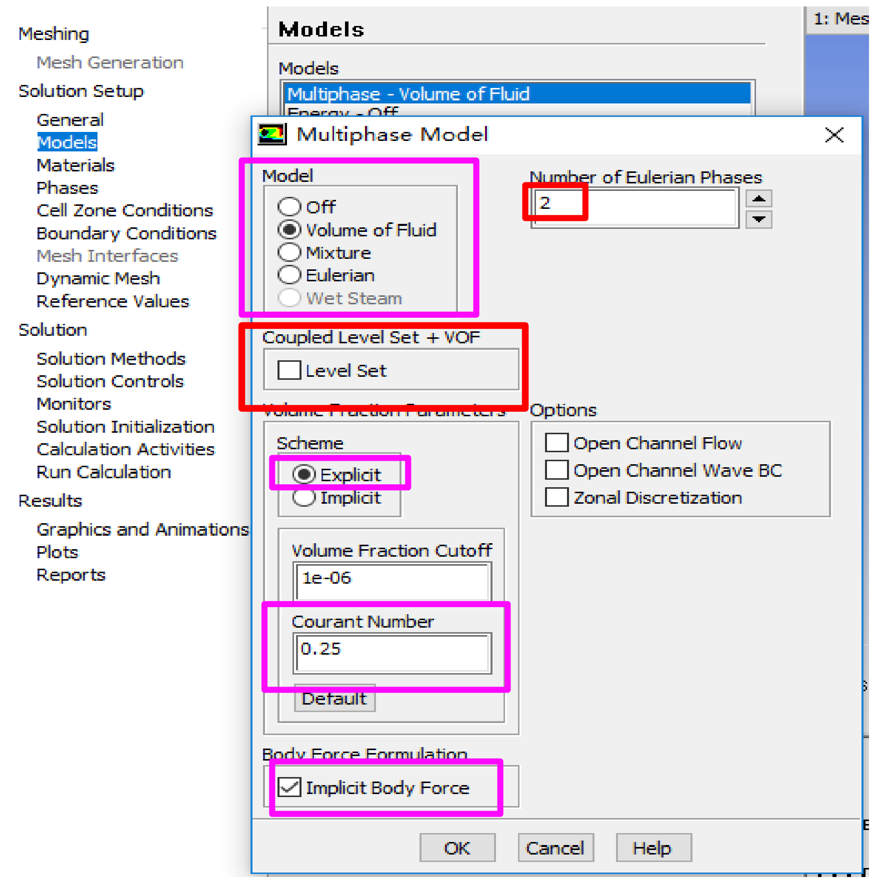
- **Select Gravity**
- **Write -9.8 in the Gravitational Acceleration box of Y.**

Density-based method cannot be used for VOF.

2.6 Choose the models

Solution Setup→Models





- Choose **Volume of Fluid** as Multiphase Model
- Coupled Level set +VOF?

Coupled Level Set +VOF

Spatial gradient (interface curvature and surface tension force)

LS function is smooth and continuous, accurate! ✓ Advantages	VOF is discontinues across the interface, not accurate ✗

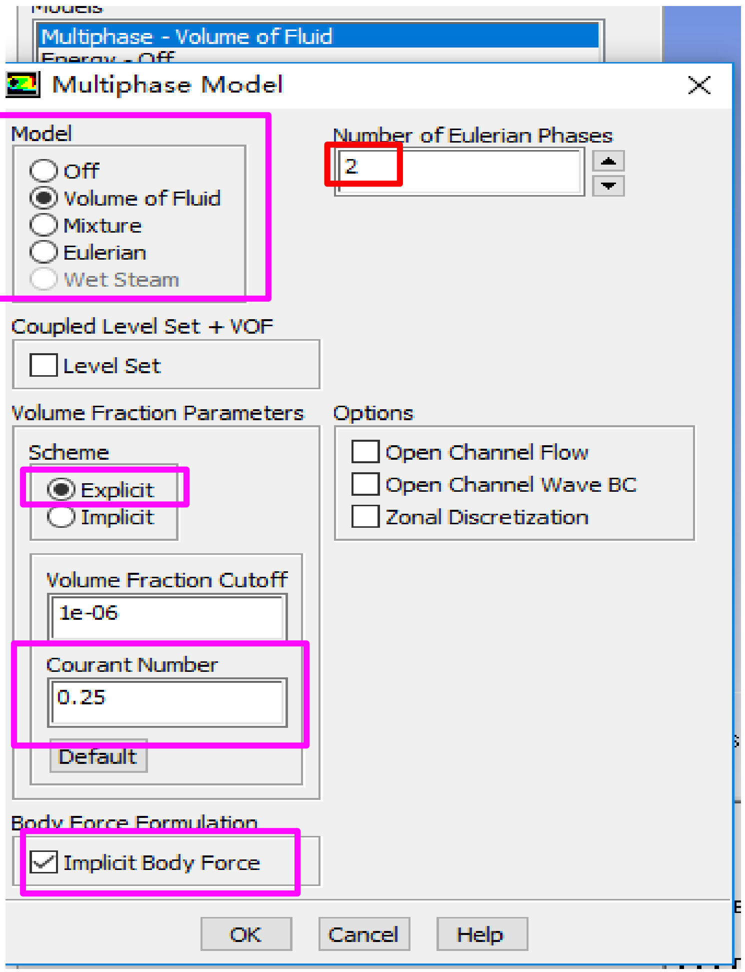
Mass conservation

✗ LS is not good.	VOF is good. ✓

Combining the advantages of LS and VOF

2.6 Choose the models

Body force formulation



Large body forces (for example, gravity or surface tension forces) are included.

Implicit body force is adopted to improves solution convergence by accounting for the partial equilibrium of the pressure gradient and surface tension forces.

Forces in Momentum equation

Multiphase flow is controlled by a set of forces.

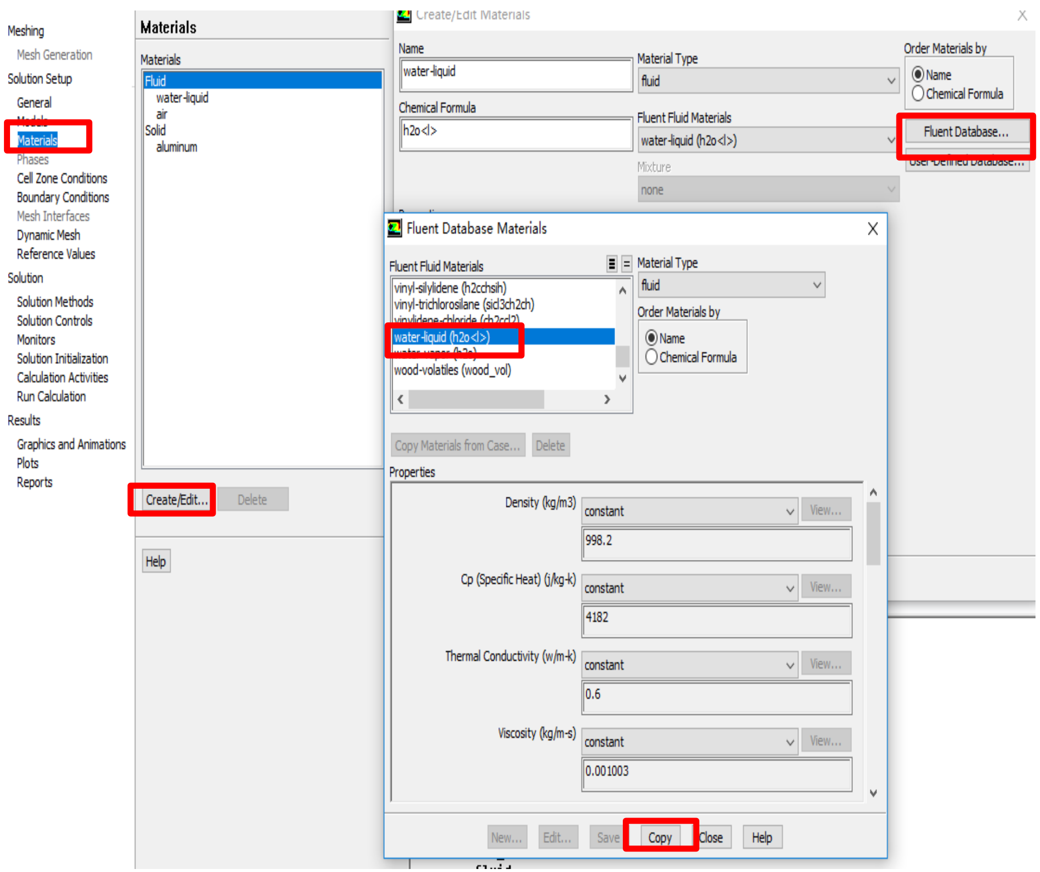
$$\frac{\partial(\rho\mathbf{u})}{\partial t}+\overbrace{\nabla\cdot(\rho\mathbf{u}\mathbf{u})}^{\text{Inertial force}}=\underbrace{-\nabla p}_{\text{Pressure}}+\overbrace{\nabla\cdot[\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^{\mathrm{T}})]}^{\text{Viscous force}}+\overbrace{\boxed{\underbrace{\rho\mathbf{g}}_{\text{gravity}}+\underbrace{2\sigma k\frac{\rho\nabla C_{\mathrm{l}}}{(\rho_{\mathrm{l}}+\rho_{\mathrm{g}})}}_{\text{Surface tension force}}}}^{\text{Body force}}$$

