

Numerical Heat Transfer

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



Li Chen, Wen-Quan Tao
CFD-NHT-EHT Center

Key Laboratory of Thermo-Fluid Science & Engineering, XJTU
Xi'an, 2021-Dec.-29

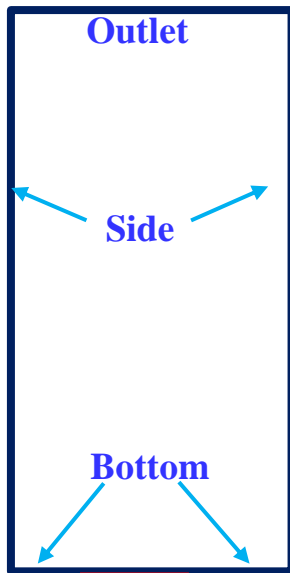


“西”望你我，“安”然无恙

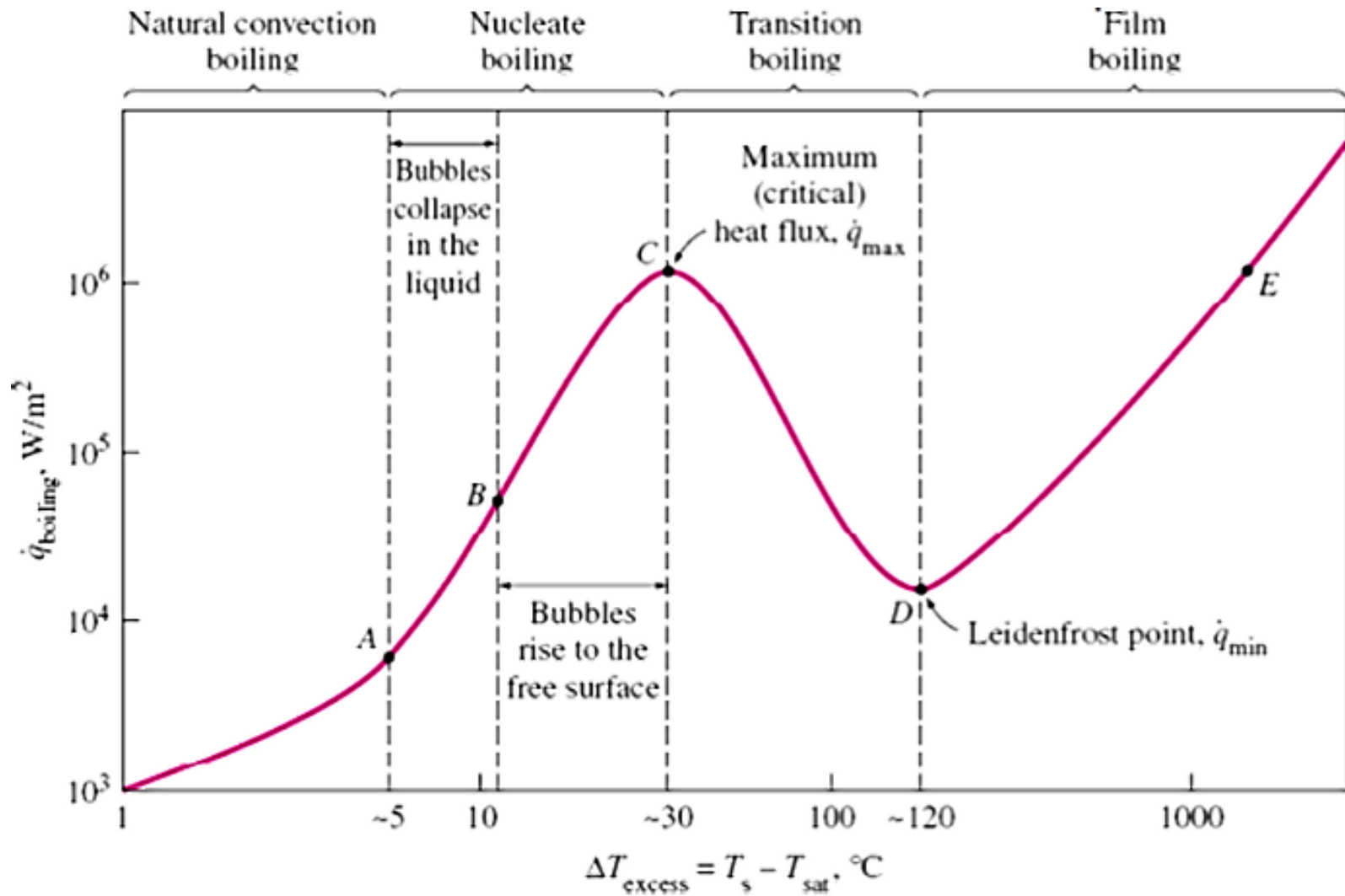
Hope Everyone Safe and Sound

13.3 Pool boiling heat transfer

Problem descriptions: A pool is filled with liquid water. Part of the bottom wall is heated with constant temperature. Phase change takes place and bubbles will be generated, leading to the pool boiling heat transfer process.



	Boundary conditions
Side walls	Symmetry
Outlet	Pressure outlet
Bottom: constant temperature	Wall, 573.15K , 10°
Bottom: adiabatic	Wall, adiabatic, 120°



S. Nukiyama, The maximum and minimum values of the heat Q transmitted from metal to boiling water under atmospheric pressure. Int. J. Heat Mass Transf., 9 (12) (1934), pp. 1419-1433

- **Find:** bubble dynamic behaviors during the pool boiling.

Solution:

Continuity, momentum and energy equation for pooling boiling heat transfer?

The governing equations for pool boiling phase change heat transfer is a little different from the original NS equation. Equations for multiphase flow is required. We will study background information of multiphase flow and then derive the equations.

1). Background of Multiphase flow

Multiphase fluid flows are ubiquitous 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) :

Liquid water and water vapor system

Multiple components (多组分多相) :

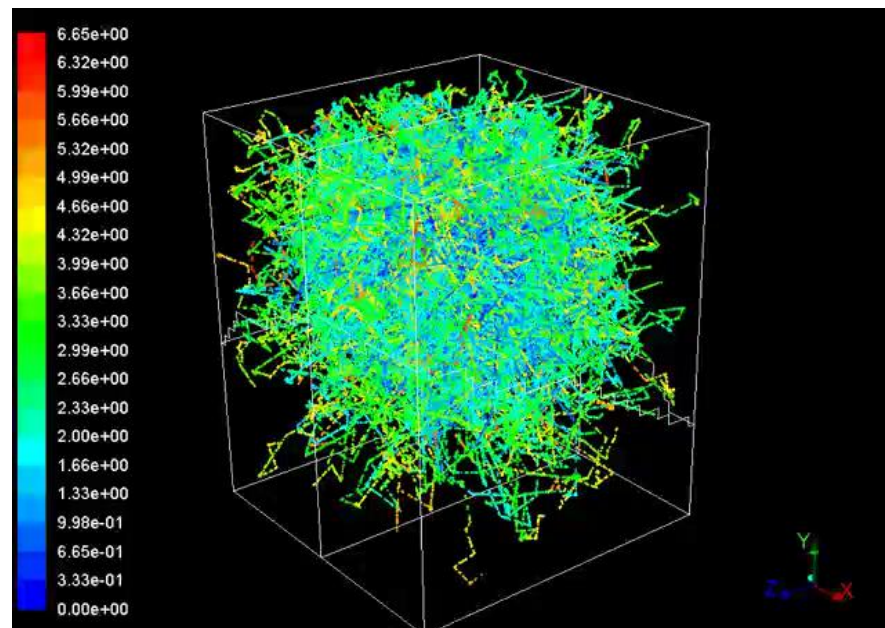
Small density: water and oil system

Large density ratio: water and air system

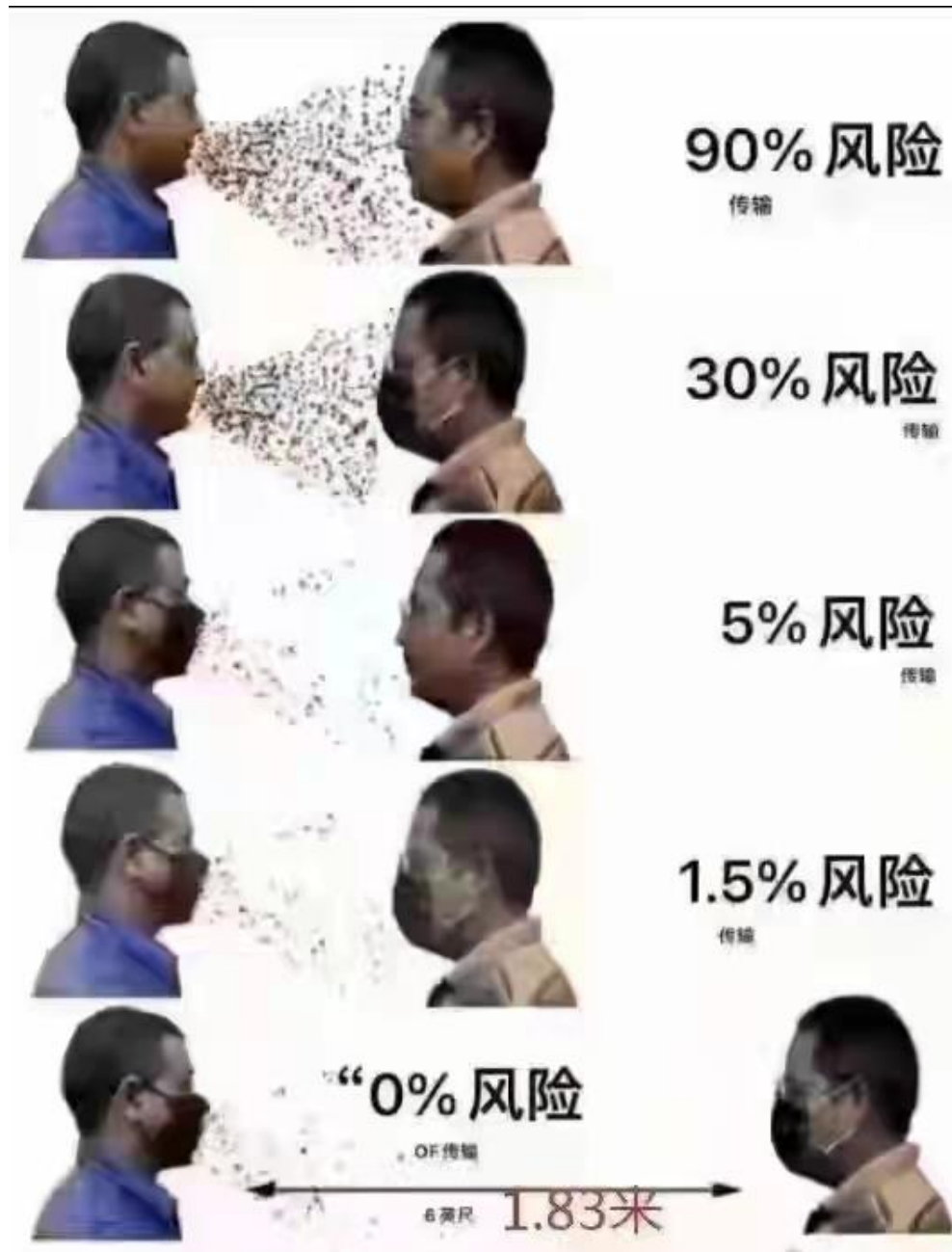


Crown

Multiphase flow during cough



Droplet spread(飞沫传播) , provided by PhD Meng-Yi Wang from NHT group



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



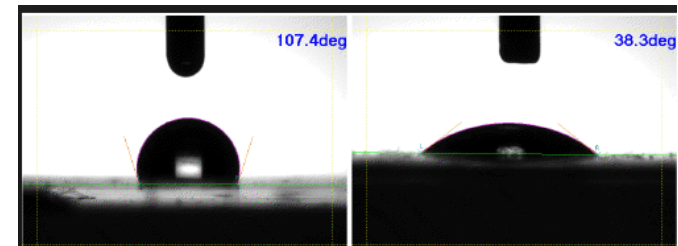
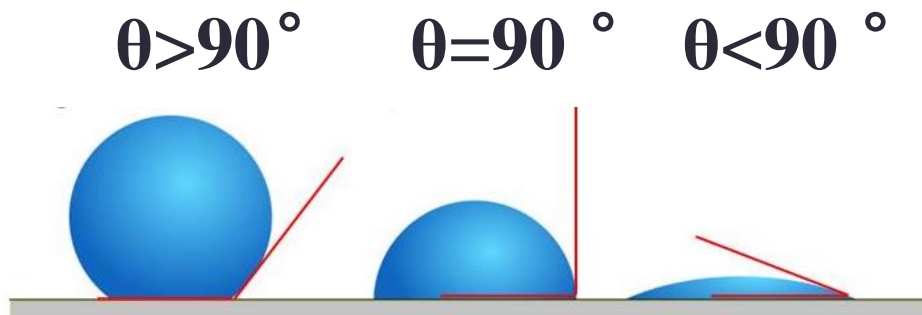
Water striders stay on top of water
(水面上的水黽)



Lotus effect (荷叶效应)

Contact angle

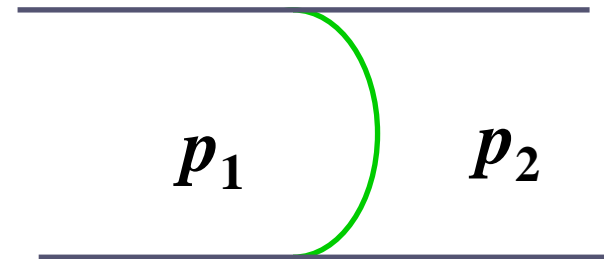
measurement of the surface wettability. The angle of the triple-phase line. **Hydrophilic surface (亲水)** with angle less than 90, liquid tends to spread. **Hydrophobic surface (疏水)** with angle higher than 90, liquid tends to form droplet. **Neutral surface (中性表面)** with angle as 90.



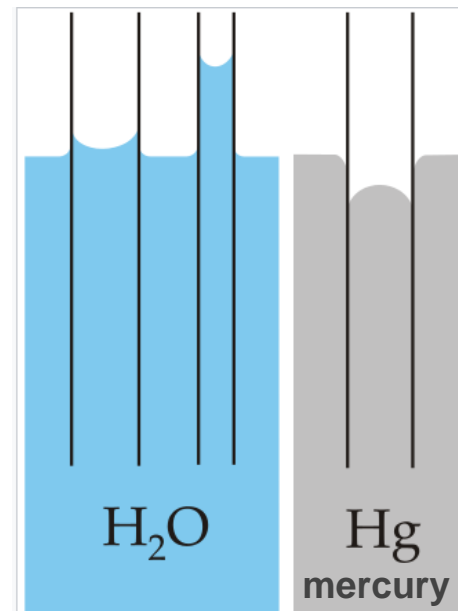
Capillary pressure

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.



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

3). Different methods for multiphase flow

Macroscopic

Volume of Fluid (VOF) 流体体积法

Level Set (LS) 水平集法

Phase-field 相场方法

Front tracking 前沿跟踪方法



VOSET

by NHT group

Mesososcopic

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)

4). 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*

C. W. HIRT AND B. D. NICHOLS

Los Alamos Scientific Laboratory, Los Alamos, New Mexico 87545

Received November 1, 1979

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 ...