*** Here surface tension force is converted to body force using CSF.**

2.7 Define the materials

Solution Setup→ Materials→Create/Edit Material

■ Create water-liquid

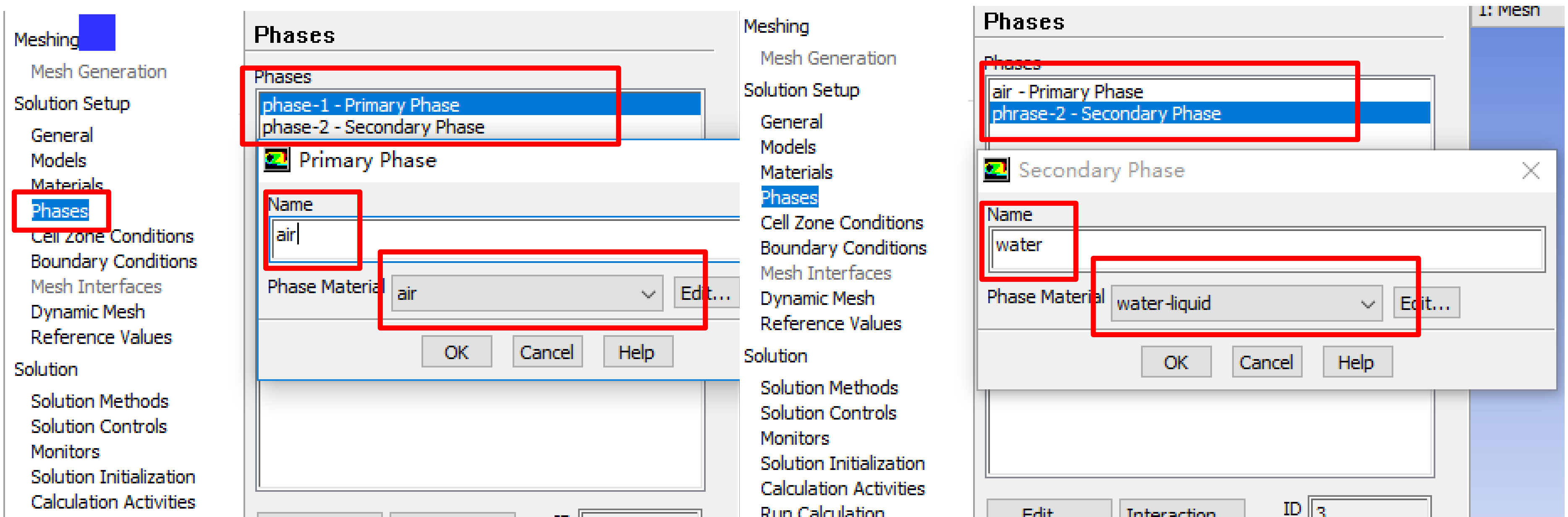


1. Click Fluent Database
2. Choose water-liquid
3. Click Copy

2.8 Define the phases

Solution Setup→ Phases

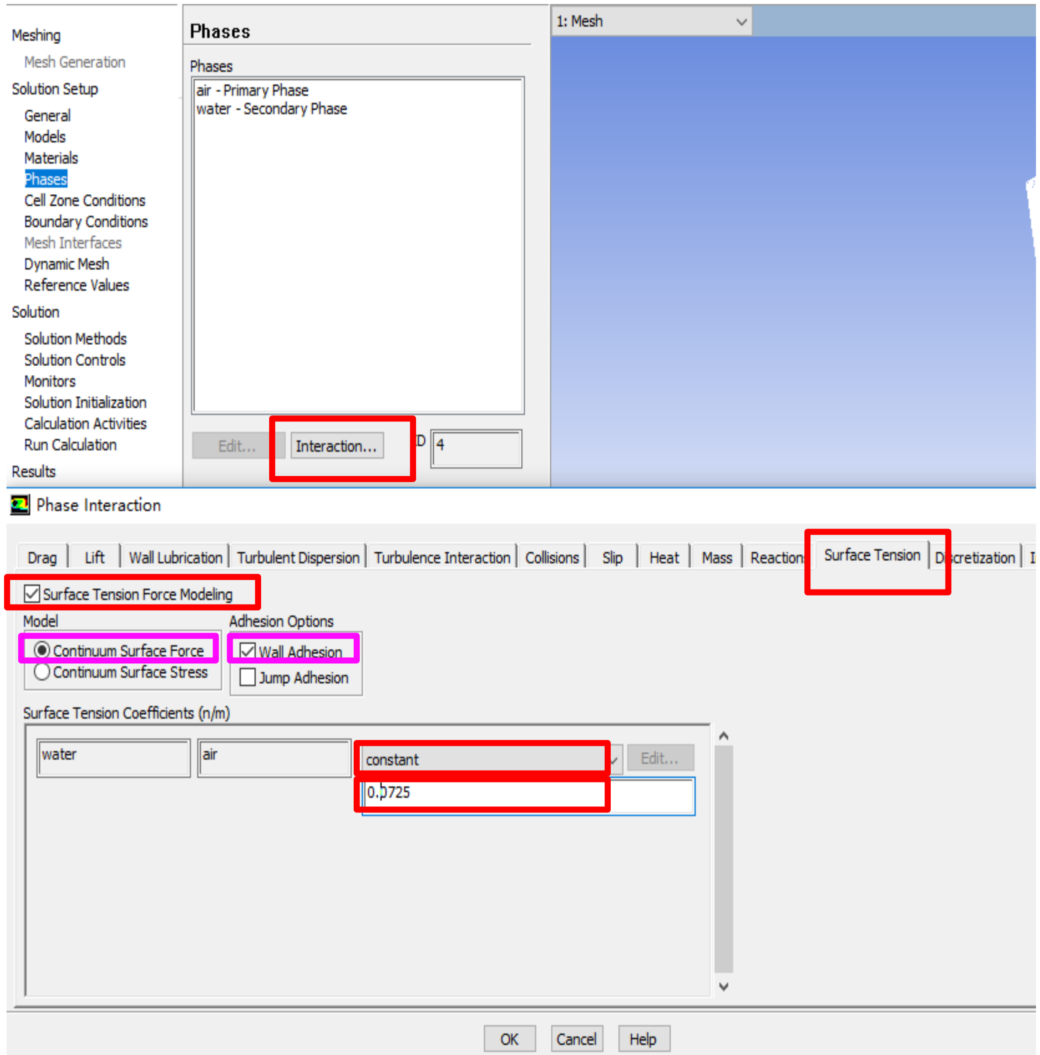
- **Choose air as Primary Phase**
- **Choose water-liquid as Secondary Phase**



Primary phase is usually set as the one dominated in the computational domain.

2.8 Define the phases

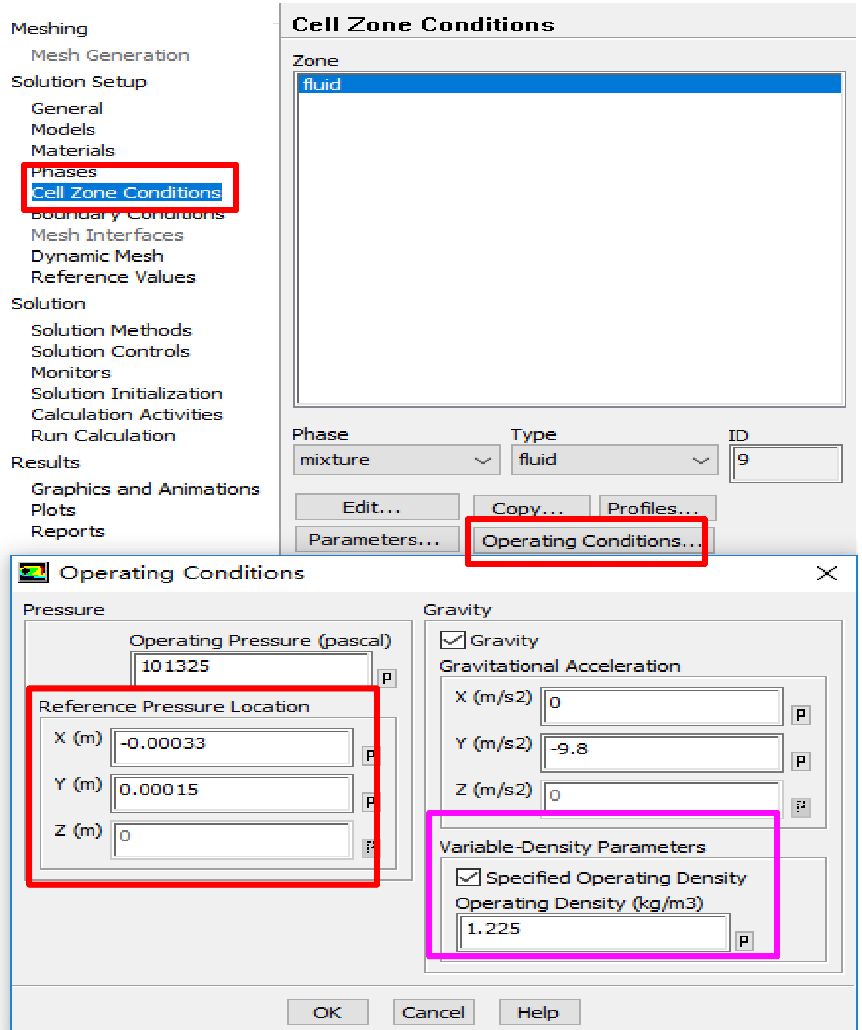
Define surface tension force



1. Click **Interaction**
2. Click **Surface Tension**
3. Select **Surface Tension Force Modeling**
4. Choose **Continuum Surface Force** and **Wall Adhesion.**
5. Choose **constant** and write **0.0725**.

2.9 Define cell zone conditions

Solution Setup→ Cell Zone Condition →Operating Conditions

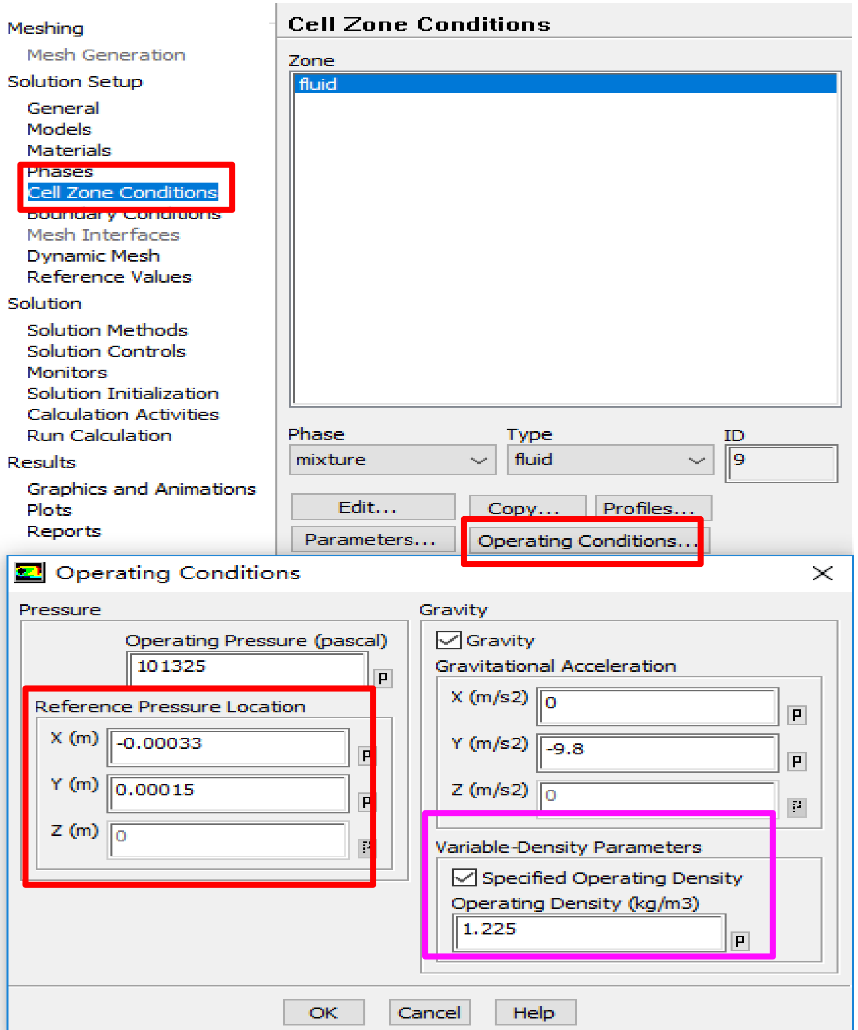


Operating pressure

- **In Fluent, operating pressure is the same as reference pressure.**
- **Input location and value of operating pressure.**

2.9 Define cell zone conditions

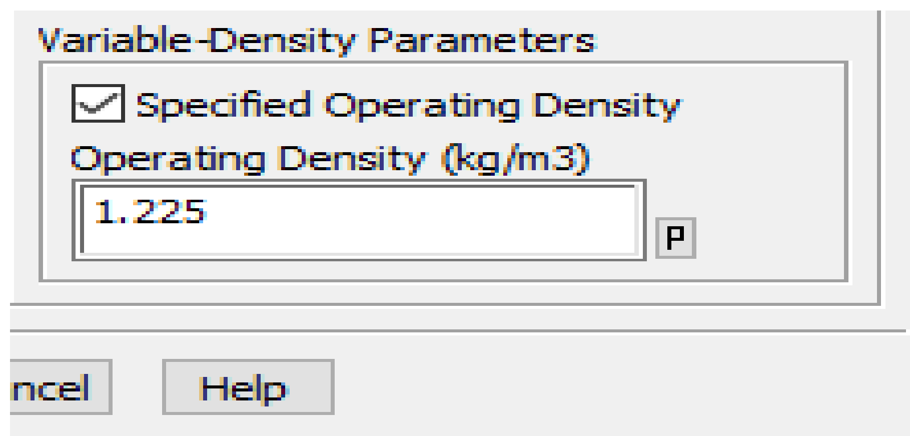
Solution Setup→ Cell Zone Condition →Operating Conditions



Specified operating density

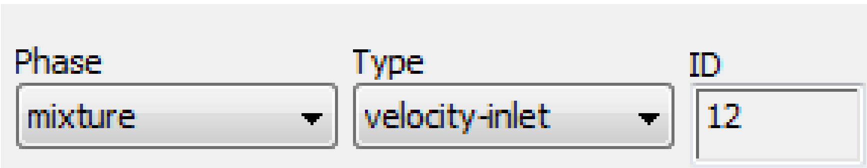
Set the operating density to be the density of the lightest phase.

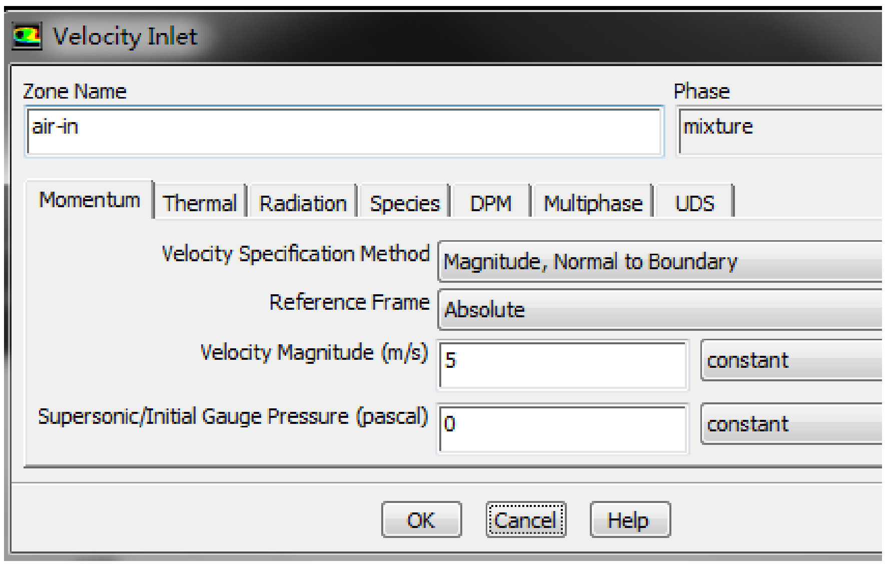
Here input the density of the air.

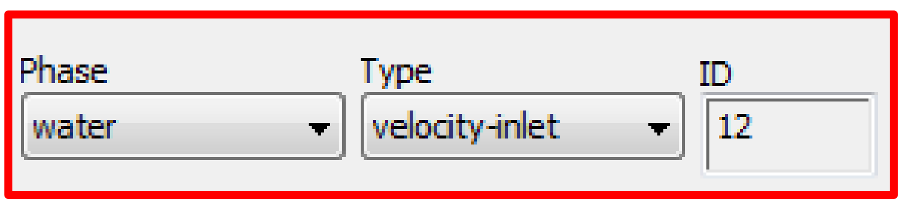


2.10 Define the boundary conditions

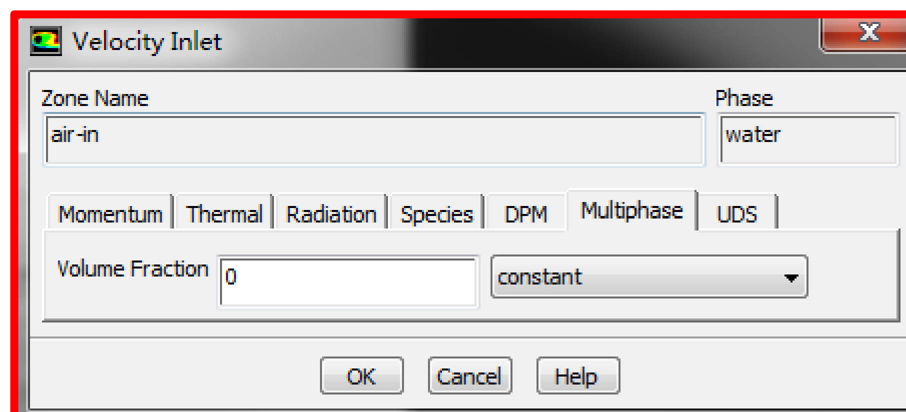
Solution Setup→ Boundary Condition





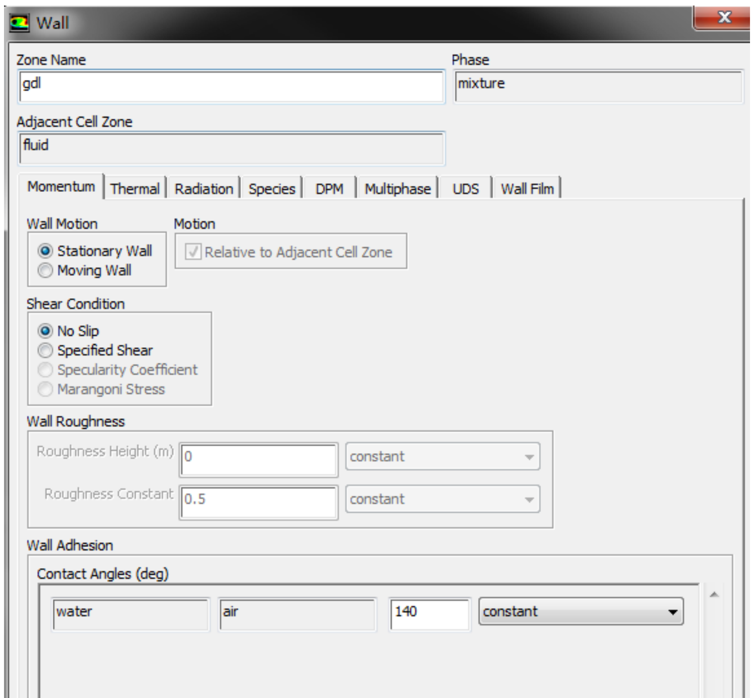


- For the left inlet, pure air flows into the domain.
- Velocity inlet is adopted for the mixture.
- Volume fraction of each phase should be given.

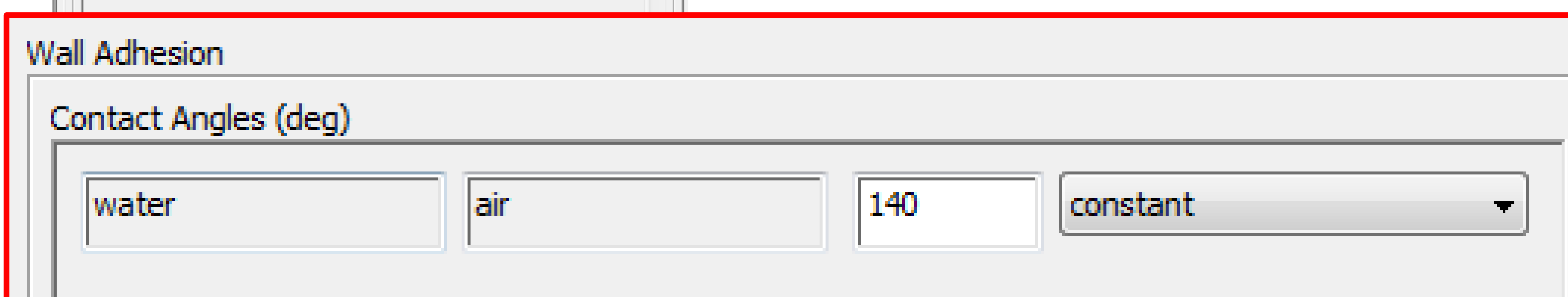


2.10 Define the boundary conditions

For the top and bottom surface, Define contact angle

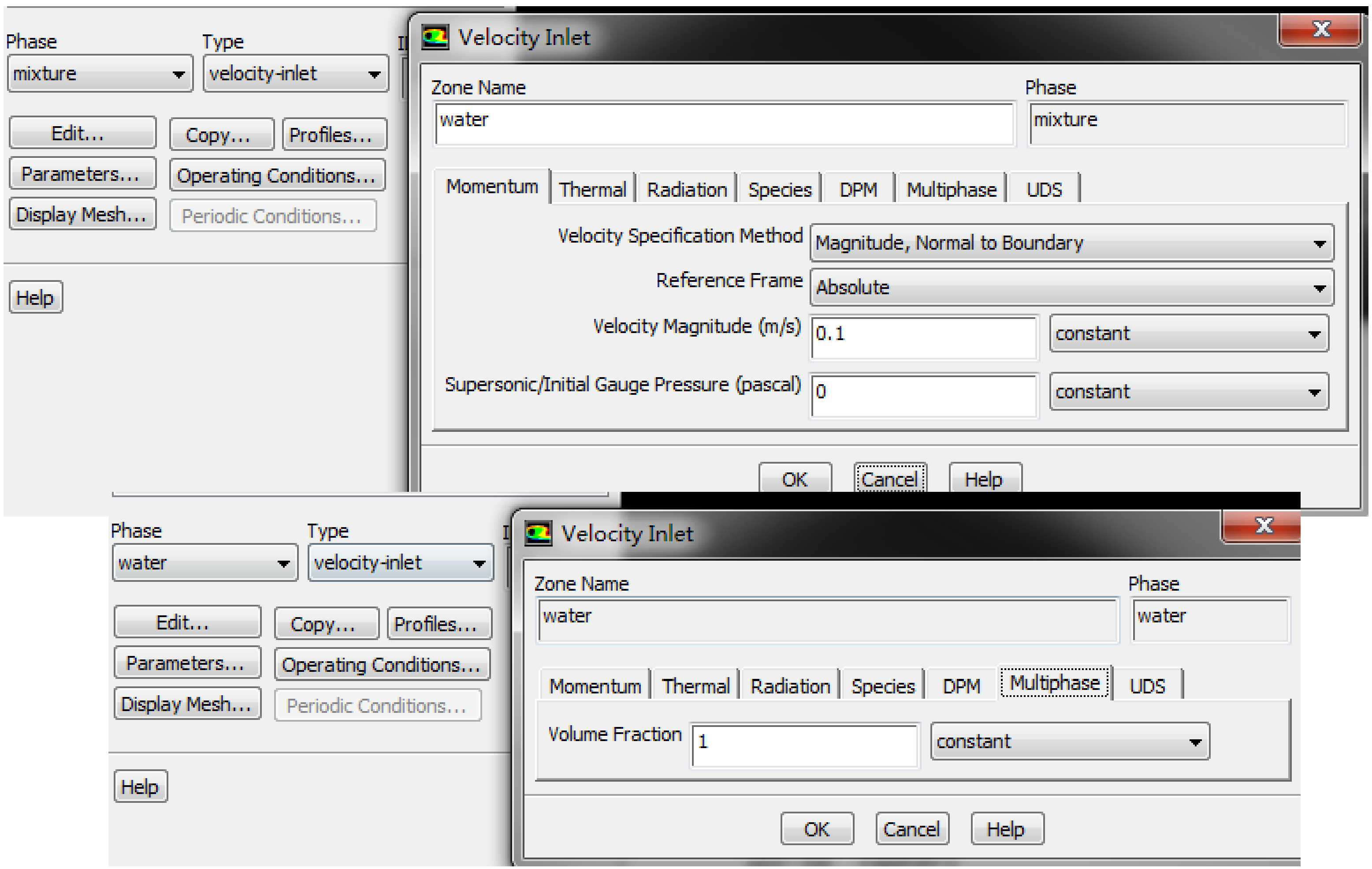


- Choose **wall** as **Type**
- Input value of the **contact angle 140º**
- **The angle is measured by water here.**