☆ 被引用次数: 12539 相关文章 所有 17 个版本

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

The volume fraction of each fluid in a computational cell

$$C_m = \frac{V_m}{V_{\text{cell}}}$$

$$\sum C_m = 1$$

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

$$C_1 = 1$$

The cell is filled with the primary phase

$$C_1 = 0$$

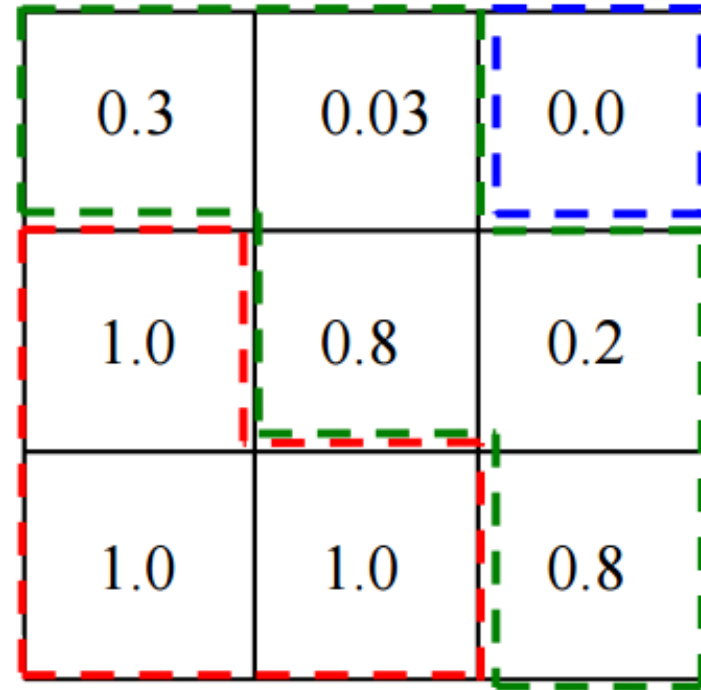
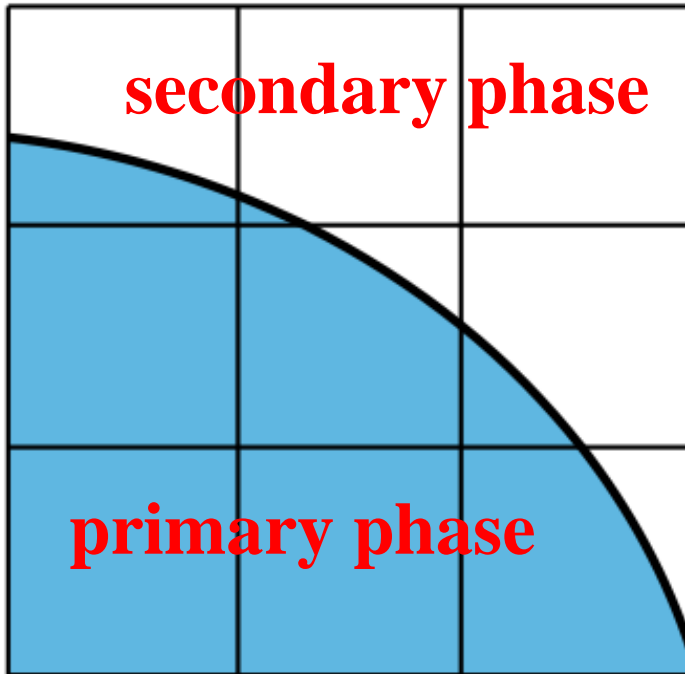
The cell is free of primary phase

$$C_1 \in (0, 1)$$

The cell is partially filled with primary phase

Schematic of 2D two-phase flow system

C is a scalar.



$$C_1 = 1$$

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

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

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 single phase flow with VOF

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

Surface tension force

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

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

$$\rho = C_1 \rho_1 + C_g \rho_g \quad \mu = C_1 \mu_1 + C_g \mu_g$$

$$\mathbf{F} = 2\sigma k \frac{\rho \nabla C_1}{(\rho_1 + \rho_g)}$$

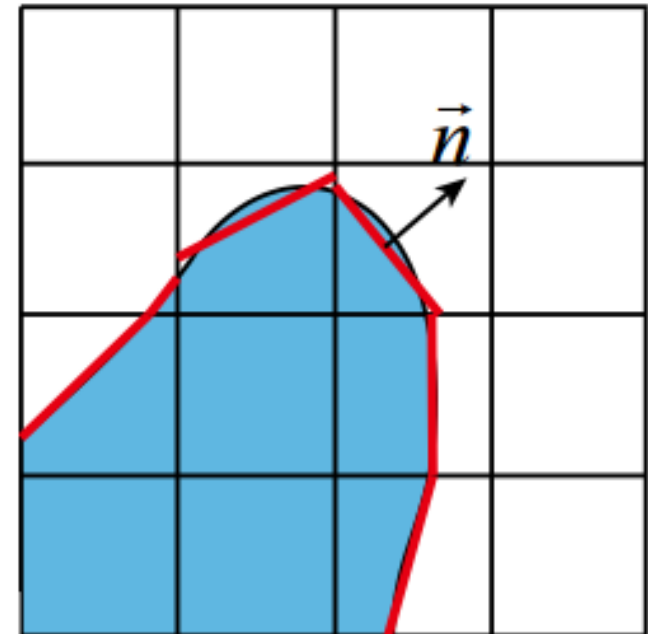
Two-way coupled with each other

Continuum surface force (CSF) model

The form of volumetric force (体积力, N/m^3) is required in NS equation. However, surface tension force is a kind of surface force, rather than volumetric force.

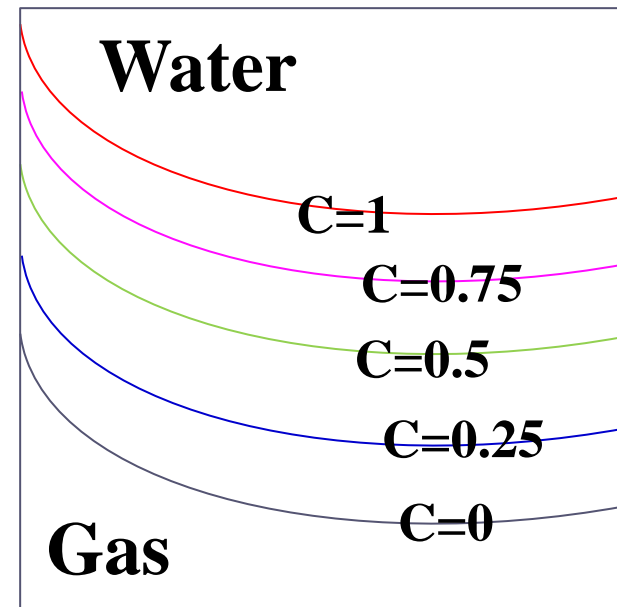
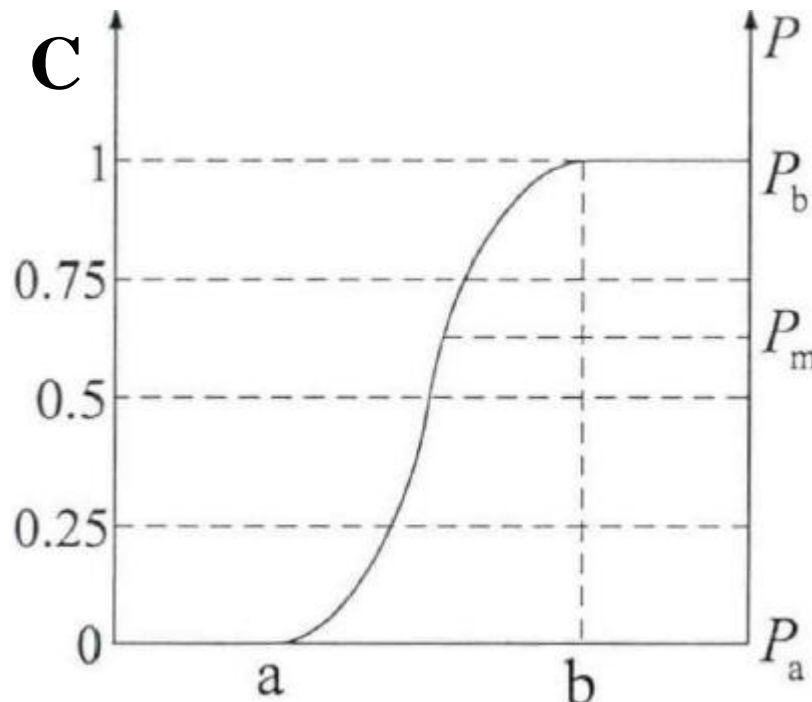
✓ Smooth C

- VOF in fact is a sharp-interface model.
- The thickness of the interface is zero.
- The fluid property 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

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

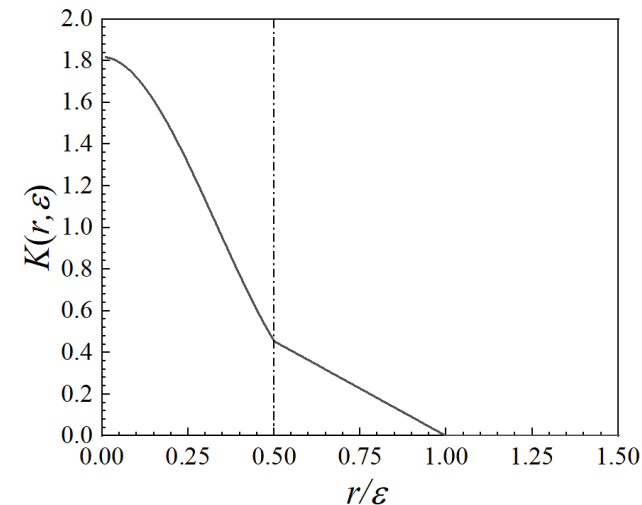
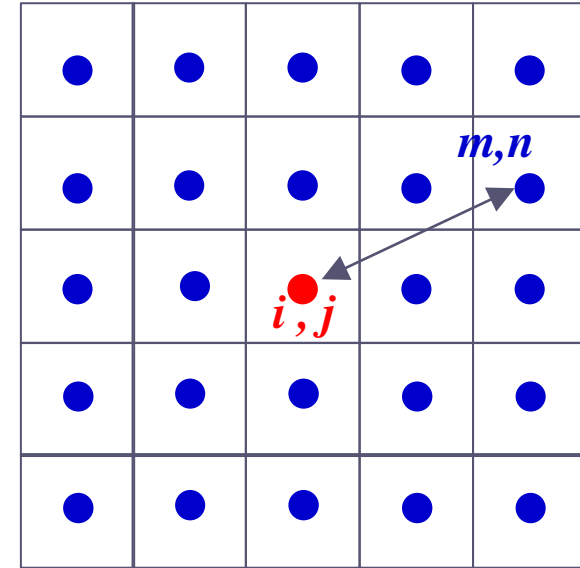
ε
Control the thickness of the interface! 3Δ

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

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

K Smooth integration kernel

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



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

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

interface mean curvature

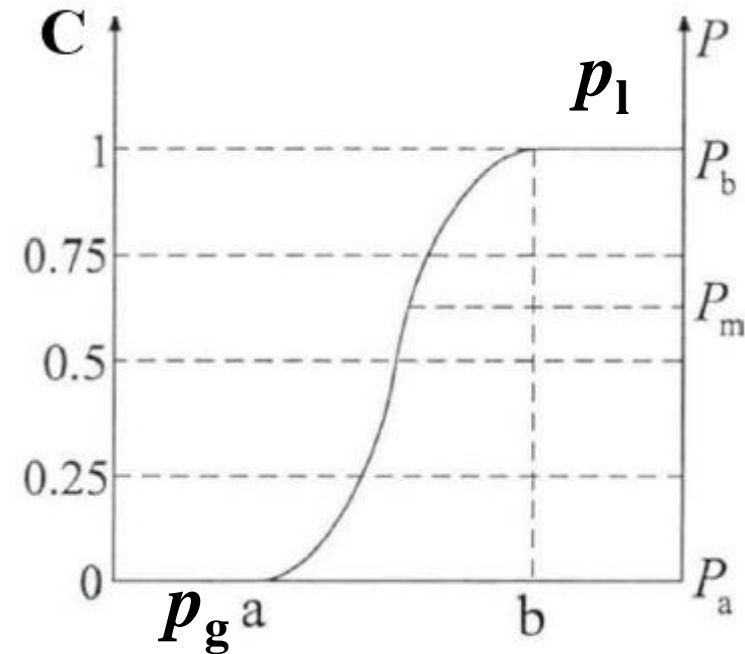
$$k = \nabla \cdot \left(\frac{\nabla \tilde{C}_1}{|\nabla \tilde{C}_1|} \right)$$

pressure in the transition region is

$$P_x = P_g + \sigma k (C_x - C_g) = P_g + \sigma k C_x$$

$$\begin{aligned} \mathbf{F} &\sim \nabla (P_x - P_g) = \nabla (\sigma k (C_x - C_g)) \\ &= \sigma k \nabla C \end{aligned}$$

Suppose local k is constant.

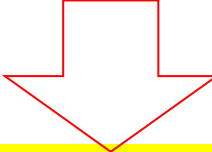


$$\mathbf{F} = 2\sigma k \frac{\rho \nabla C_1}{(\rho_1 + \rho_g)}$$

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 introduced in NHT.

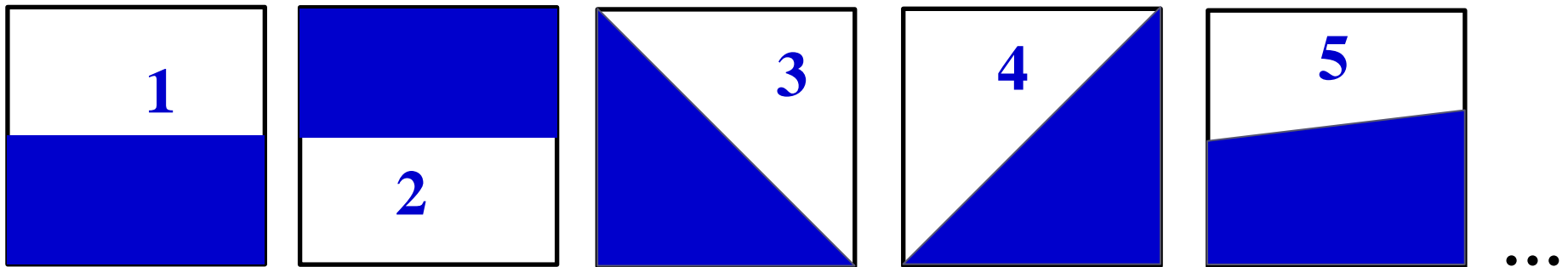
However, because C is not a continuous function. Such method may result in false diffusion, leading to gradually increasing thickness of the interface.

(2) . Reconstruction method

Step 1. Interface reconstruction

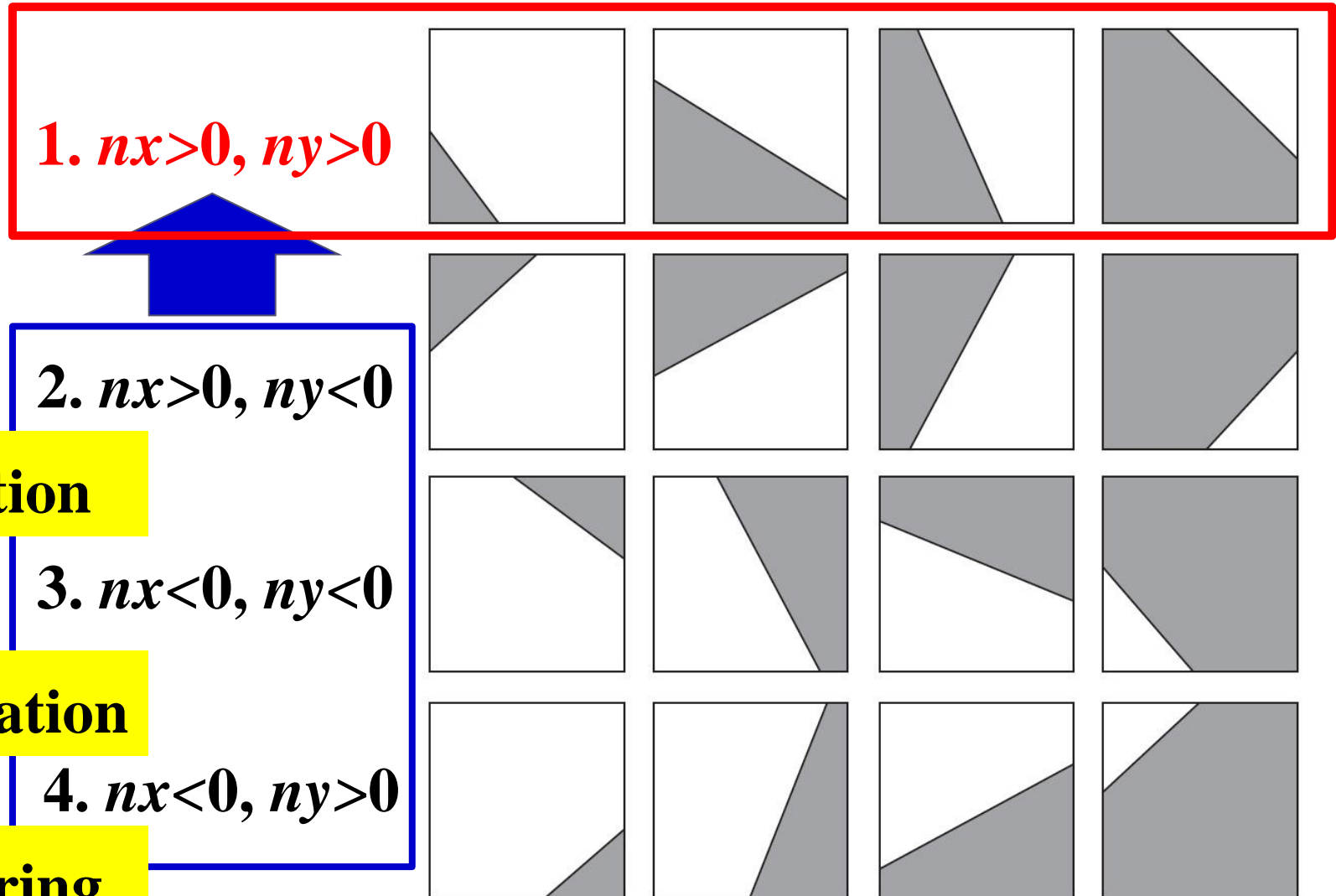
For a value of C in a computational cell, the pattern of interface should be determined first.