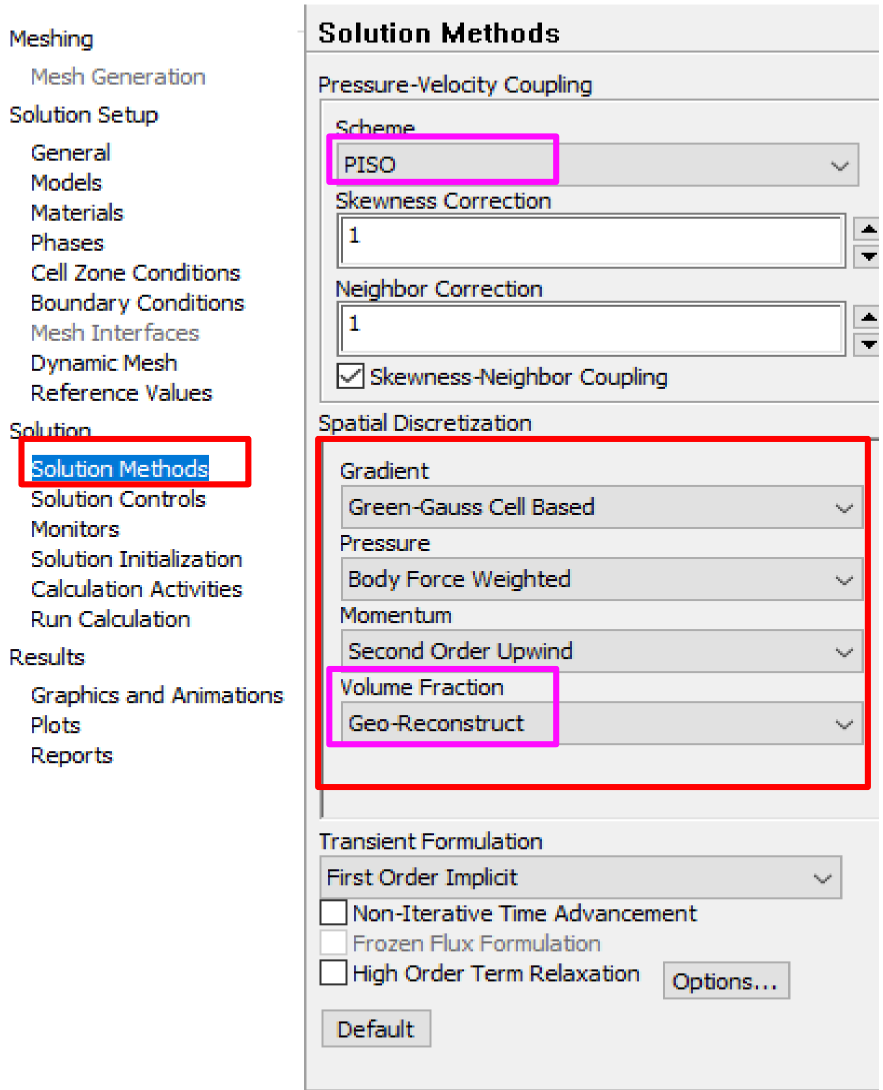
2.10 Define the boundary conditions

For the water inlet, define velocity inlet condition; define the volume fraction of water as 1.



2.11 Choose the solution methods

Solution → Solution Methods



- Choose PISO (Scheme)
- Choose Green-Gauss Cell Based (Gradient)
- Choose Body Force Weighted (Pressure)
- Choose Second Order Upwind (Momentum)
- Choose Geo-Reconstruct (Volume Fraction)

The Pressure-Implicit with Splitting of Operators (PISO)

The PISO also belongs to the family of SIMPLE.

There are one time of prediction step (预估） and correction step (校正） in SIMPLEC.

Prediction step: determine u* and v* based on u^0 and v^0

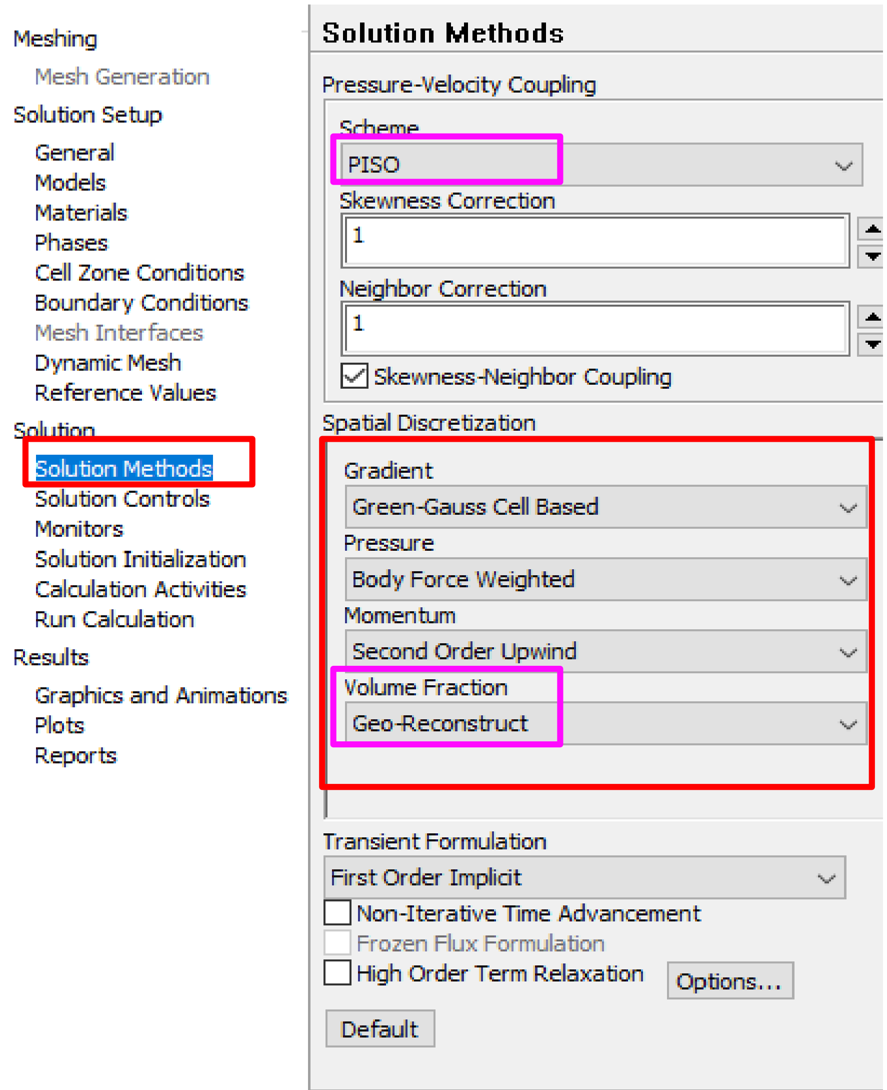
Correction step: solve pressure correction, obtain u and v that satisfying the Mass Conservation Equation.

In PISO, two times of correction steps are conducted, thus improving the convergence.

PISO is recommended for transient problem.

2.11 Choose the solution methods

Solution → Solution Methods



- Choose PISO (Scheme)
- Choose Green-Gauss Node Based (Gradient)
- Choose Body Force Weighted (Pressure)
- Choose Second Order Upwind (Momentum)
- Choose Geo-Reconstruct (Volume Fraction)

Gradient calculation

1. Green-Gauss Cell-Based (格林-高斯基于单元法）
2. Green-Gauss Node-Based (格林-高斯基于节点法)
3. Least-Squares Cell Based 基于单元体的最小二乘法

It is the default scheme for gradient calculation.

The former two are based on Green-Gauss Theory

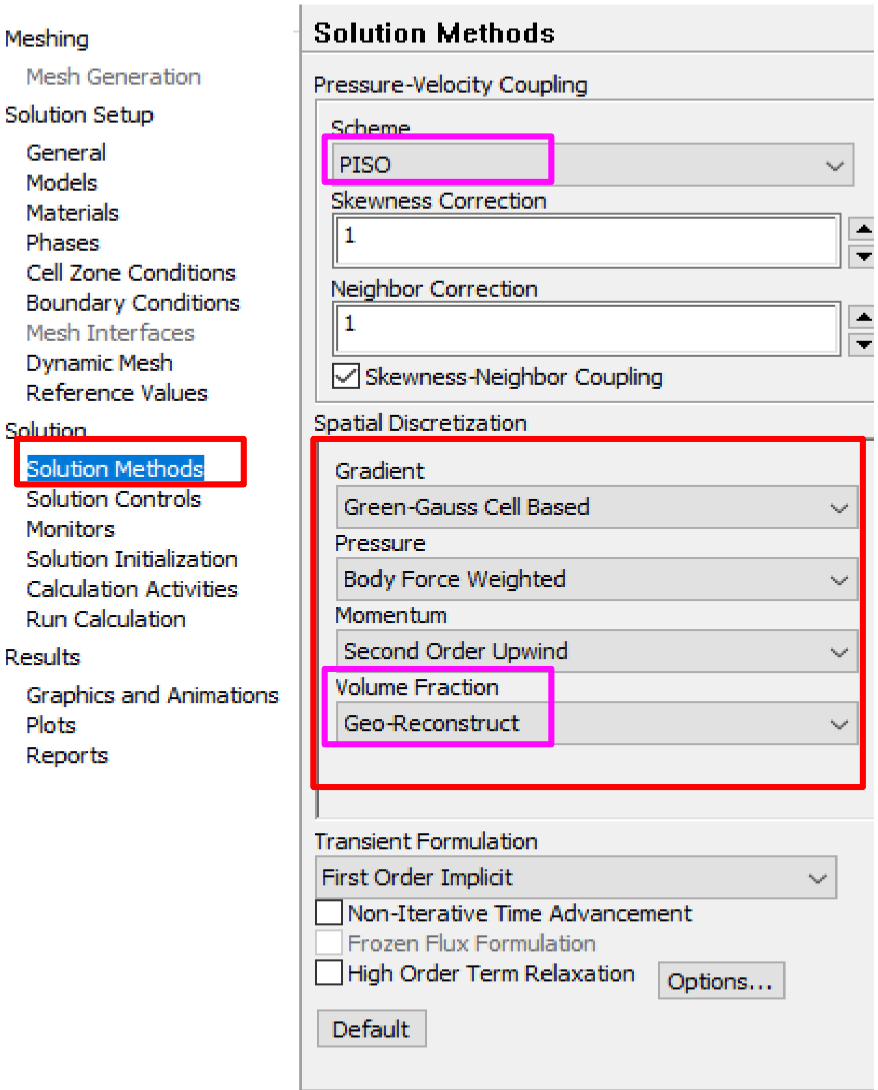
$$<\nabla\phi>=\frac{1}{V_C}\int_{V_C}\nabla\phi dV=\frac{1}{V_C}\int\phi\cdot\mathbf{n}dS=\sum\phi_f\cdot\mathbf{n}S$$

The least-square cell based is based on

$$\xi=\sum_{i=1}^{N}\left\{w_i\left(\phi_{Ci}-\phi_{C0}-\left[\frac{\partial\phi}{\partial x}\Delta x_i+\frac{\partial\phi}{\partial y}\Delta y_i+\frac{\partial\phi}{\partial z}\Delta z_i\right]\right)^2\right\}$$

2.11 Choose the solution methods

Solution → Solution Methods



- Choose PISO (Scheme)
- Choose Green-Gauss Node Based (Gradient)
- Choose Body Force Weighted (Pressure)
- Choose Second Order Upwind (Momentum)
- Choose Geo-Reconstruct (Volume Fraction)

Pressure calculation

1. Linear scheme

2. Standard scheme

3. Second Order

4. Body Force Weighted scheme

Calculate the pressure according to the body force.

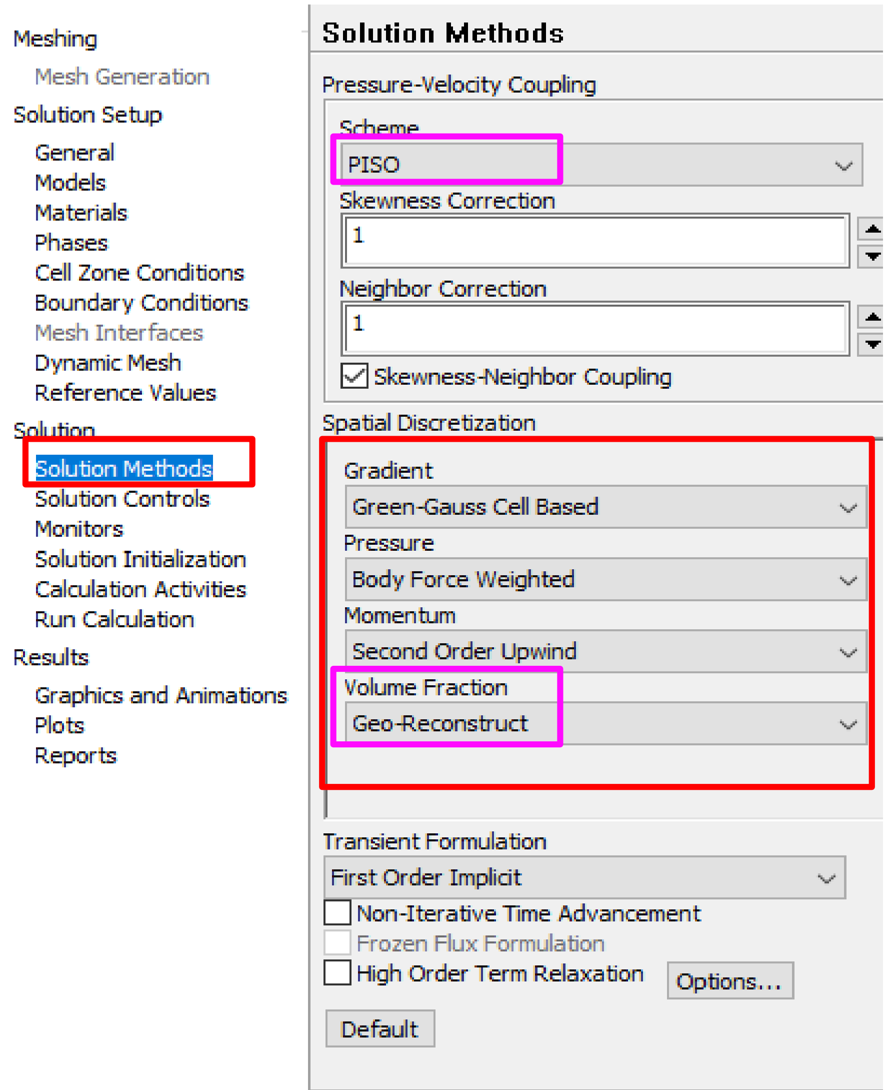
- ✓ Multiphase flow such as VOF (Volume of Fluid，体积函数法) or LS (Level Set, 水平集): recommended.
- ✓ For porous media: not recommended!

5. PRESTO! (Pressure Staggering Option) scheme

For problem with high pressure gradient.

2.11 Choose the solution methods

Solution → Solution Methods



- **Choose PISO (Scheme)**
- **Choose Green-Gauss Node Based (Gradient)**
- **Choose Body Force Weighted (Pressure)**
- **Choose Second Order Upwind (Momentum)**
- **Choose Geo-Reconstruct (Volume Fraction)**

Solving methods for VOF equation

The geometric reconstruction interpolation scheme is recommended when time-accurate transient behaviors of the multiphase are required. In other words, it can accurately predict the sharp interface.

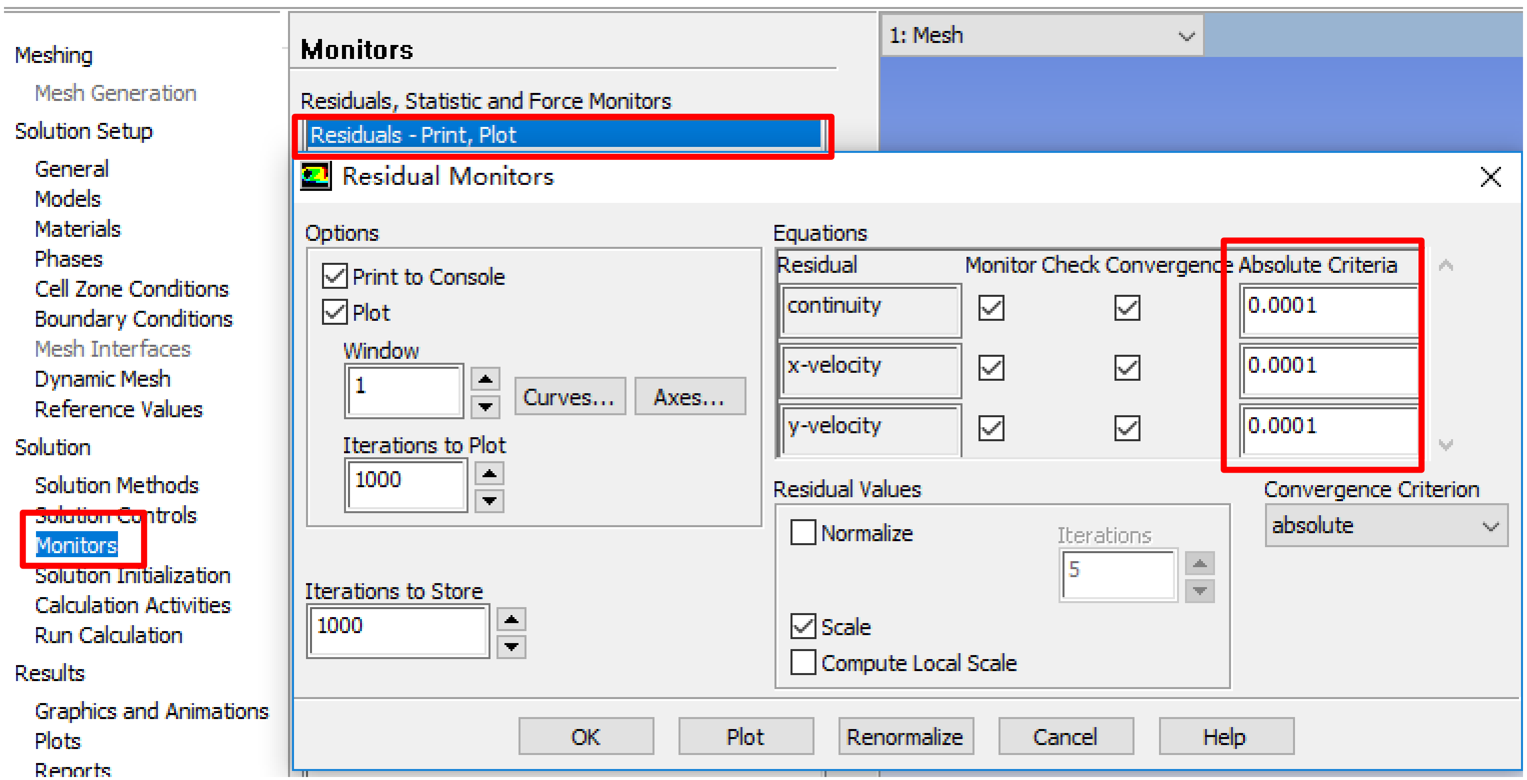
This scheme is the *most accurate* and is *applicable for general unstructured meshes*.

Modified HRIC, Compressive, and CICSAM schemes are less computationally expensive than the Geo-Reconstruct scheme, the interface between phases will *not be as sharp* as the geometric reconstruction interpolation scheme.