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

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

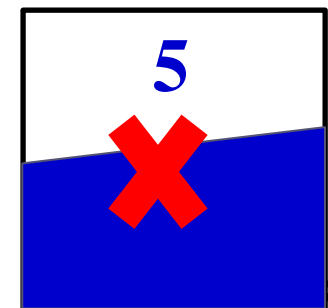
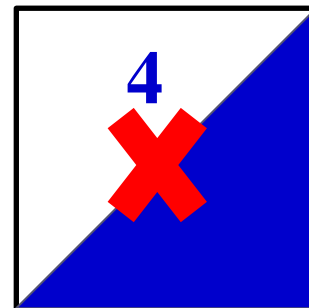
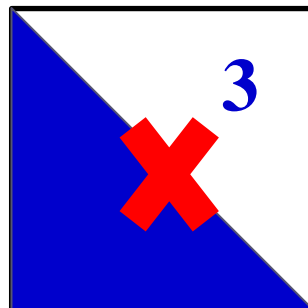
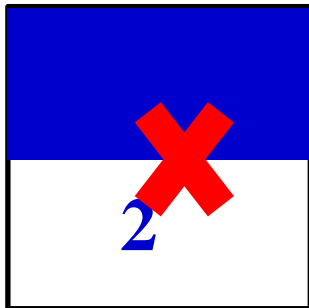
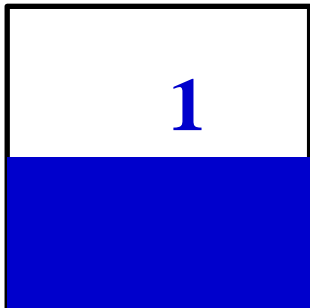
$$n_{i,j}^y = (\tilde{C}_{i+1,j+1} + 2\tilde{C}_{i,j+1} + \tilde{C}_{i-1,j+1} - \tilde{C}_{i+1,j-1} - 2\tilde{C}_{i,j-1} - \tilde{C}_{i-1,j-1}) / \delta y$$

Interface normal direction

Volume of fraction

Interface is reconstructed!

For example, for $C=0.5$, $n_x=0$, $n_y=1$



Normal direction of the interface

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

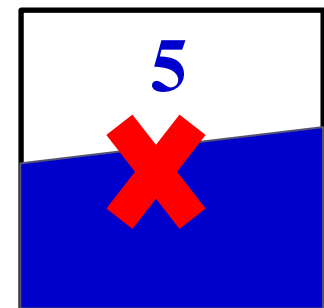
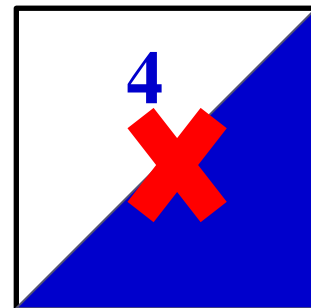
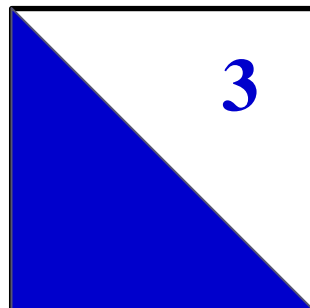
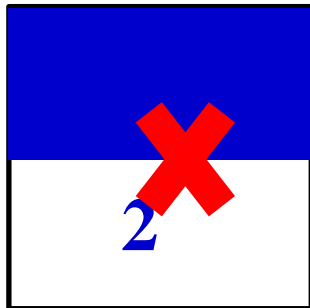
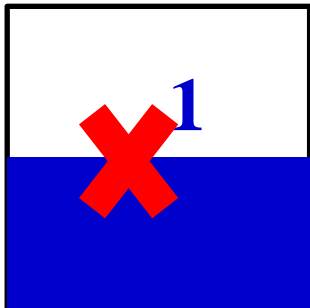
$$n_{i,j}^y = (\tilde{C}_{i+1,j+1} + 2\tilde{C}_{i,j+1} + \tilde{C}_{i-1,j+1} - \tilde{C}_{i+1,j-1} - 2\tilde{C}_{i,j-1} - \tilde{C}_{i-1,j-1}) / \delta y$$

Interface normal direction

Volume of fraction

} Interface is reconstructed!

For example, for $C=0.5$, $n_x = \sqrt{2}/2$, $n_y = \sqrt{2}/2$

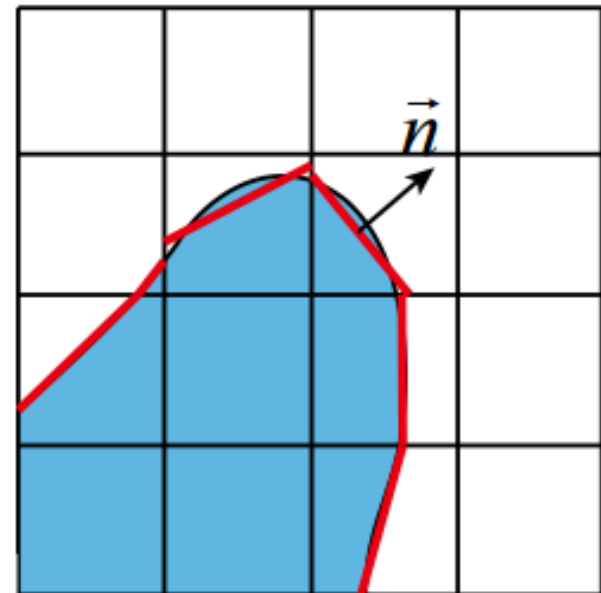


By the reconstruction scheme, the phase interface is determined in each computational cell.

Piecewise linear interface calculation (PLIC)

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

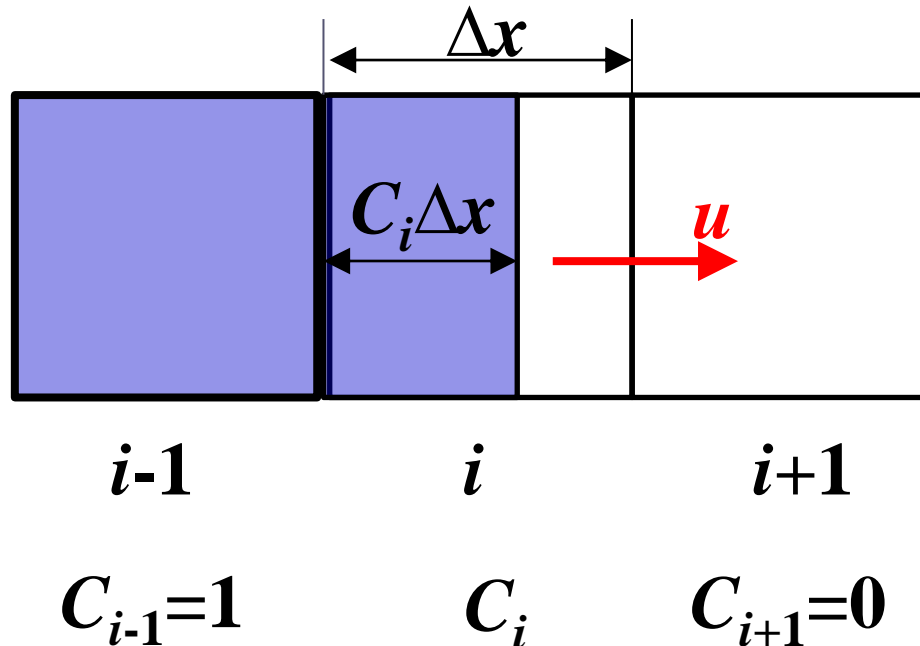
D.L. Youngs, Time-dependent multi-material flow with large fluid distortion, Numerical methods for Fluid Dynamics, 1982, 24(2), 273-285

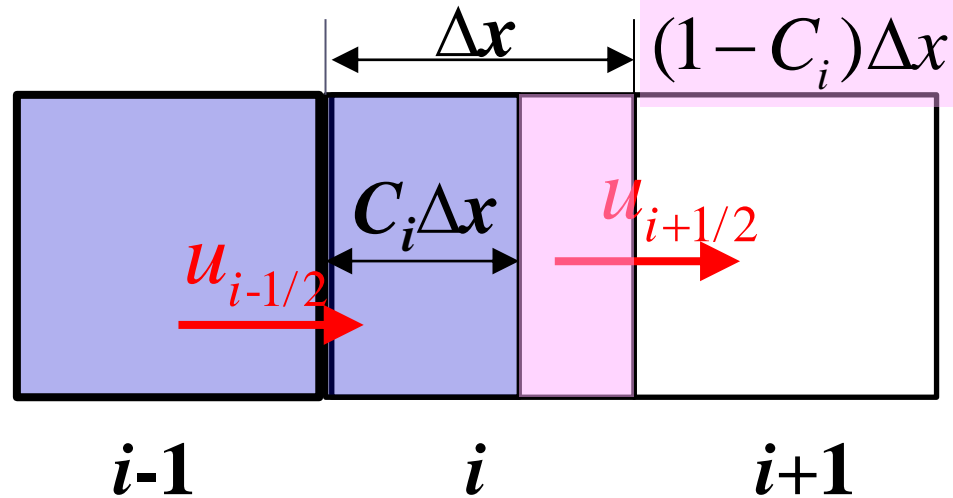


Step 2. Interface advection in a given velocity field

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

Taking 1D interface as example.





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

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

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

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

Counrant 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 Counrant number < 1** .

5. Phase-change model: Lee Model

$$\frac{\partial C_1}{\partial t} + \mathbf{u} \cdot \nabla C_1 = \frac{S_1}{\rho}$$

$$S_v = -S_1 = \alpha_g \rho_g \frac{T - T_{\text{sat}}}{T_{\text{sat}}}$$

for condensation ($T_{\text{sat}} > T$)

$$\frac{\partial C_v}{\partial t} + \mathbf{u} \cdot \nabla C_v = \frac{S_v}{\rho}$$

$$S_v = -S_1 = \alpha_l \rho_l \frac{T - T_{\text{sat}}}{T_{\text{sat}}}$$

for evaporation ($T_{\text{sat}} < T$)

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

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

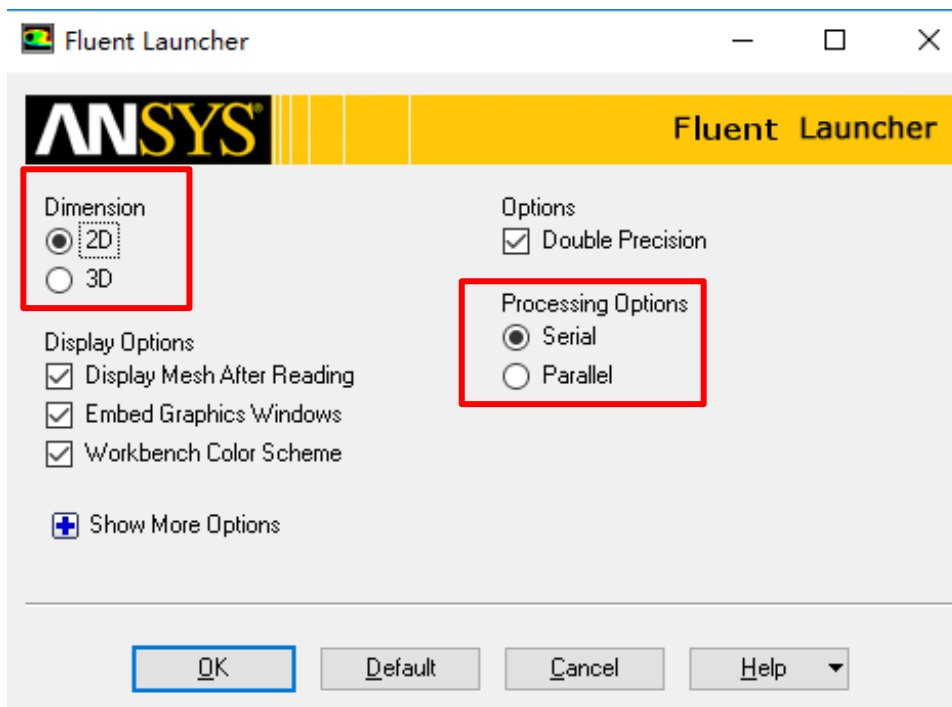
$$\frac{\partial(\rho C_p T)}{\partial t} + (\mathbf{u} \cdot \nabla)(\rho C_p T) = \nabla(\lambda \nabla T) + S_l h$$

$$\frac{\partial C_1}{\partial t} + \mathbf{u} \cdot \nabla C_1 = \frac{S_1}{\rho}$$

$$\frac{\partial C_v}{\partial t} + \mathbf{u} \cdot \nabla C_v = \frac{S_v}{\rho}$$