2.12 Define the monitors

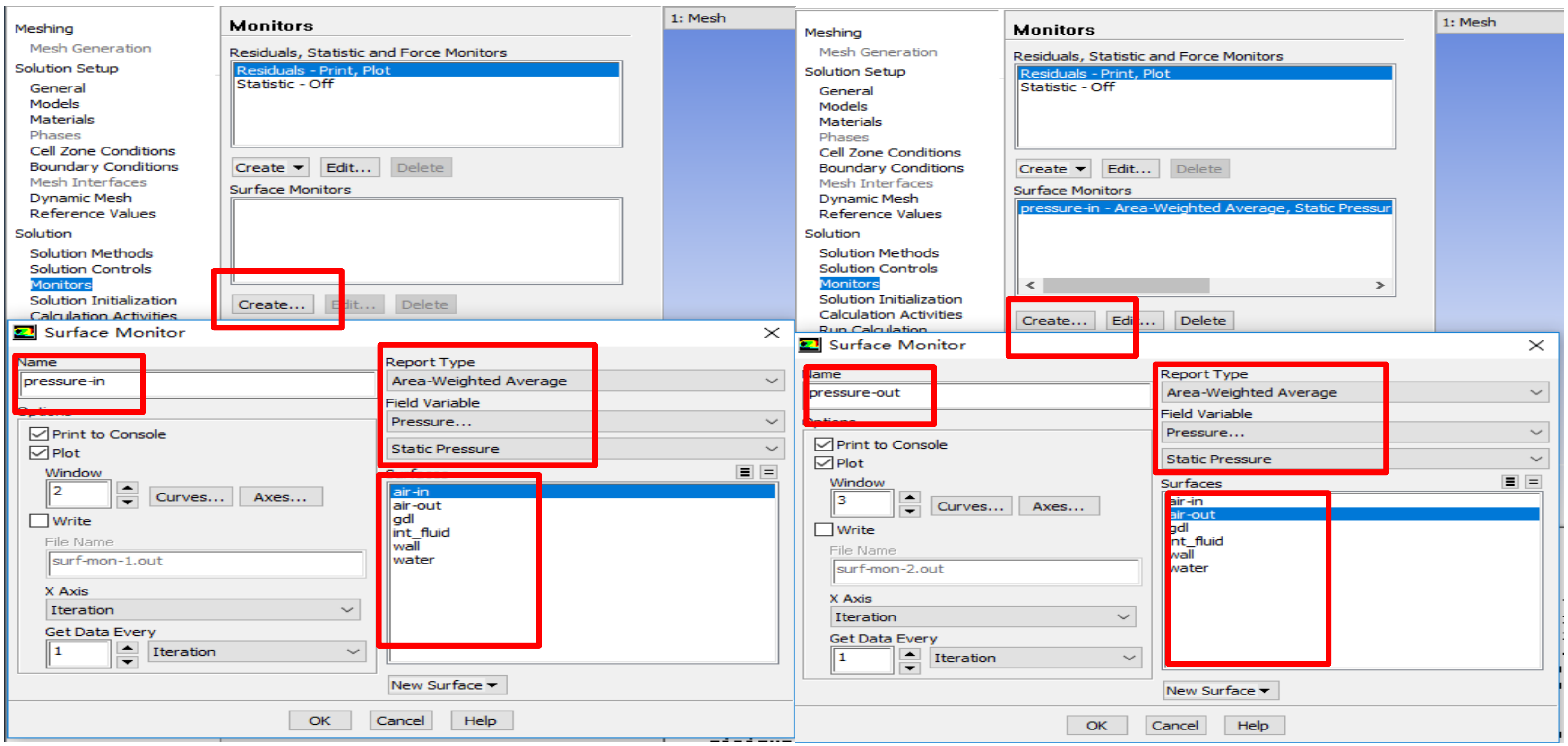
Solution → Monitors

- Define the Residuals Monitor and write 0.0001 in the Absolute Criteria box



2.12 Define the monitors

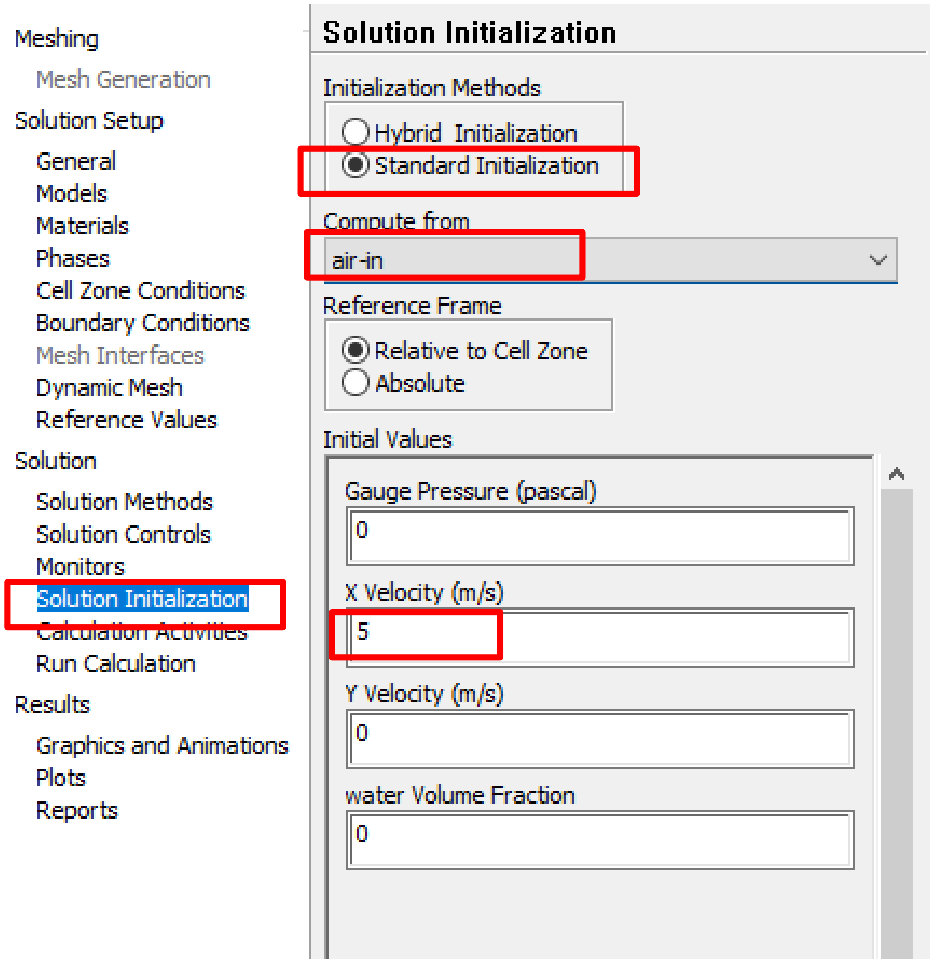
■ Create the Surface Monitor



1. Change the Name
2. Choose the Report Type and Surface

2.13 Initialize

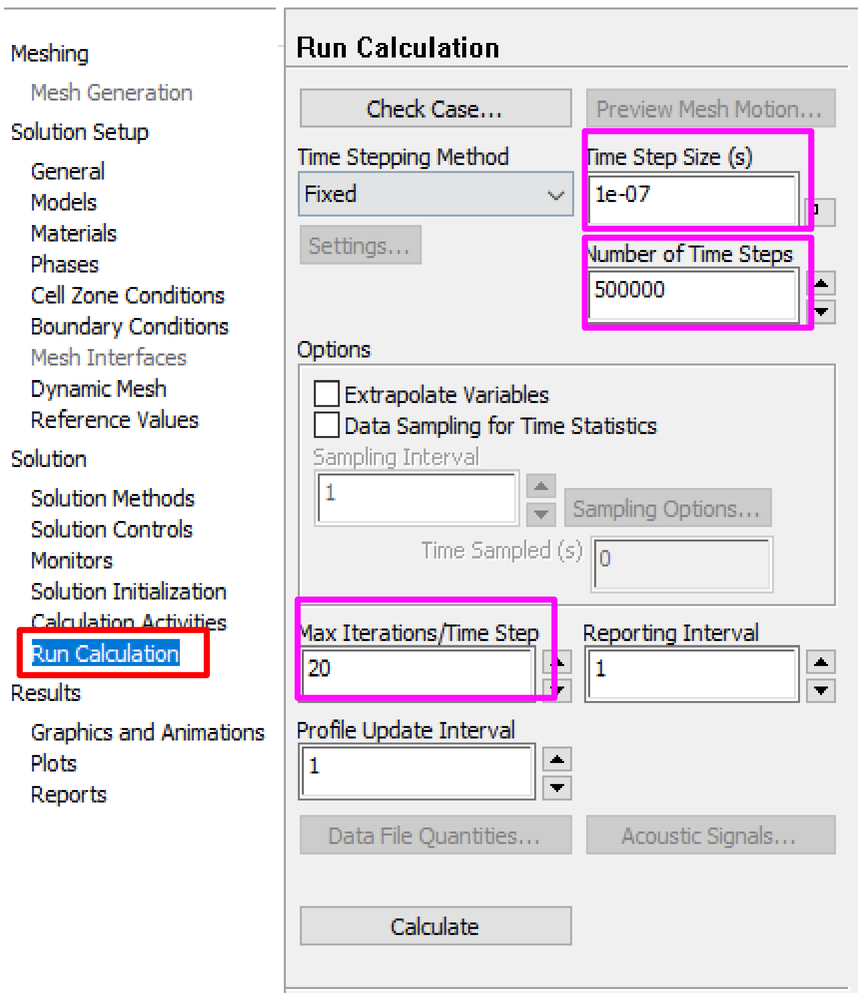
Solution → Solution Initialization



- Choose **Standard Initialization**
- Choose **air-in**
- Click **Initialize**

2.14 Run calculation

Solution → Run Calculation



- Write **Number** in Time Step Size box
- Write **Number** in Number of Time Step box according to situations
- Write **Number** in Max Iterations according to situations

2.14 Run calculation

- **Time Step Size(s)** sets the magnitude of the (physical) time step Δt. **Courant number < 1 should be satisfied.**
- **Number of Time Steps** sets the number of time steps to be performed.
- **Max Iterations/Time Step** sets the maximum number of iterations to be performed per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step.

Summary

1.Read mesh

2.Check mesh

3.Scale domain

4.Choose solver (transient, gravity)

5.Choose model (VOF, explicit, courant number, implicit body force)

6.Define material (water-liquid)

7.Define phase (primary phase, secondary phase, interaction, surface tension, wall adhesion)

8.Denfine operating condition (reference pressure location, specified operating density)

9.Define boundary condition (volume fraction, contact angle)

10.Choose solution method (PISO, Green-Gauss cell based, Body force weighted)

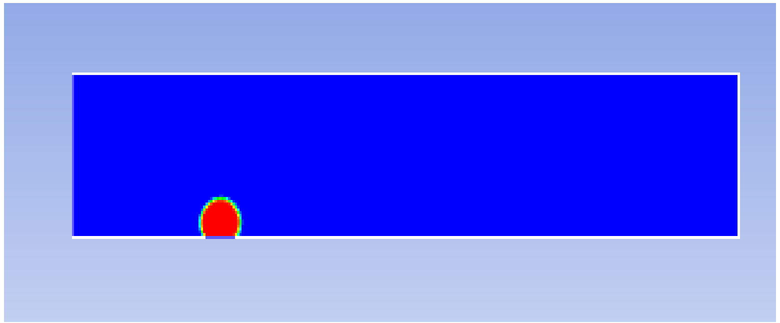
11.Define monitor (residuals monitor, surface monitor)

12.Initailize (standard initialization)

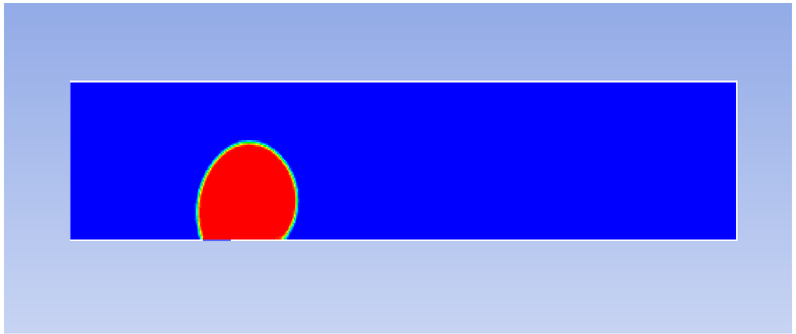
13.Run the simulation (time step size)

14.Post-process

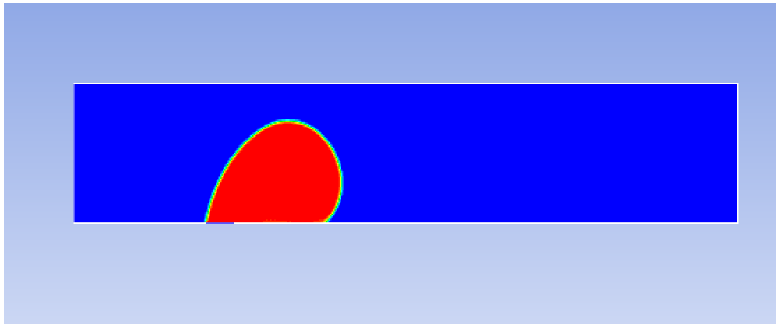
3 Results



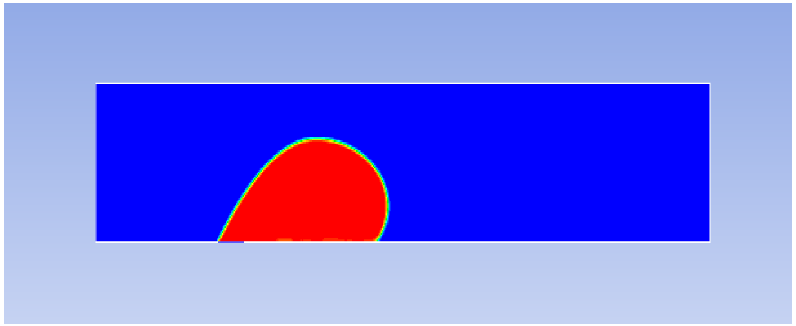
T=8.868e-4



T=5.7064e-3



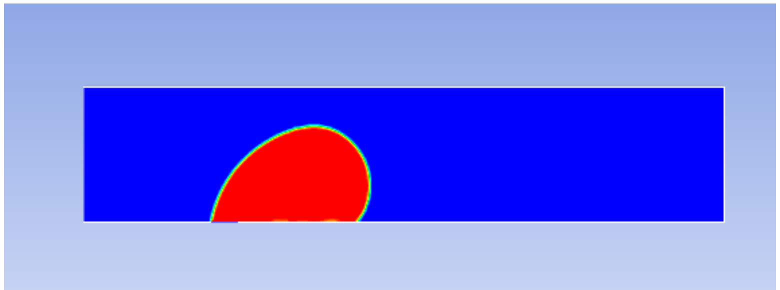
T=8.5285e-3



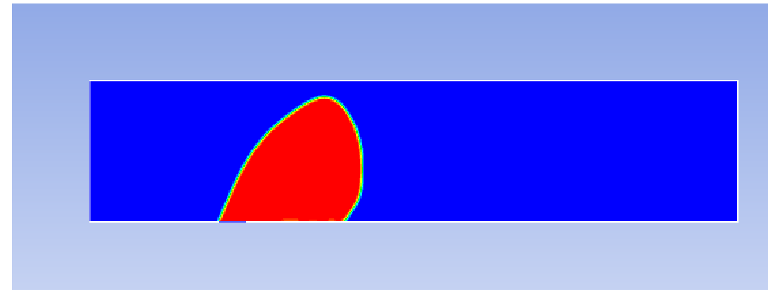
T=9.9285e-3

Fig.2 water behavior

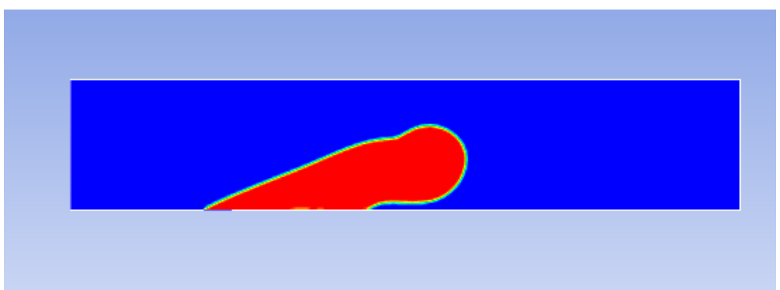
3 Results



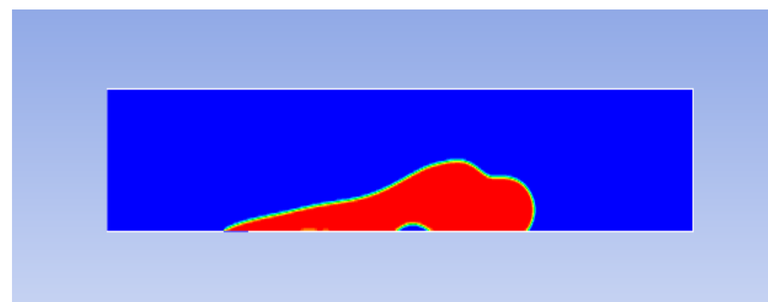
T=1.1028e-2



T=1.0328e-2

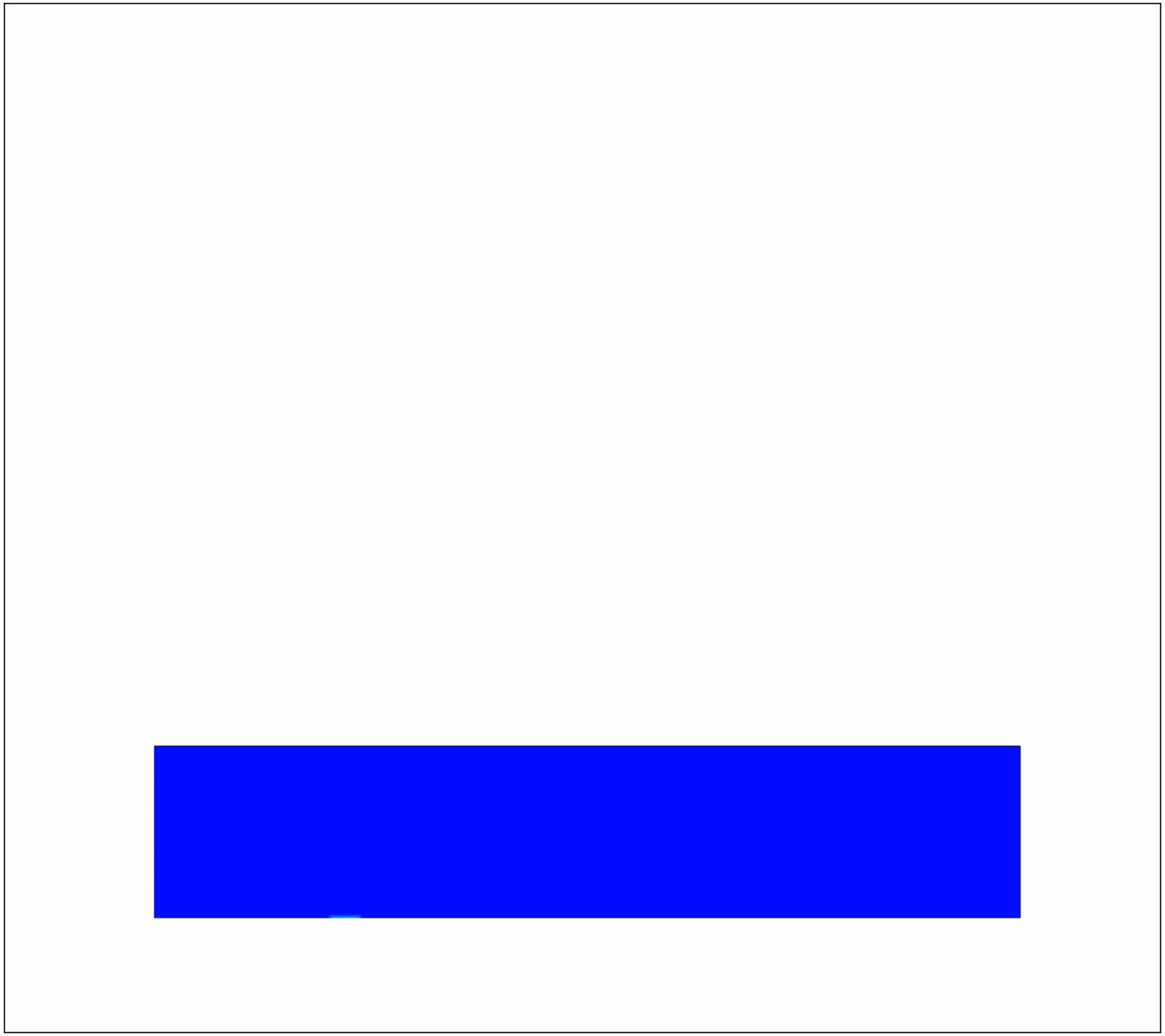


T=1.0529e-2

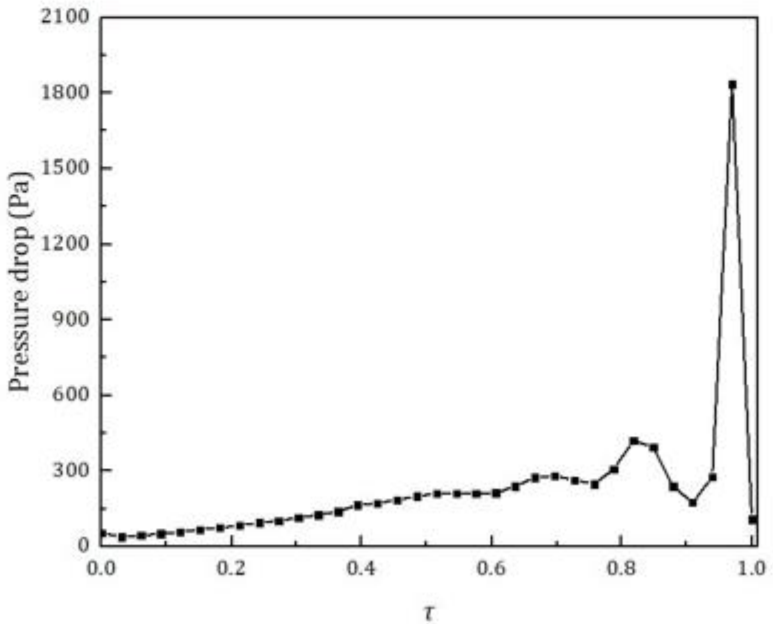


T=1.0729e-2

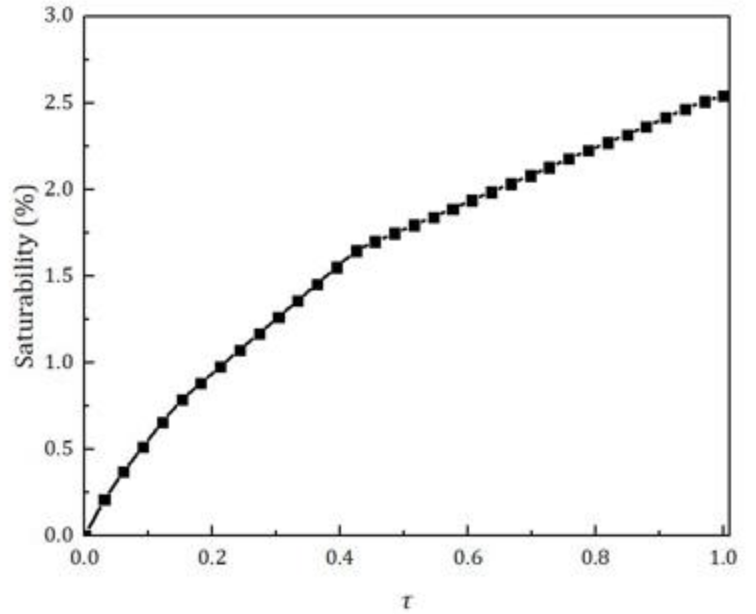
Fig.2 water behavior



Pressure drop

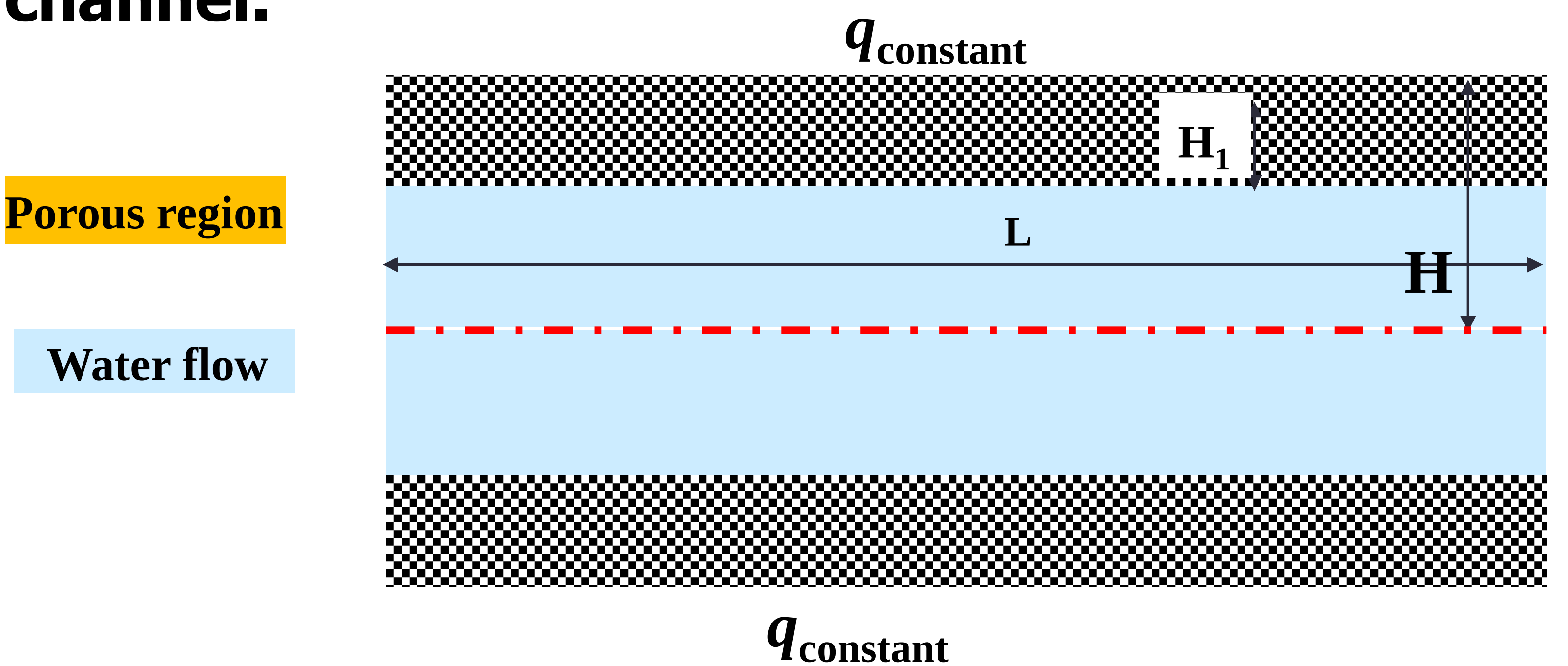


Saturation



Homework

Known: Porous medium is partially inserted into an 2D channel to enhance the heat transfer. The fluid is water and the porous region is aluminum metal foam. The porous region is attached to the outer wall of the channel.



Computational domain

Parameter	H	L	ρ_{water}	η_{water}	$c_{p\ water}$	λ_{water}	ε
Value (Unit)	0.06 (m)	3 (m)	998.2 $(kg \cdot m^{-3})$	998×10^{-6} $(kg \cdot m^{-1} \cdot s^{-1})$	4182 $(J \cdot kg^{-1} \cdot K^{-1})$	0.597 $(W \cdot m^{-1} \cdot K^{-1})$	0.9

Assumptions:

constant physical properties, steady, laminar flow

Boundary conditions

Boundary	Condition
x=0	$v_x = U_{in}$, T=300 K
x=L	p=0 Pa , $T_{backflow}$=300 K
$y=0$	symmetry
y=H	$v_x = v_y = 0$ $q = 500 Wm^{-2}$
Porous region	Thermal-equilibrium model

Solve:

$$PN = \frac{Nu / Nu_{\text{base case}}}{\Delta p / \Delta p_{\text{base case}}}$$

Simulate the heat transfer and laminar flow based on above conditions. Analyze the *Nu*, pressure drop (ΔP) and PN at different *Re* and permeabilities (every one in the same group will run totally 16 cases). Thermal-equilibrium model is adopted for porous medium region. KC equation is adopted to calculate the permeability.

$$Re = \frac{\rho u_{\text{inlet}} \mathrm{H}}{\mu} \qquad k = \frac{D_{\mathrm{p}}^{\ 2} \varepsilon^{3}}{150(1-\varepsilon)^{2}}$$

Traffic control in Xi'an

In each working day, cars with two values of tail number are not allowed on road.