2 Process of simulation

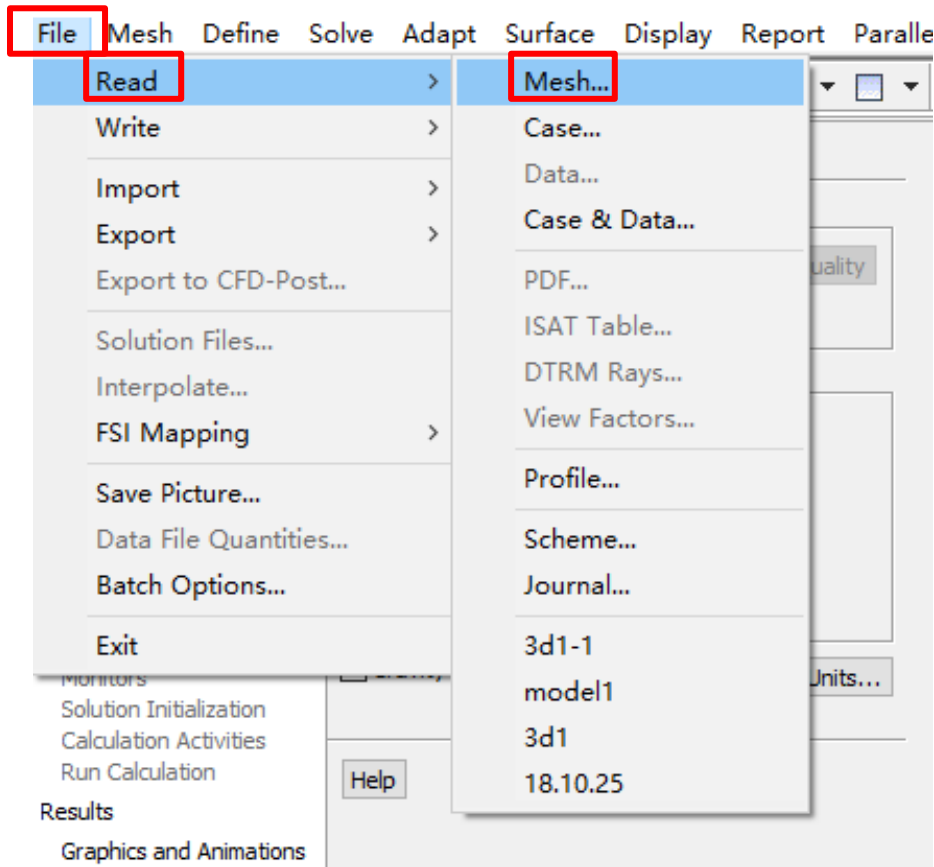
2.1 Launch ANSYS Fluent



- 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
    
```

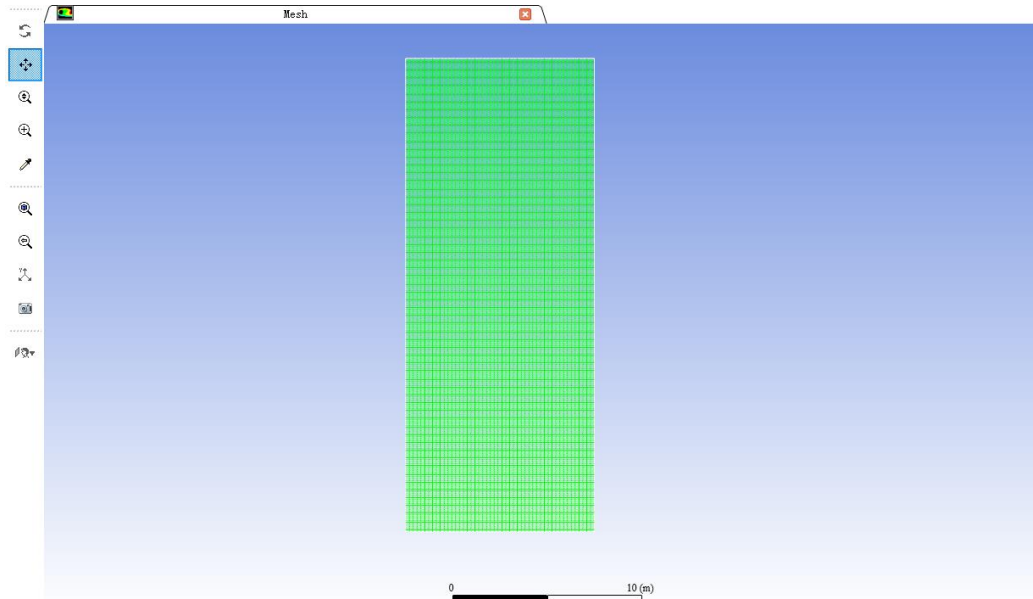
Done.

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:

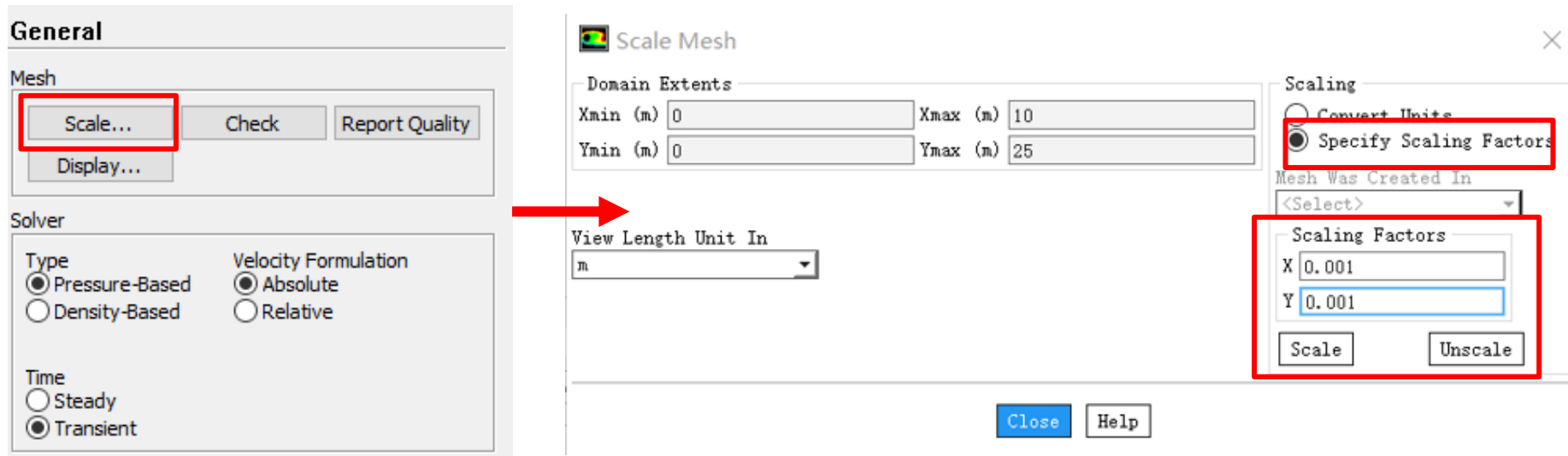
minimum face area (m2): $5.000000e+00$

maximum face area (m2): $5.000000e+00$

Checking mesh.....
Done.

2.4 Scale the domain size

General → **Mesh** → **Scale**

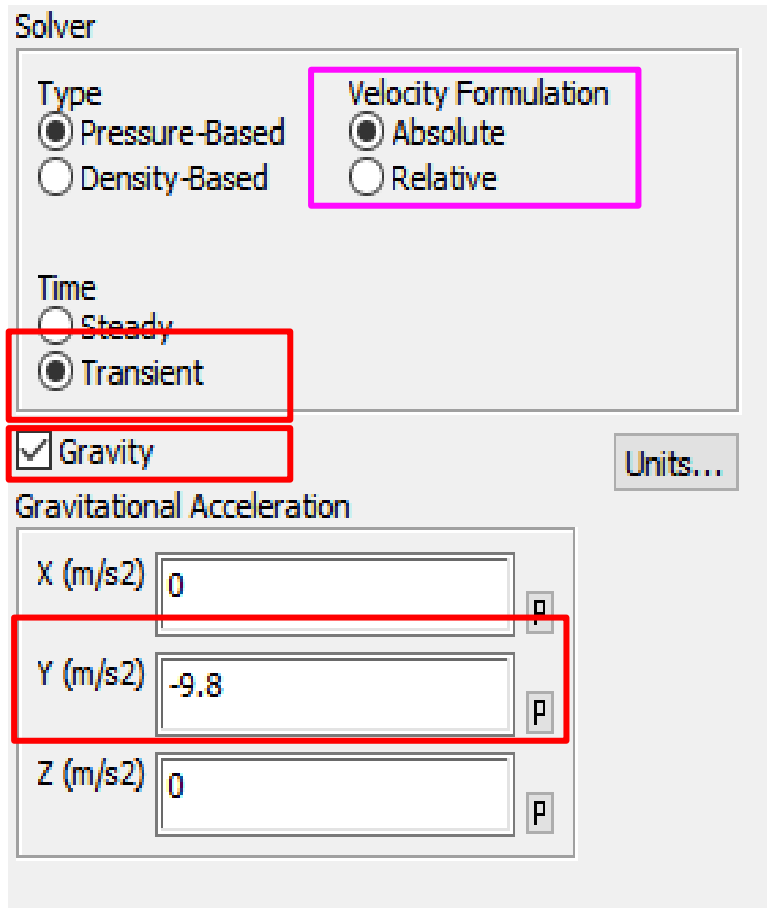


The screenshot shows the ANSYS Fluent interface. On the left, the 'General' panel has the 'Scale...' button highlighted with a red box. A red arrow points from this button to the 'Scale Mesh' dialog box. In the 'Scale Mesh' dialog, the 'Specify Scaling Factors' radio button is selected and highlighted with a red box. The 'Scaling Factors' section shows 'X' and 'Y' both set to 0.001, also highlighted with a red box. The 'View Length Unit In' dropdown is set to 'm'. The 'Domain Extents' section shows Xmin (m) 0, Xmax (m) 10, Ymin (m) 0, and Ymax (m) 25. The 'Mesh Was Created In' dropdown is set to '<Select>'. The 'Close' and 'Help' buttons are visible at the bottom of the dialog.

- Choose **Specify Scaling Factor**
- Convert the unite from **m** to **mm**.

2.5 Choose the solver

General → Solver



Solver

Type

Pressure-Based

Density-Based

Velocity Formulation

Absolute

Relative

Time

Steady

Transient

Gravity

Units...

Gravitational Acceleration

X (m/s²) 0

Y (m/s²) -9.8

Z (m/s²) 0

- Choose **Transient**

The dynamic behaviors of bubble is to be studied.

- Select **Gravity**

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

Density-based method cannot be used for VOF.

The velocity formulation resulting in most of the flow domain having the smallest velocities is recommended, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution.

Velocity Formulation

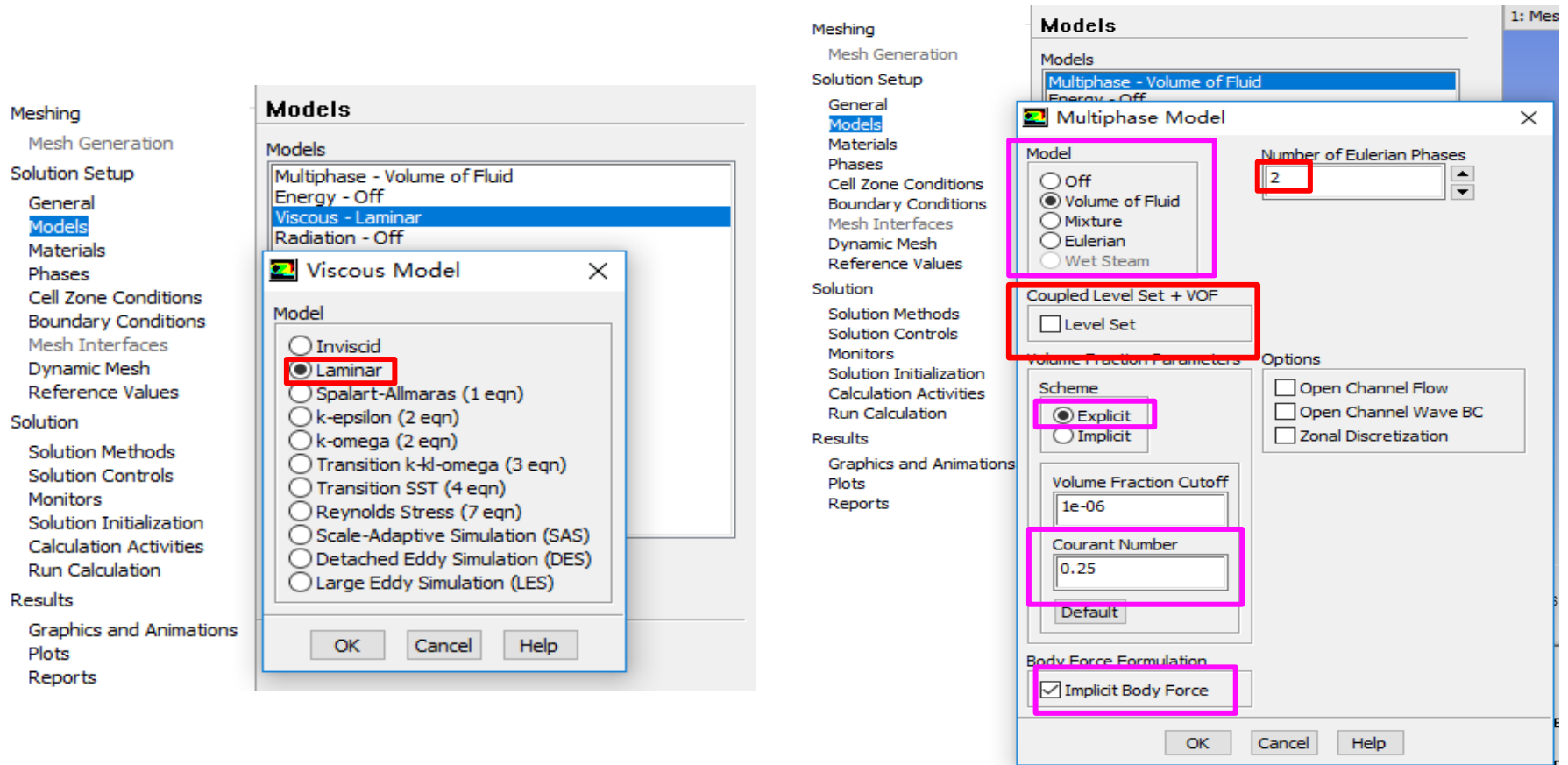
Absolute

Relative

- The **absolute velocity formulation** is preferred in applications where the flow in most of the domain is not moving (for example, a fan in a large room).
- The **relative velocity formulation** is appropriate when most of the fluid in the domain is moving, as in the case of a large impeller in a mixing tank.

2.6 Choose the models

Solution Setup → Models

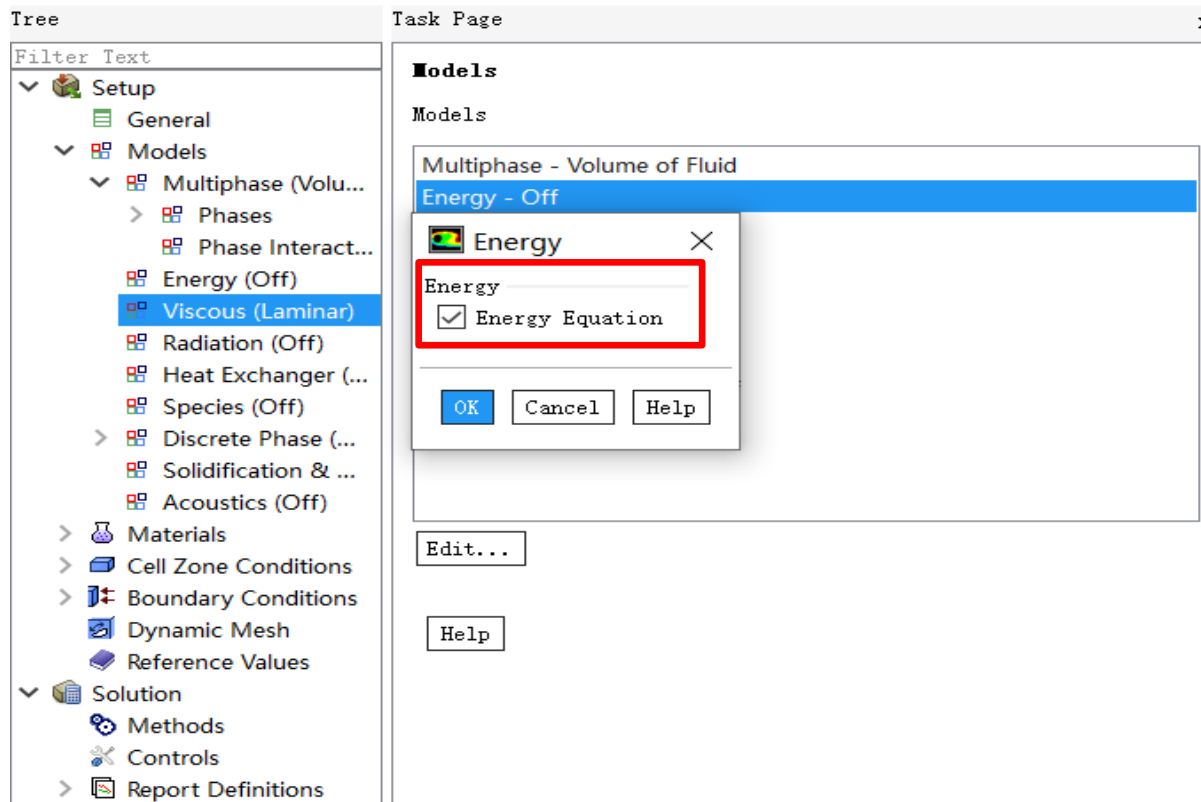


The image shows the ANSYS Fluent Solution Setup - Models dialog box. The 'Viscous Model' dialog is open, showing the 'Laminar' model selected. The 'Multiphase Model' dialog is also open, showing the 'Volume of Fluid' model selected, the 'Number of Eulerian Phases' set to 2, the 'Scheme' set to 'Explicit', the 'Volume Fraction Cutoff' set to 1e-06, the 'Courant Number' set to 0.25, and the 'Implicit Body Force' checkbox checked.

■ Choose **Volume of Fluid** as Multiphase Model

2.6 Choose the models

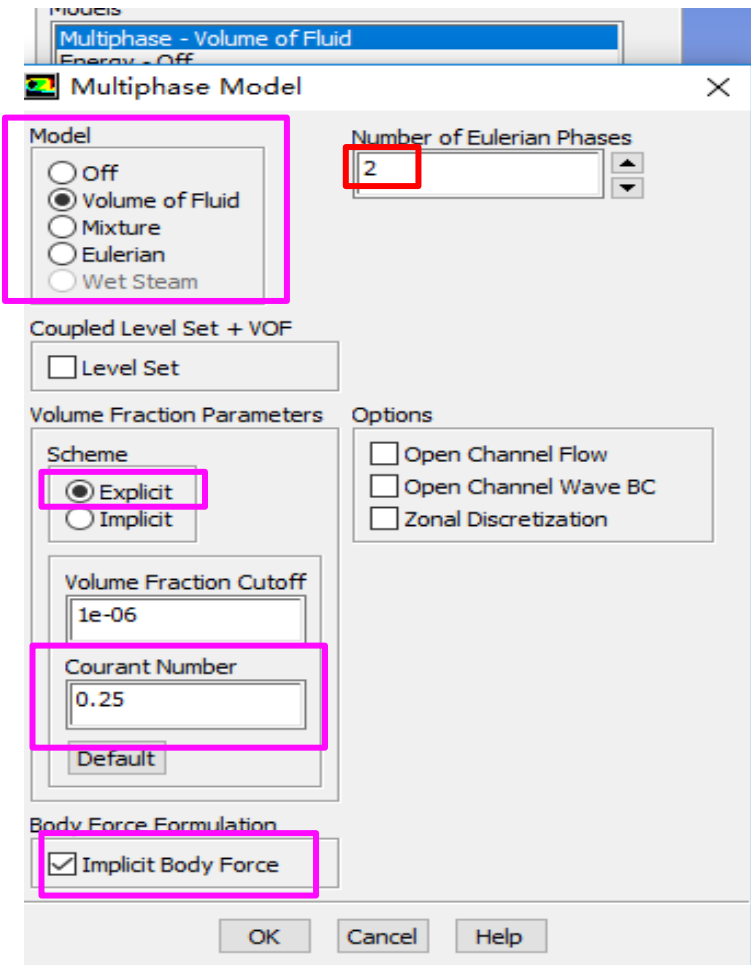
Solution Setup → Models



- Select to open **Energy Equation**

2.6 Choose the models

Body force formulation



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

Body force and pressure gradient are almost in equilibrium.

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

Forces in VOF

Multiphase flow is controlled by a set of forces.

Inertial force

Viscous force

Body force

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot [\mu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \rho \mathbf{g} + 2\sigma k \frac{\rho \nabla C_1}{(\rho_1 + \rho_g)}$$

Pressure

gravity

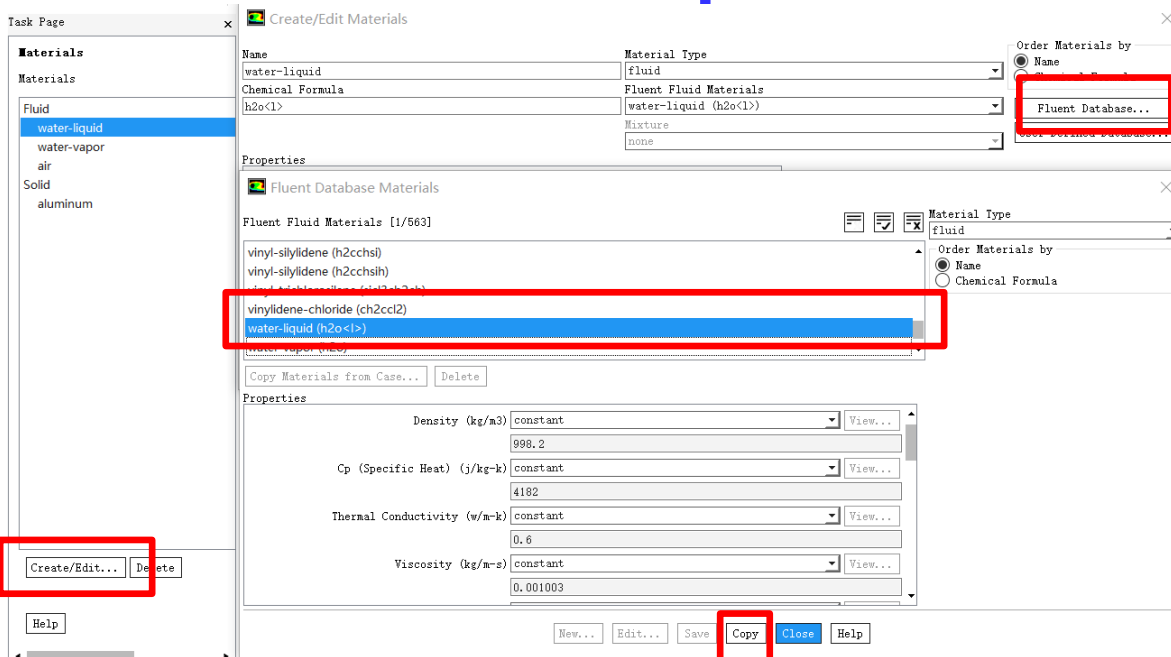
**Surface
tension
force**

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

2.7 Define the materials

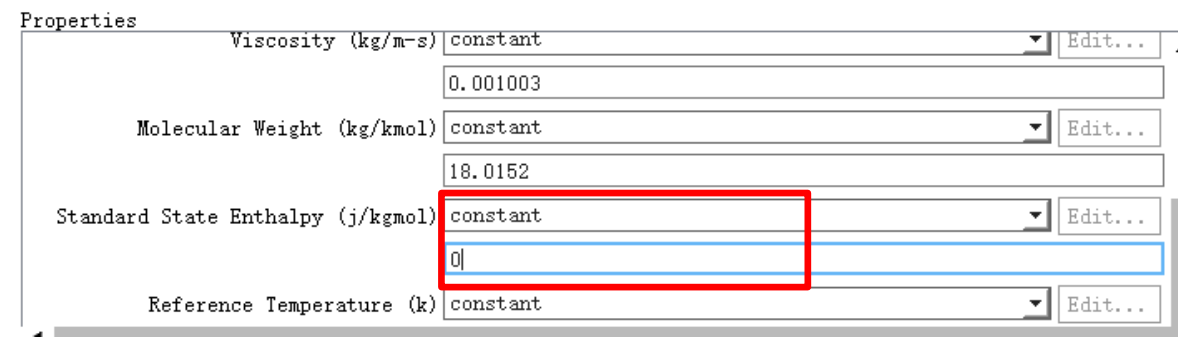
Solution Setup → Materials → Create/Edit Material

■ Create water-liquid



The screenshot shows the 'Create/Edit Materials' dialog box. The 'Materials' list on the left has 'water-liquid' selected. The 'Fluent Database Materials' list shows 'water-liquid (h2o<1>)' selected. The 'Properties' section shows various material properties like Density, Cp, Thermal Conductivity, and Viscosity. The 'Copy' button is highlighted with a red box.

1. Click **Fluent Database**
2. Choose **water-liquid**
3. Set **Standard State Enthalpy to 0**
4. Click **Copy**



The screenshot shows the 'Properties' dialog box for 'water-liquid (h2o<1>)'. The 'Standard State Enthalpy (j/kgmol)' field is set to 0 and is highlighted with a red box.

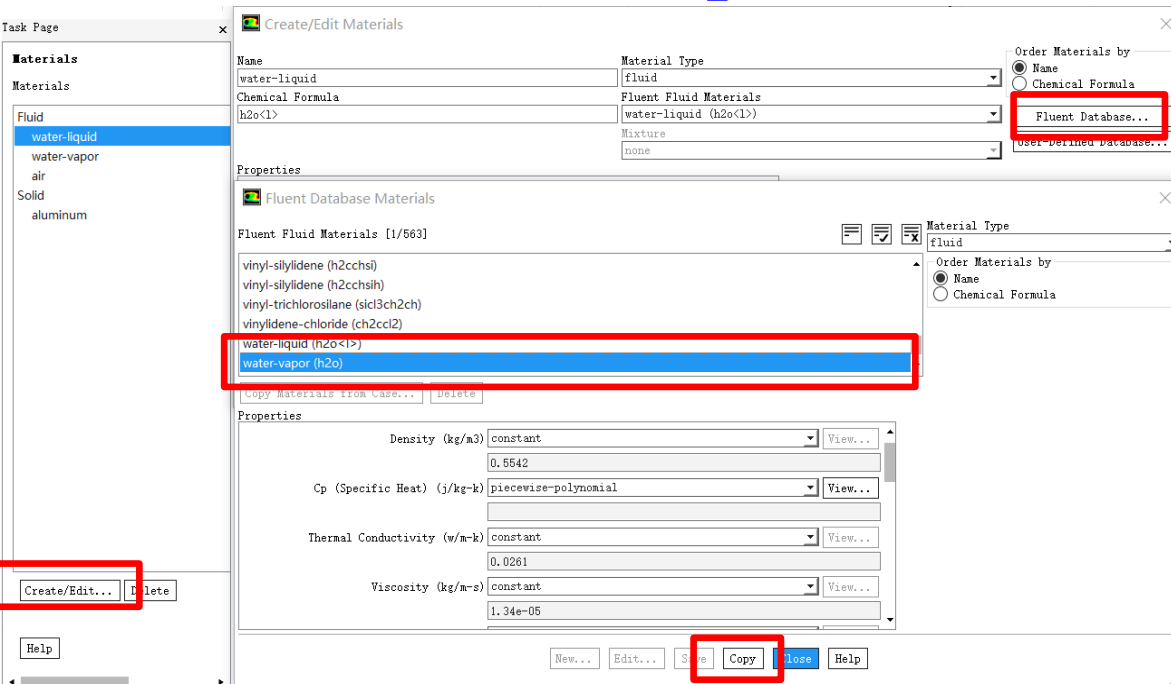
Property	Value
Viscosity (kg/m-s)	0.001003
Molecular Weight (kg/kmol)	18.0152
Standard State Enthalpy (j/kgmol)	0
Reference Temperature (k)	constant

2.7 Define the materials

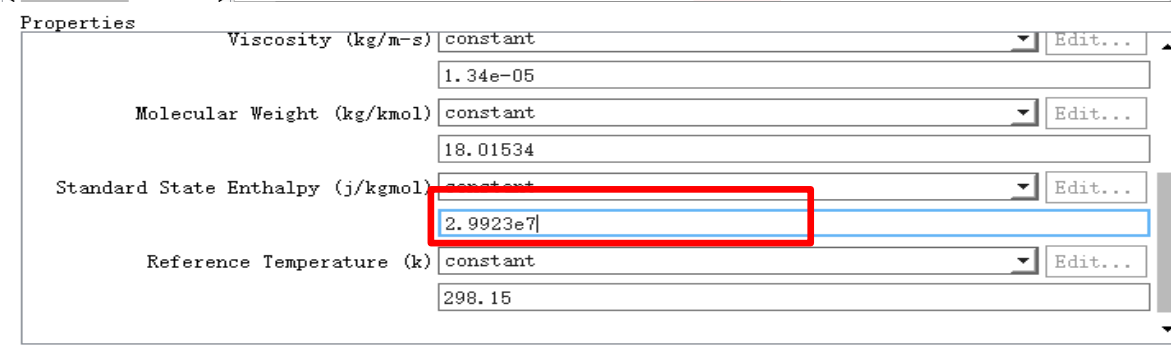
Solution Setup → Materials → Create/Edit Material

■ Create water-vapor

1. Click **Fluent Database**
2. Choose **water-vapor**
3. Set Standard State Enthalpy to **2.9923e7**
4. Click **Copy**



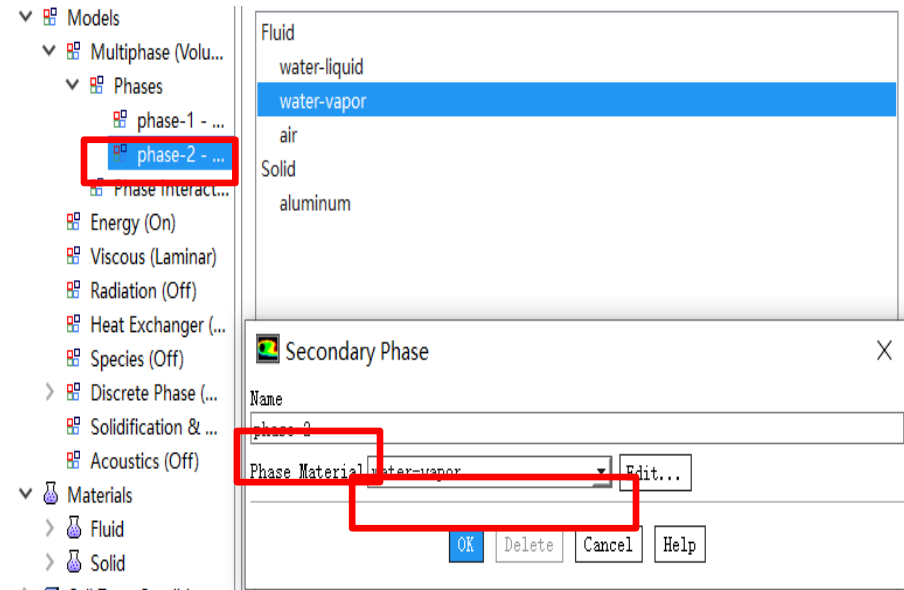
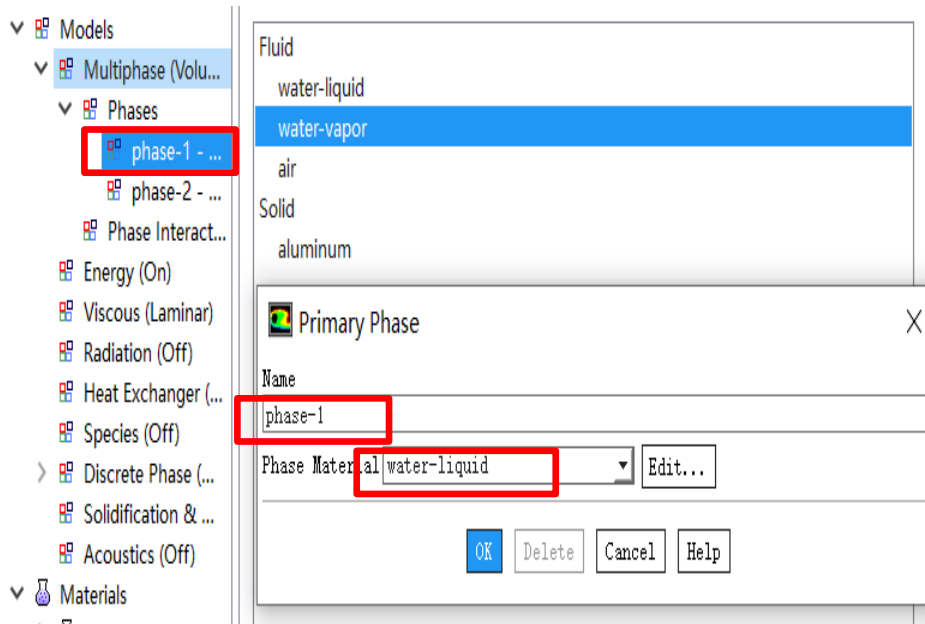
The difference between the Standard State Enthalpy of two phases is the **latent heat** of phase change.



2.8 Define the phases

Solution Setup → Phases

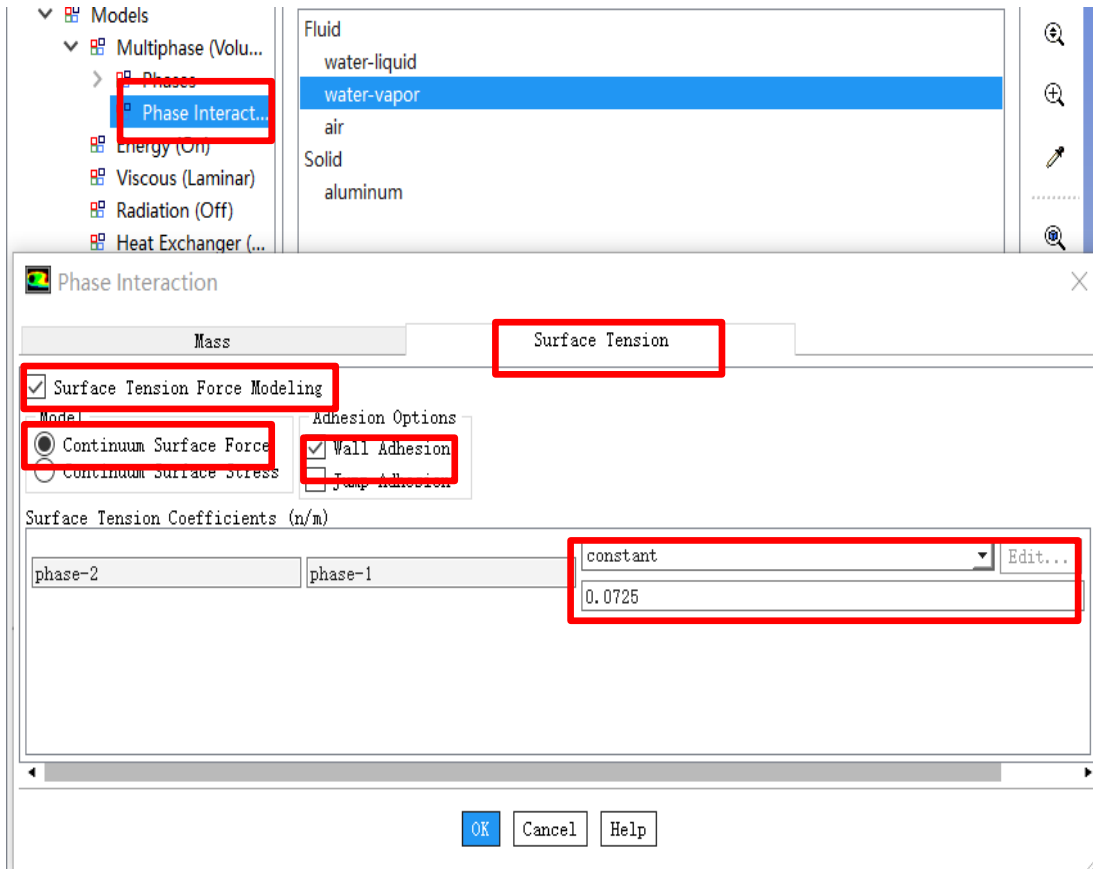
- Choose **water-liquid** as Primary Phase
- Choose **water-vapor** as Secondary Phase



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

2.8 Define the phases

Define surface tension force

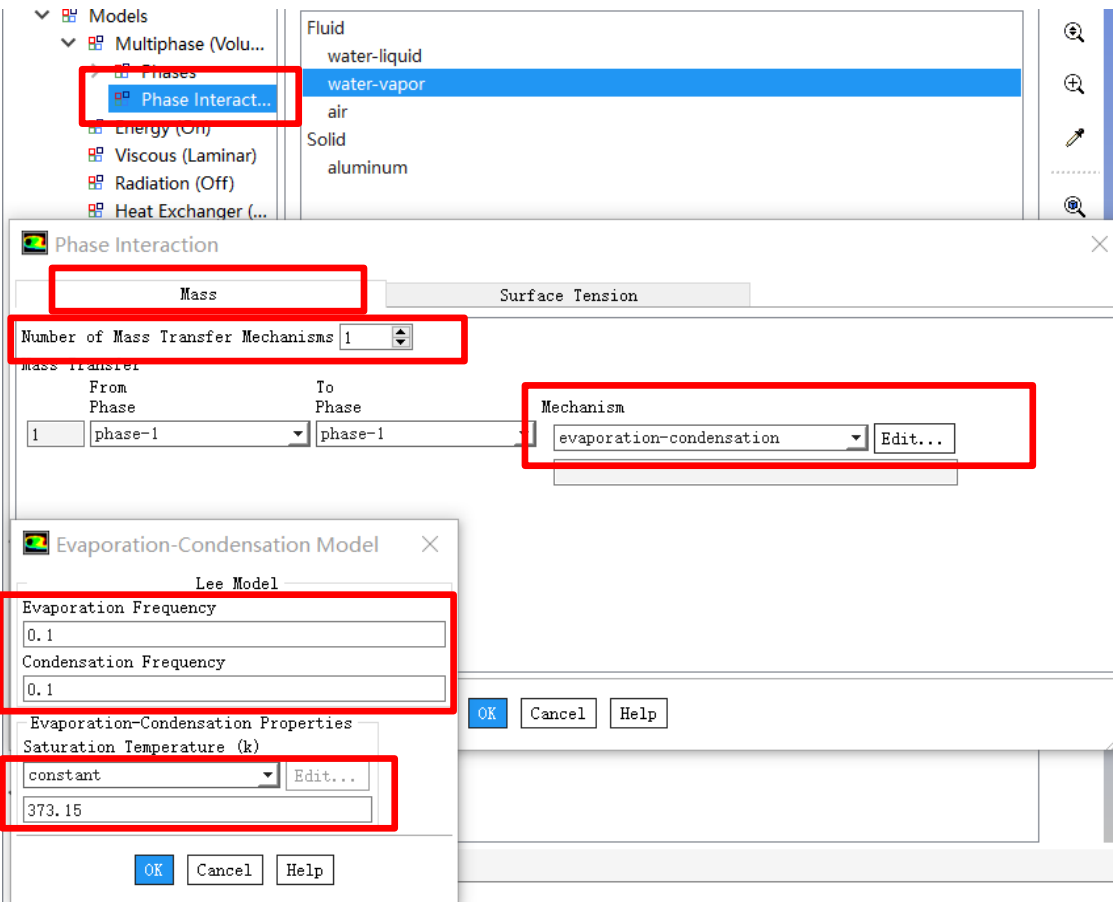


1. Click Interaction
2. Click Surface Tension
3. Select Surface Tension Force Modeling
4. Choose Continuum Surface Force and Wall Adhesion
5. Choose constant and write 0.0725

2.8 Define the phases

Define phase-change model

1. Click Interaction
2. Click Mass
3. Change Number of Mass Transfer Mechanisms to 1
4. Select Mechanism as evaporation-condensation
5. Set Evaporation and Condensation Frequency to 0.1
6. Set Saturation Temperature to 373.15



The screenshot displays the ANSYS Fluent interface for defining a phase-change model. The 'Phase Interaction' dialog box is open, showing the 'Mass' tab. The 'Number of Mass Transfer Mechanisms' is set to 1. A table lists the mass transfer mechanisms, with the first entry showing a mechanism from 'phase-1' to 'phase-1' using 'evaporation-condensation'. The 'Evaporation-Condensation Model' dialog is also open, showing the 'Lee Model' with 'Evaporation Frequency' and 'Condensation Frequency' both set to 0.1, and 'Saturation Temperature (k)' set to 373.15.