Monday: 1, 6

Tuesday: 2,7

Wednesday: 3,8

Thursday: 4,9

Friday: 5,0

Homework

The homework is divided into 5 groups based on different tail numbers of student ID.

Tail numbers of student ID: 1, 6

Group 1	Thickness of porous region H_1
	0.02 (m)

Re / D_p	Re=100	Re=200	Re=300	Re=400
0.0025	0.0025	0.0025	0.0025	0.0025
0.00079	0.00079	0.00079	0.00079	0.00079
0.00025	0.00025	0.00025	0.00025	0.00025
0.000079	0.000079	0.000079	0.000079	0.000079

Base case

D_p is the diameter of particles

Tail numbers of student ID: 2, 7

Group 2	Thickness of porous region H_1
	0.03 (m)

Re / D_p	Re=100	Re=200	Re=300	Re=400
0.0025	0.0025	0.0025	0.0025	0.0025
0.00079	0.00079	0.00079	0.00079	0.00079
0.00025	0.00025	0.00025	0.00025	0.00025
0.000079	0.000079	0.000079	0.000079	0.000079

Base case

D_p is the diameter of particles

Tail numbers of student ID: 3, 8

Group 3	Thickness of porous region H_1
	0.04 (m)

Re / D_p	Re=100	Re=200	Re=300	Re=400
0.0025	0.0025	0.0025	0.0025	0.0025
0.00079	0.00079	0.00079	0.00079	0.00079
0.00025	0.00025	0.00025	0.00025	0.00025
0.000079	0.000079	0.000079	0.000079	0.000079

Base case

D_p is the diameter of particles

Tail numbers of student ID: 4, 9

Group 4	Thickness of porous region H_1
	0.05 (m)

D_p \ Re	Re=100	Re=200	Re=300	Re=400
0.0025	0.0025	0.0025	0.0025	0.0025
0.00079	0.00079	0.00079	0.00079	0.00079
0.00025	0.00025	0.00025	0.00025	0.00025
0.000079	0.000079	0.000079	0.000079	0.000079

Base case

D_p is the diameter of particles

Tail numbers of student ID: 5, 0

Group 1	Thickness of porous region H_1
	0.055 (m)

Re / D_p	Re=100	Re=200	Re=300	Re=400
0.0025	0.0025	0.0025	0.0025	0.0025
0.00079	0.00079	0.00079	0.00079	0.00079
0.00025	0.00025	0.00025	0.00025	0.00025
0.000079	0.000079	0.000079	0.000079	0.000079

Base case

D_p is the diameter of particles

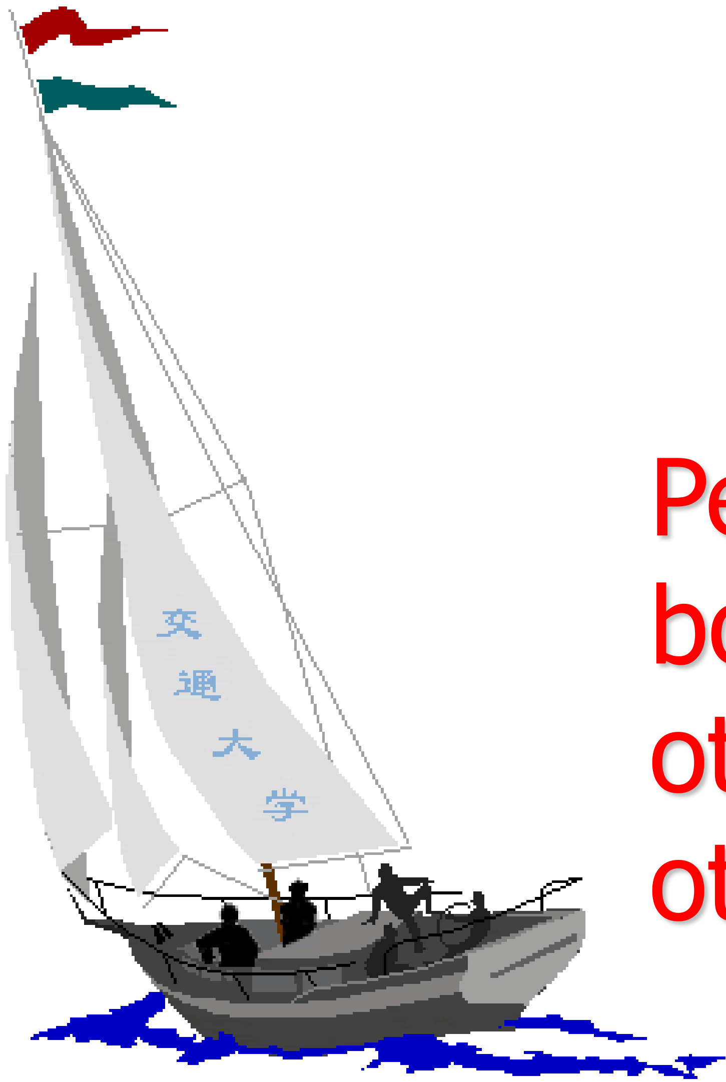
感谢各位同学

感谢陶老师！

Happy New Year

同舟共济渡彼岸!

People in the same boat help each other to cross to the other bank, where….



同舟共济
渡彼岸！

People in the same boat help each other to cross to the other bank, where….