From Phase	To Phase	Mechanism
1	phase-1	evaporation-condensation

2.8 Define the phases

phase-change model: Lee Model

$$\frac{\partial C_1}{\partial t} + \mathbf{u} \cdot \nabla C_1 = S_1$$

$$\frac{\partial C_v}{\partial t} + \mathbf{u} \cdot \nabla C_v = S_v$$

$$S_v = -S_1 = \alpha_g \rho_g \frac{T - T_{\text{sat}}}{T_{\text{sat}}}$$

for condensation ($T_{\text{sat}} > T$)

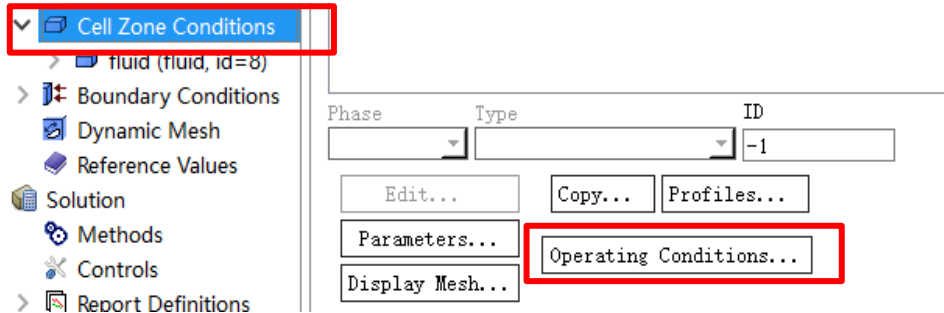
$$S_v = -S_1 = \alpha_l \rho_l \frac{T - T_{\text{sat}}}{T_{\text{sat}}}$$

for evaporation ($T_{\text{sat}} < T$)

Simplified model with phase change defined such that saturating conditions at the interface can be achieved.

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.

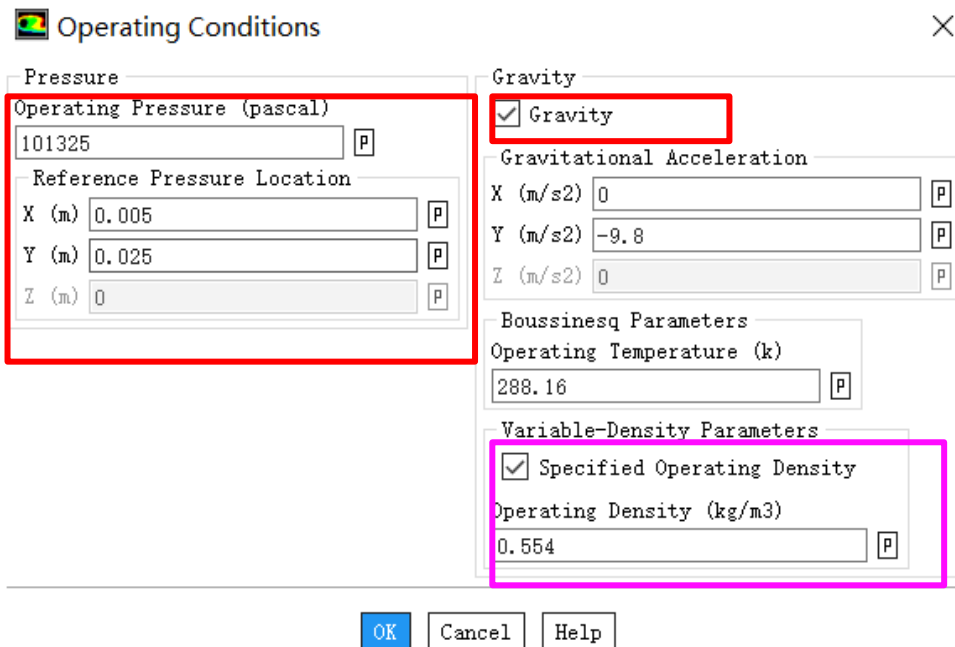
Here input the density of the water-vapor.

Variable-Density Parameters

Specified Operating Density

Operating Density (kg/m³)

0.554



2.10 Define the boundary conditions

Solution Setup → Boundary Condition

Phase	Type	ID
mixture	symmetry	11

- For the boundary on both sides, define symmetry condition.

Phase	Type	ID
mixture	pressure-outlet	11

- For the outlet, define pressure outlet

- Set backflow temperature

Pressure Outlet

Zone Name: wall1 Phase: mixture

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Gauge Pressure (pascal) 0 constant

Pressure Profile Multiplier 1

Backflow Direction Specification Method: Normal to Boundary

Backflow Pressure Specification: Total Pressure

OK Cancel Help

Pressure Outlet

Zone Name: wall1 Phase: mixture

Momentum Thermal Radiation Species DPM Multiphase Potential UDS

Backflow Total Temperature (k) 373 constant

OK Cancel Help

2.10 Define the boundary conditions

For the bottom surface, Define contact angle

Phase	Type	ID
mixture	wall	11

- Choose **wall** as **Type**
- Input value of the **contact angle 120°** for adiabatic wall, **10°** for high temperature wall.
- The angle is measured by **vapor** here.

Wall

Zone Name: wall1 Phase: mixture

Adjacent Cell Zone: fluid

Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film Potential

Wall Motion: Stationary Wall Moving Wall Relative to Adjacent Cell Zone

Shear Condition: No Slip Specified Shear Specularity Coefficient Marangoni Stress

Wall Roughness: Roughness Height (m) 0 constant Roughness Constant 0.5 constant

Wall Adhesion: Contact Angles (deg) phase-2 phase-1 120 constant

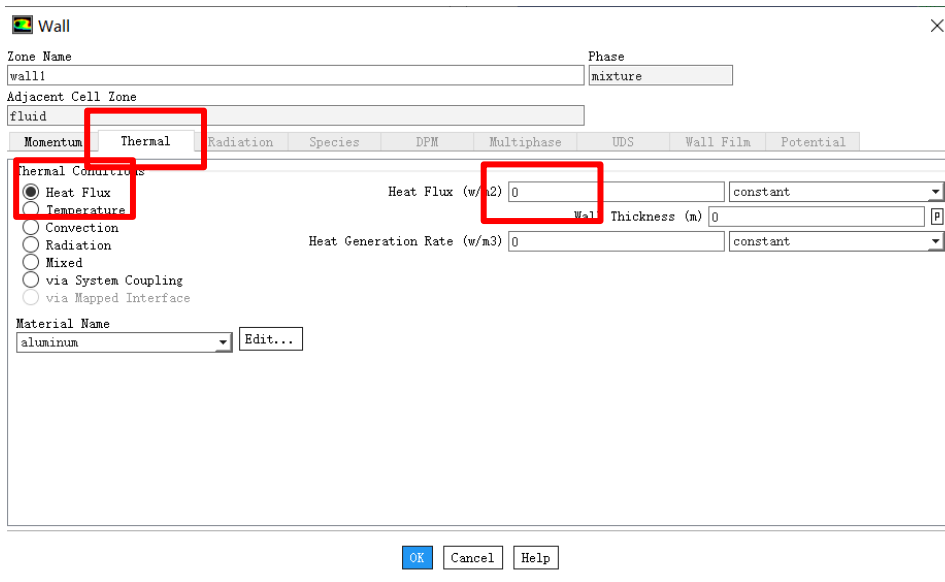
Wall Adhesion

Contact Angles (deg)

phase-2 phase-1 0 constant

2.10 Define the boundary conditions

For the bottom surface, Define heat boundary condition



- For the adiabatic wall, set heat flux to **zero**.
- For the high temperature wall, set temperature to **573.15 K**.

Thermal Conditions

- Heat Flux
- Temperature
- Convection
- Radiation
- Mixed

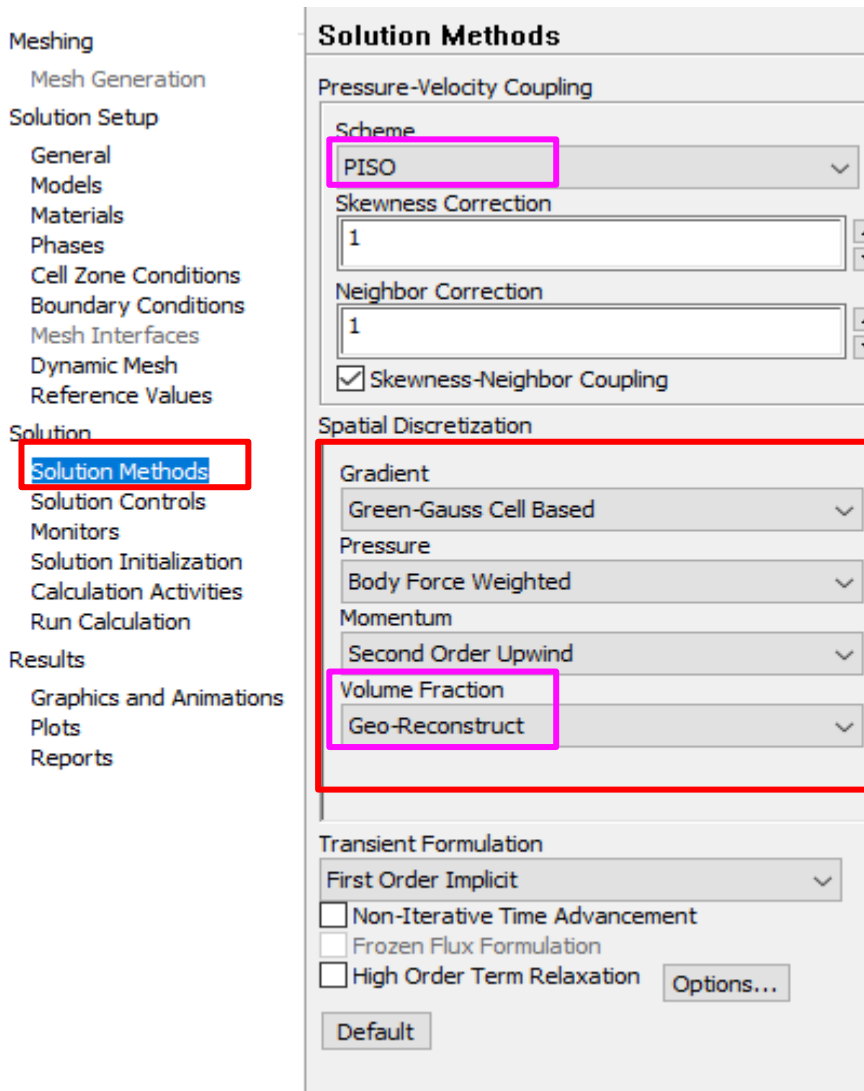
Temperature (K) constant

Wall Thickness (m) P

Heat Generation Rate (w/m³) constant

2.11 Choose the solution methods

Solution → Solution Methods



The screenshot shows the 'Solution Methods' dialog box in ANSYS Fluent. The 'Solution' tab is selected in the left-hand tree. The 'Scheme' dropdown is set to 'PISO'. The 'Skewness Correction' and 'Neighbor Correction' are both set to '1'. The 'Skewness-Neighbor Coupling' checkbox is checked. The 'Spatial Discretization' section is highlighted with a red box, and the 'Volume Fraction' and 'Geo-Reconstruct' options are highlighted with a pink box. The 'Transient Formulation' section is at the bottom, with 'First Order Implicit' selected and 'Non-Iterative Time Advancement', 'Frozen Flux Formulation', and 'High Order Term Relaxation' unchecked.

- 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

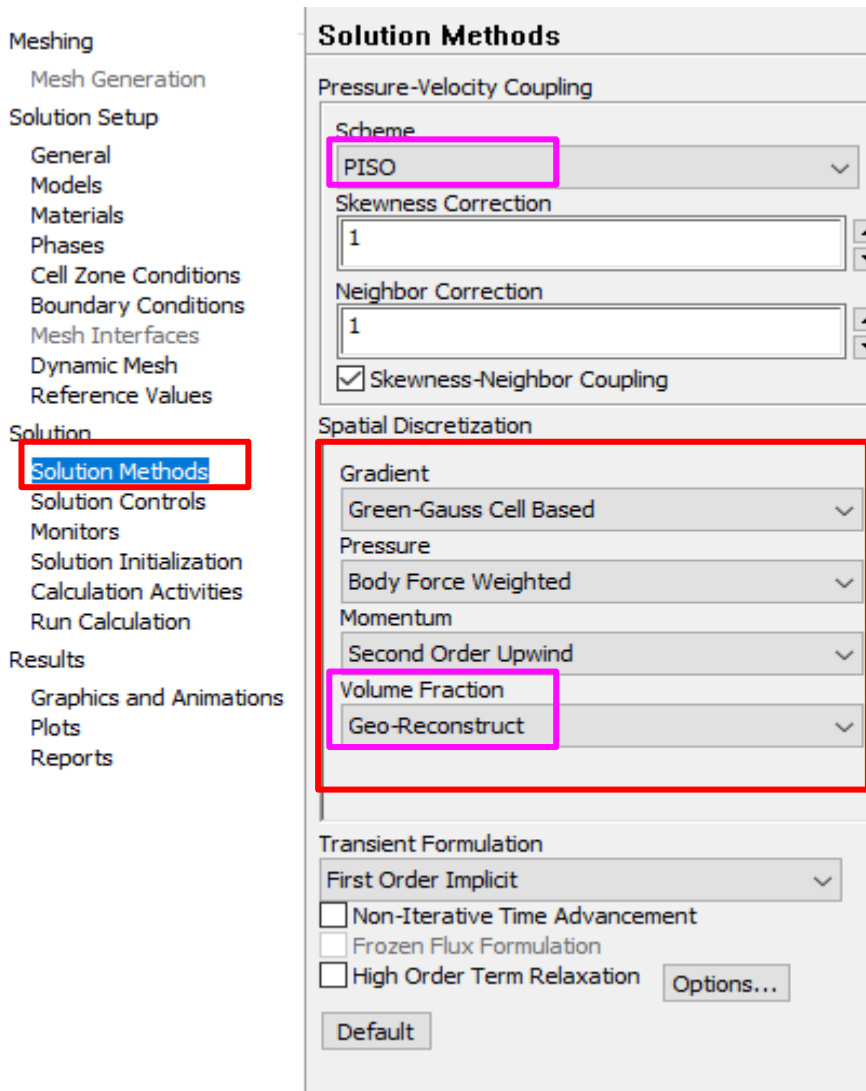
Correction step: solve pressure correction, obtain u and v that satisfying the Mass Conservation Equation.

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



The screenshot shows the 'Solution Methods' dialog box in ANSYS Fluent. The 'Solution' menu item in the left sidebar is highlighted with a red box, and the 'Solution Methods' sub-menu item is also highlighted with a red box. The 'Scheme' dropdown is set to 'PISO' (highlighted with a pink box). The 'Skewness Correction' and 'Neighbor Correction' are both set to '1'. The 'Skewness-Neighbor Coupling' checkbox is checked. The 'Spatial Discretization' section is highlighted with a red box, and the 'Volume Fraction' dropdown is highlighted with a pink box. The 'Transient Formulation' section is set to 'First Order Implicit'.

- 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

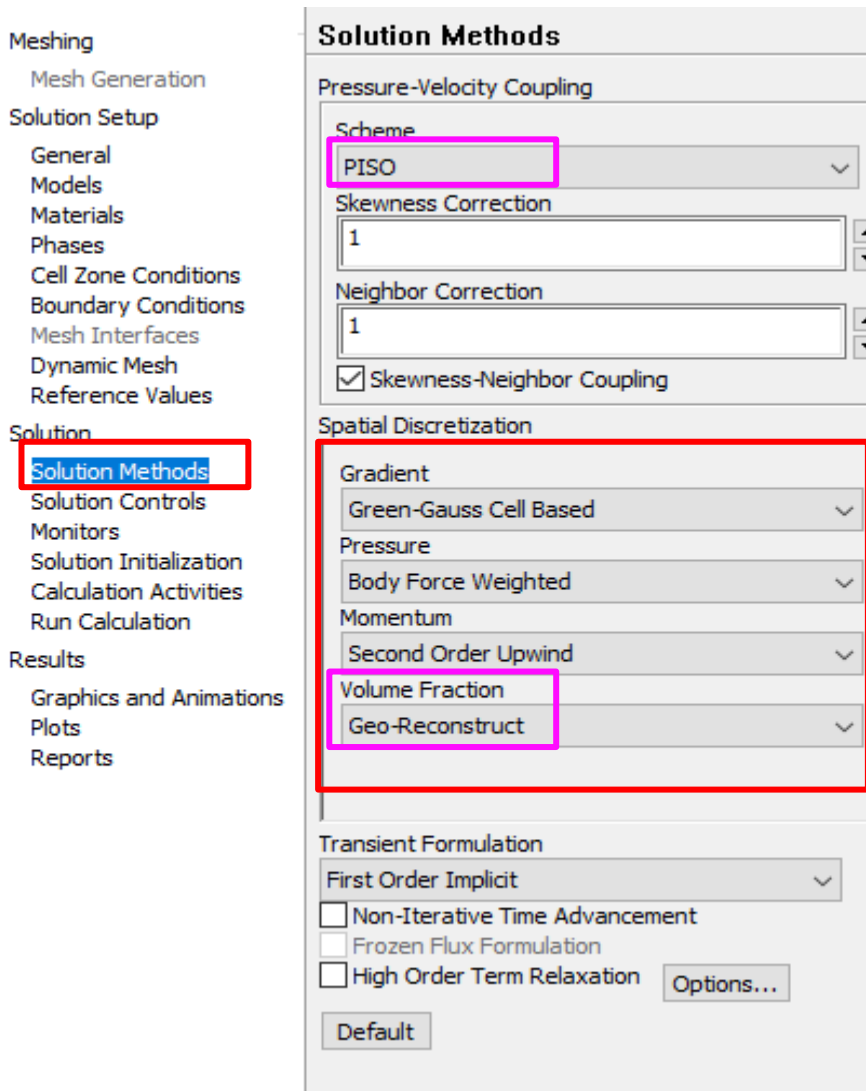
$$\langle \nabla \phi \rangle = \frac{1}{V_C} \int_{V_C} \nabla \phi dV = \frac{1}{V_C} \int \phi \cdot \mathbf{n} dS = \sum \phi_f \cdot \mathbf{n} S$$

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



The screenshot shows the ANSYS Fluent 'Solution Methods' panel. The 'Solution' menu item is highlighted in red. The 'Solution Methods' sub-menu is also highlighted in red. The 'Scheme' dropdown is set to 'PISO'. The 'Skewness Correction' and 'Neighbor Correction' are both set to '1'. The 'Skewness-Neighbor Coupling' checkbox is checked. The 'Spatial Discretization' section is highlighted in red, with 'Volume Fraction' and 'Geo-Reconstruct' highlighted in pink. The 'Transient Formulation' section is set to 'First Order Implicit'.

- 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.

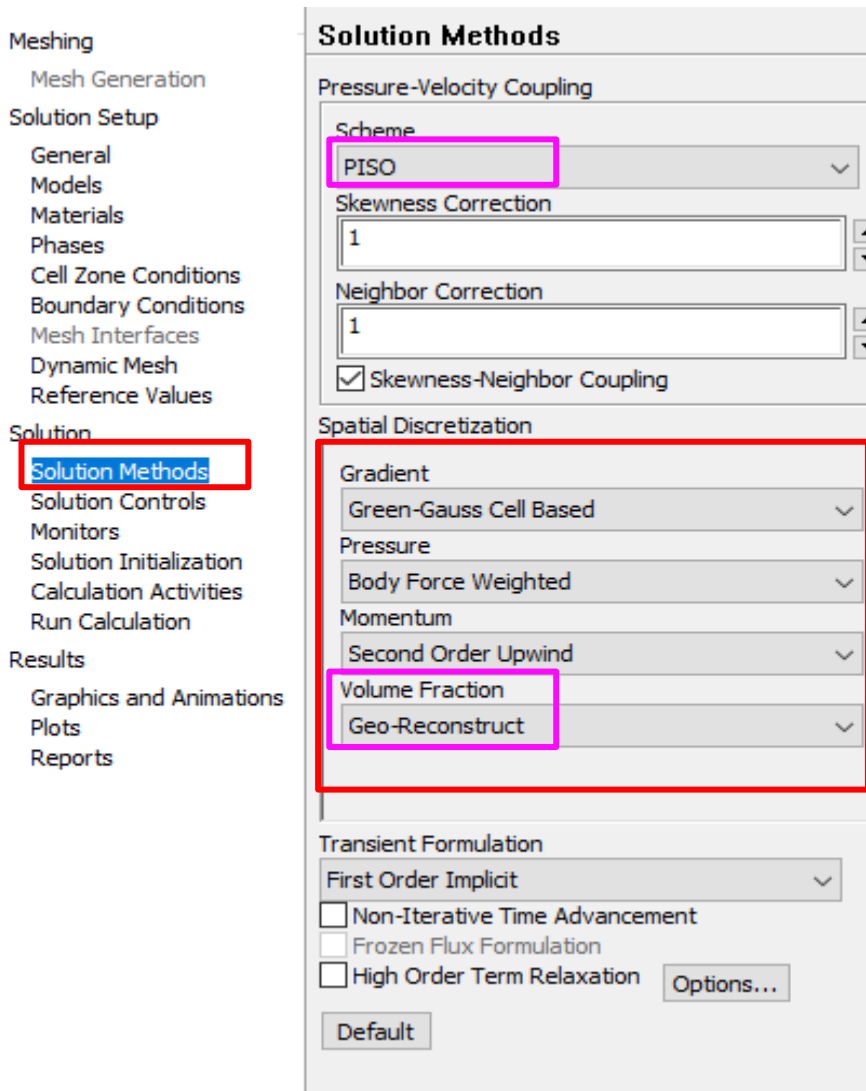
- ✓ Multiphase flow such as VOF (Volume of Fluid, 体积分函数法) or LS (Level Set, 水平集): **recommended.**
- ✓ For porous media: **not recommended!**

5. **PRESTO!** (Pressure Staggering Option) scheme

For problem with high pressure gradient.

2.11 Choose the solution methods

Solution → Solution Methods



The screenshot shows the ANSYS Fluent 'Solution Methods' dialog box. The 'Solution' tab is selected in the left-hand tree. The 'Scheme' dropdown is set to 'PISO'. The 'Skewness Correction' and 'Neighbor Correction' are both set to '1'. The 'Skewness-Neighbor Coupling' checkbox is checked. The 'Spatial Discretization' section is highlighted with a red box, and the 'Volume Fraction' dropdown is highlighted with a pink box. The 'Transient Formulation' section is at the bottom, with 'First Order Implicit' selected and 'Non-Iterative Time Advancement', 'Frozen Flux Formulation', and 'High Order Term Relaxation' unchecked.

- 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

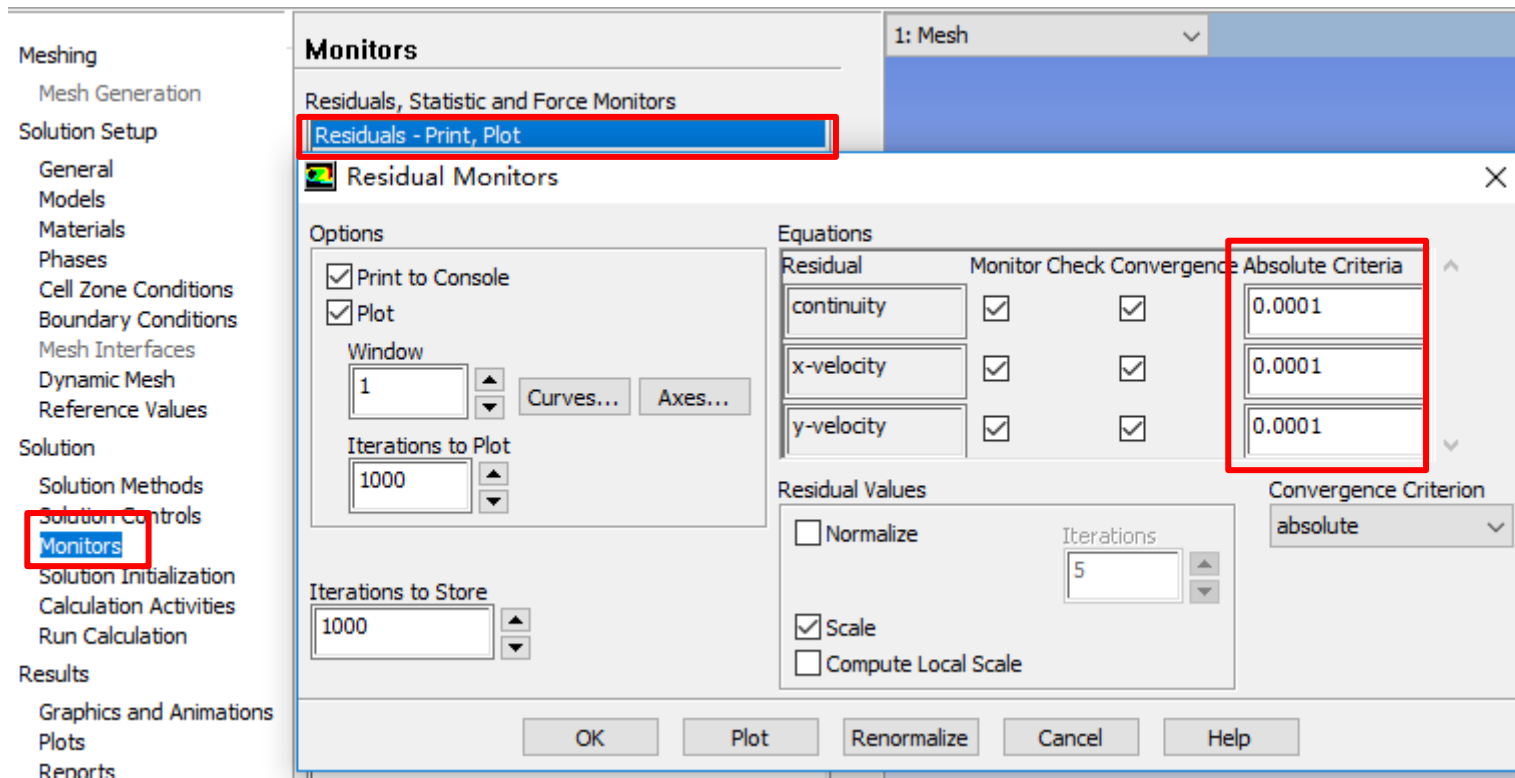
The geometric reconstruction interpolation scheme recommended when time-accurate transient behaviors of the multiphase are required. In other words, it can accurately predict the sharp interface. This scheme is the most accurate and is applicable for general unstructured meshes.

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



The screenshot shows the ANSYS Fluent interface with the **Monitors** dialog box open. The **Residuals - Print, Plot** option is selected in the **Monitors** list. The **Residual Monitors** dialog box is also open, showing the following settings:

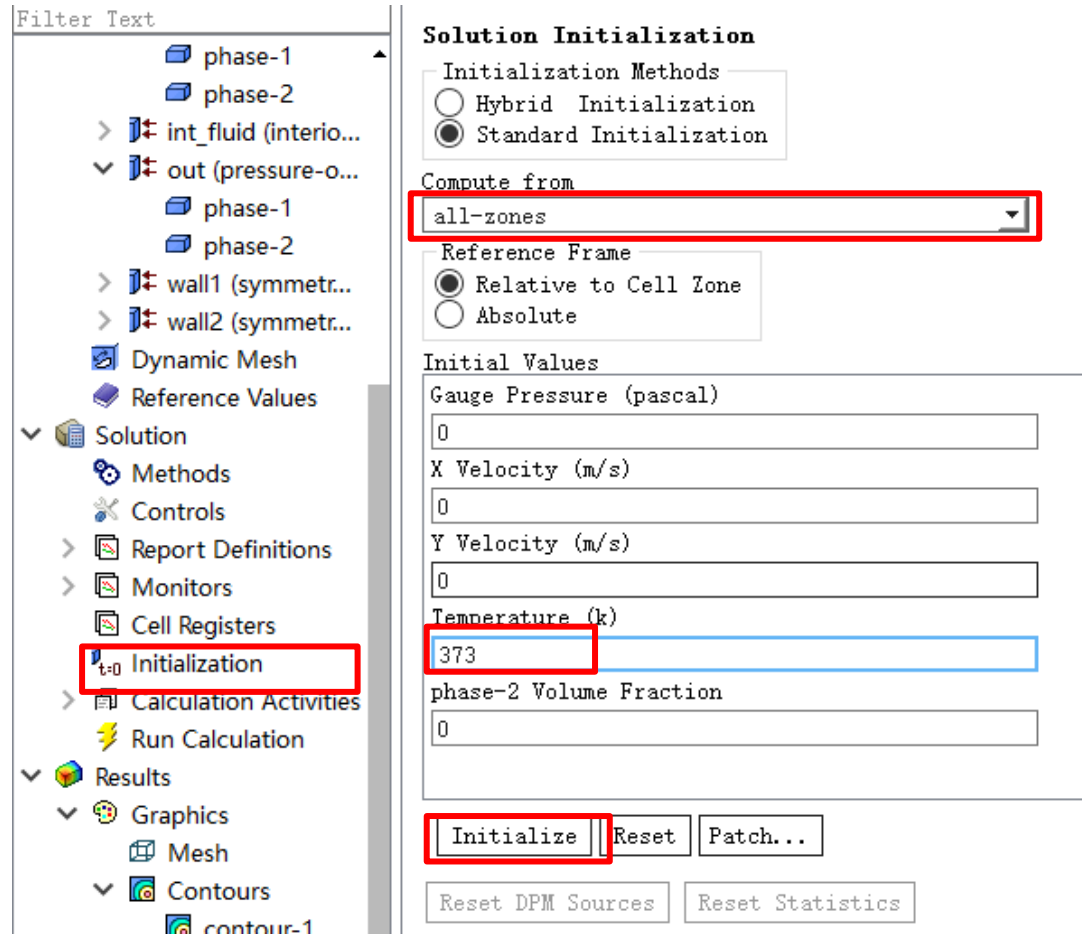
- Options:**
 - Print to Console
 - Plot
 - Window: 1
 - Iterations to Plot: 1000
 - Iterations to Store: 1000
- Equations:**

Residual	Monitor	Check	Convergence	Absolute Criteria
continuity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.0001
x-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.0001
y-velocity	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	0.0001
- Residual Values:**
 - Normalize
 - Scale
 - Compute Local Scale
 - Iterations: 5
- Convergence Criterion:** absolute

The **Absolute Criteria** values for continuity, x-velocity, and y-velocity are all set to 0.0001, which is highlighted with a red box in the image.

2.13 Initialization

Solution → Solution Initialization



Solution Initialization

Initialization Methods

Hybrid Initialization

Standard Initialization

Compute from

all-zones

Reference Frame

Relative to Cell Zone

Absolute

Initial Values

Gauge Pressure (pascal)

0

X Velocity (m/s)

0

Y Velocity (m/s)

0

Temperature (k)

373

phase-2 Volume Fraction

0

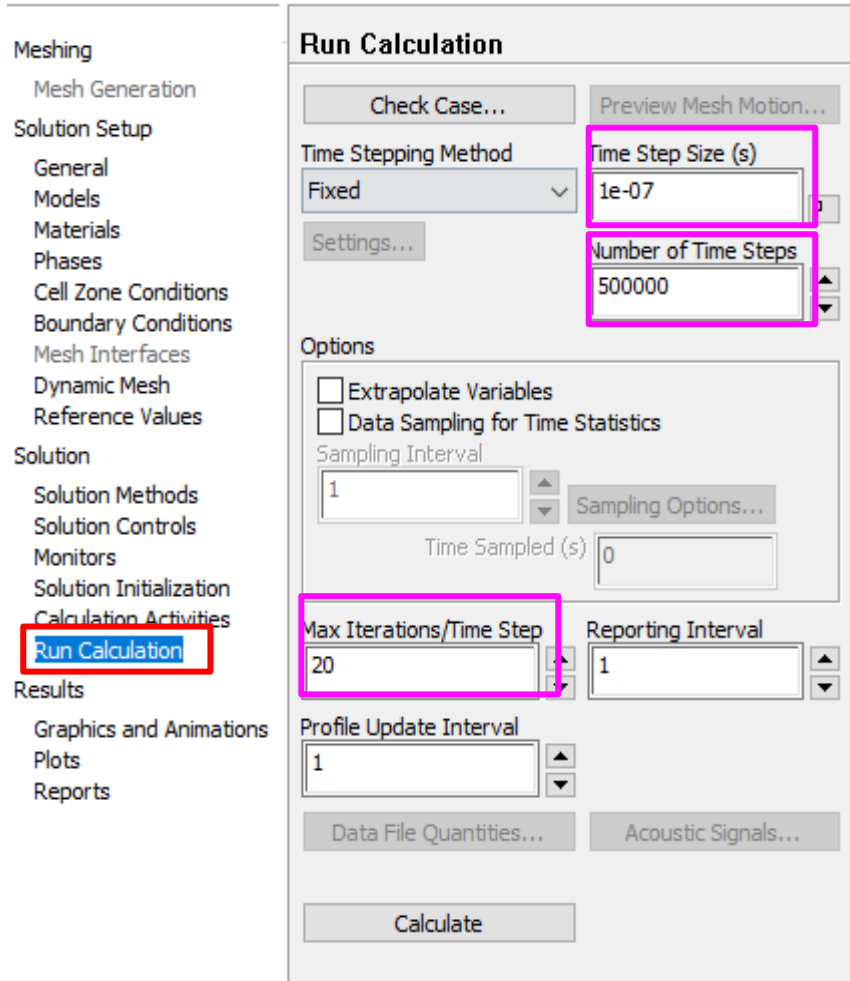
Initialize Reset Patch...

Reset DPM Sources Reset Statistics

- Choose Standard Initialization
- Choose all-zones
- Write 373
- 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.

3 Results

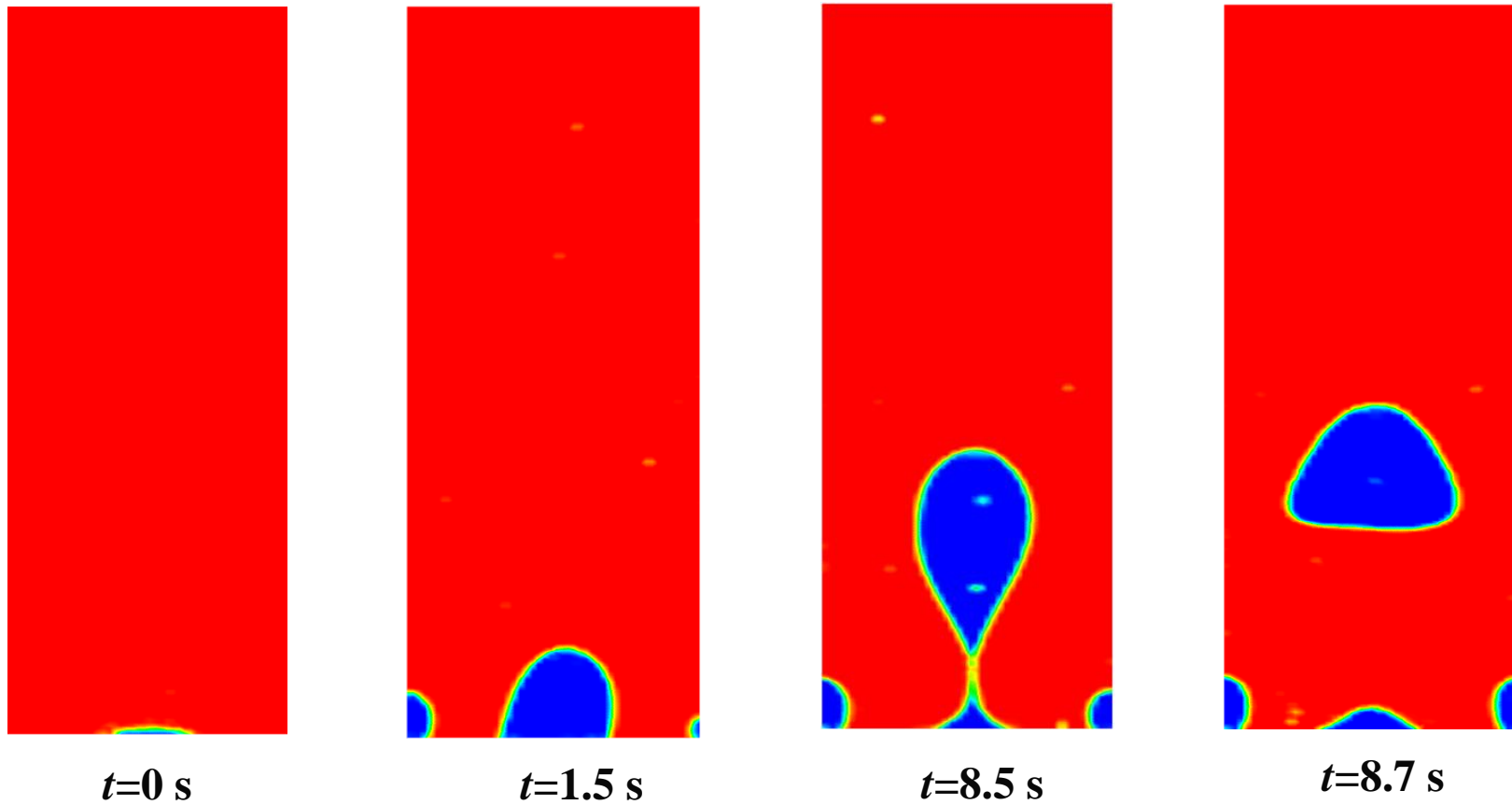


Fig.2 vapor behavior

3 Results

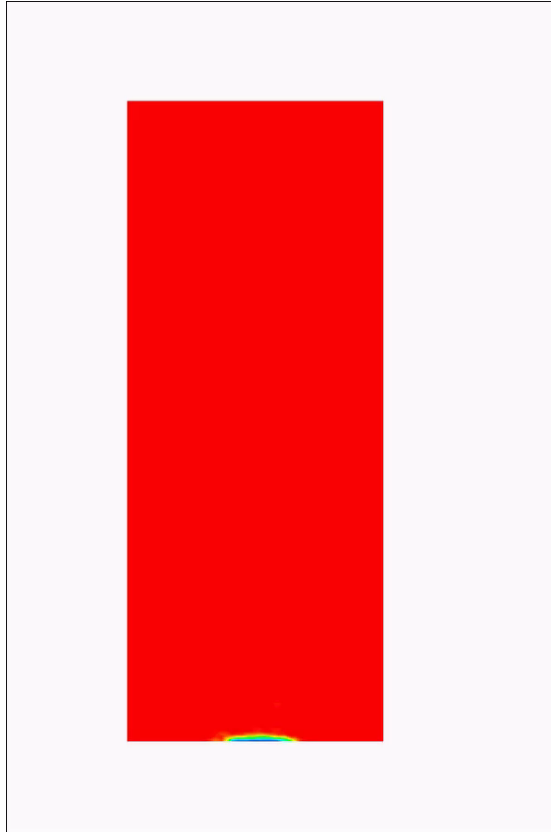


Fig.3 Boiling process

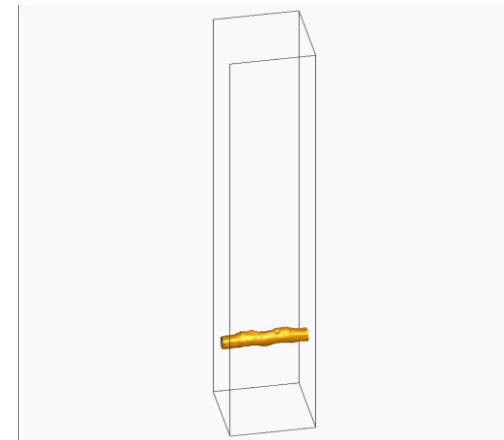
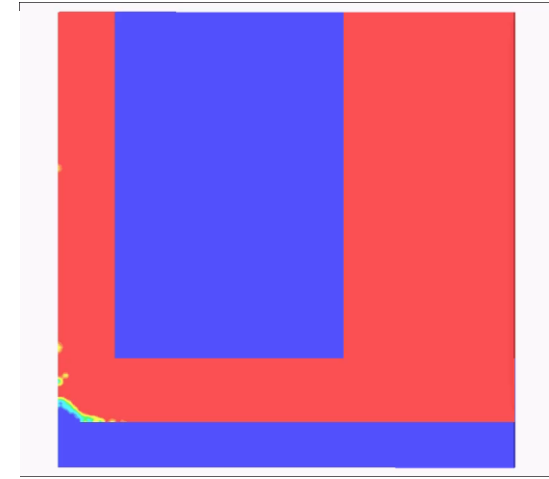
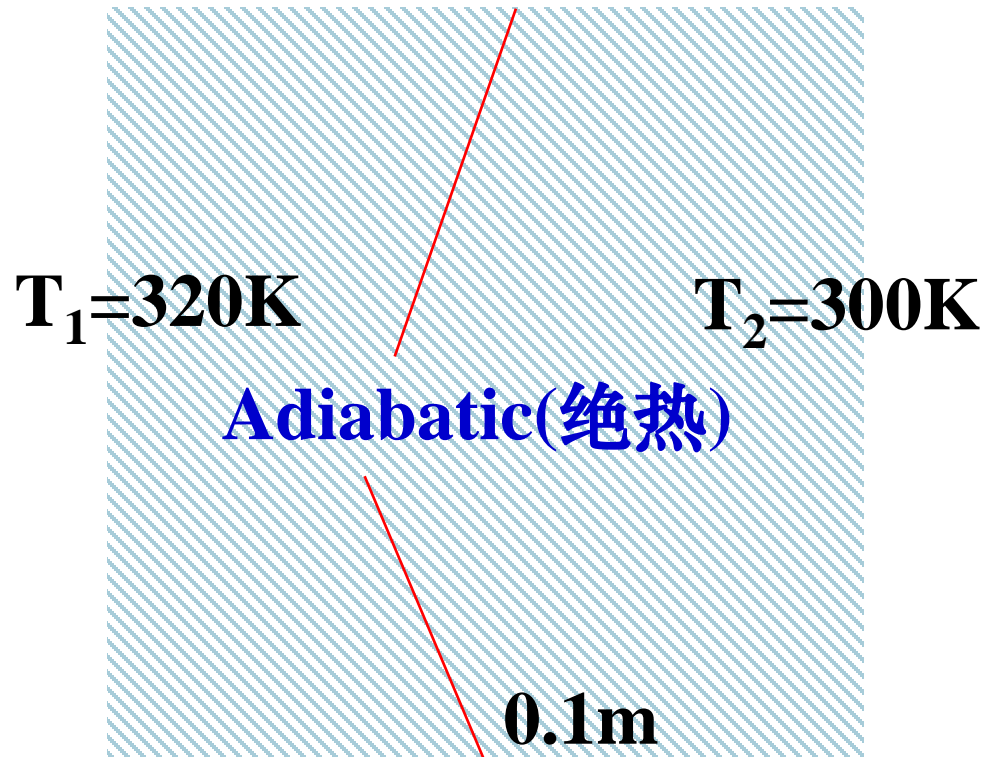


Fig.4 Other boiling results from FLUENT

Computer-aided project (3) of NHT-2021, XJTU

Known: Natural convection of air in a square filled with porous media. The left wall is with high temperature, while the right wall is with low temperature. The bottom and top walls are adiabatic.



Find: effects of **porosity**, **permeability**, **effective thermal conductivity** and **Gr** number on the **velocity field**, **temperature field** and **Nu numbers**.

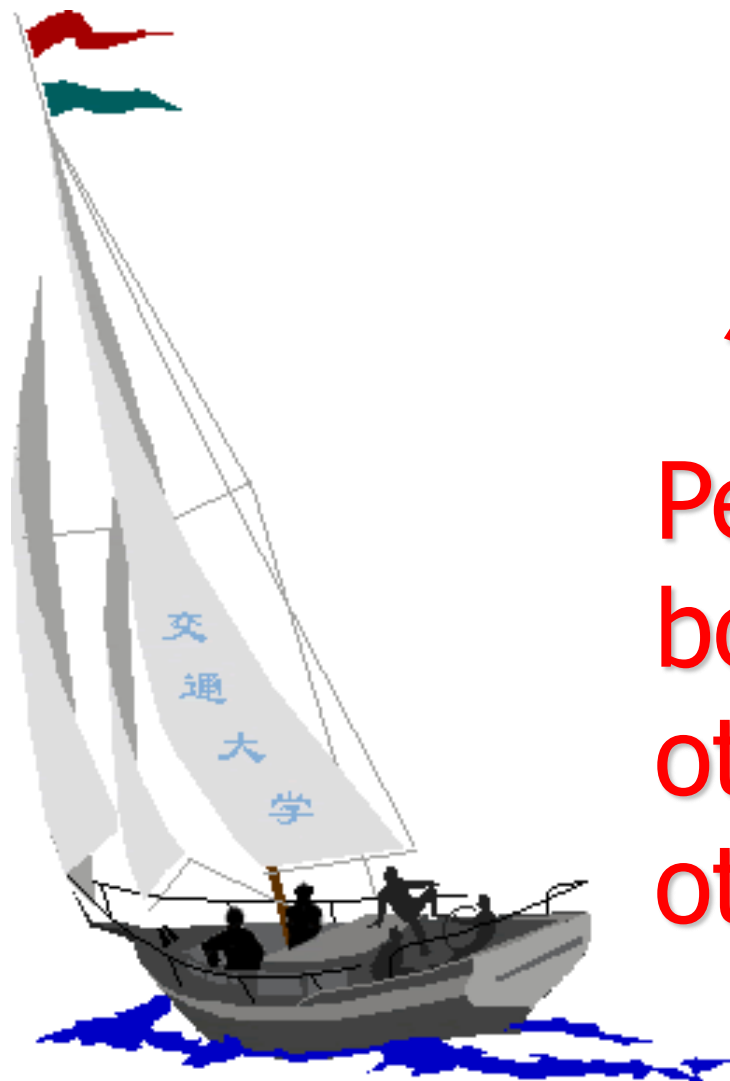


**There is not a winter that does not pass,
There is not a spring that does not come.**



“西”望你我，“安”然无恙

Hope Everyone Safe and Sound



同舟共济 渡彼岸!

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