Numerical Heat Transfer

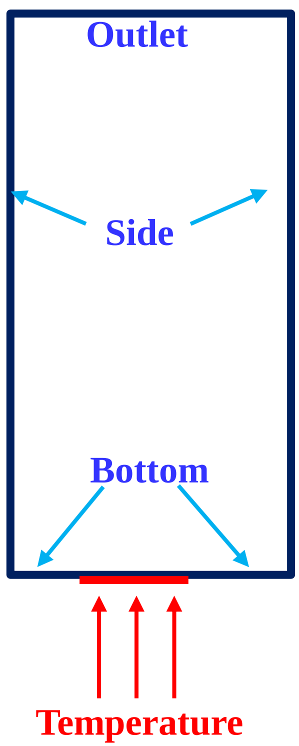
Chapter 13 Application examples of fluent for flow and heat transfer problem

Li Chen, Wen-Quan Tao
CFD-NHT-EHT Center
Key Laboratory of Thermo-Fluid Science & Engineering, XJTU
Xi'an, 2021-Dec.-29

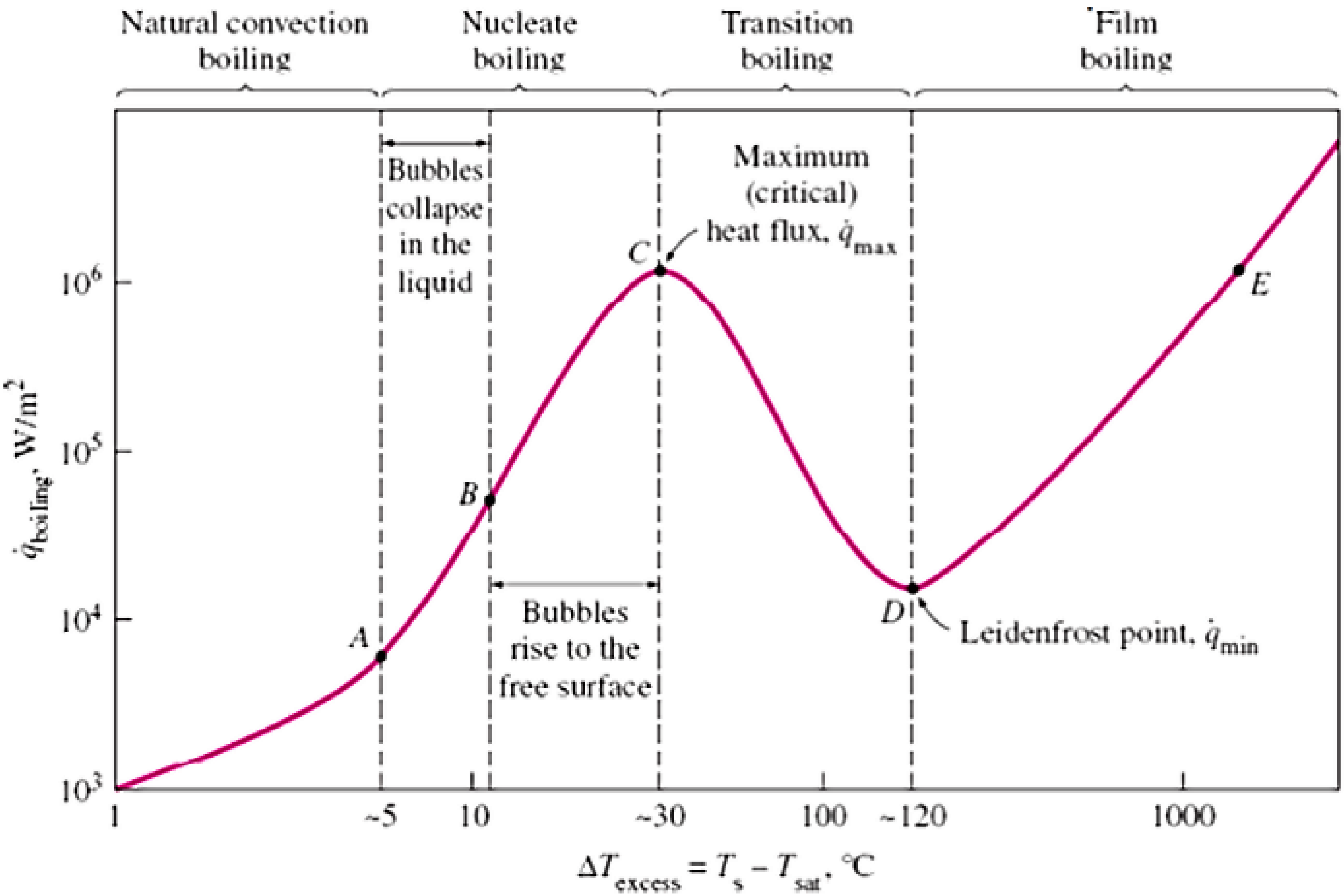
Hope Everyone Safe and Sound

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^{o}
Bottom: adiabatic	Wall, adiabatic, 120^{o}



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

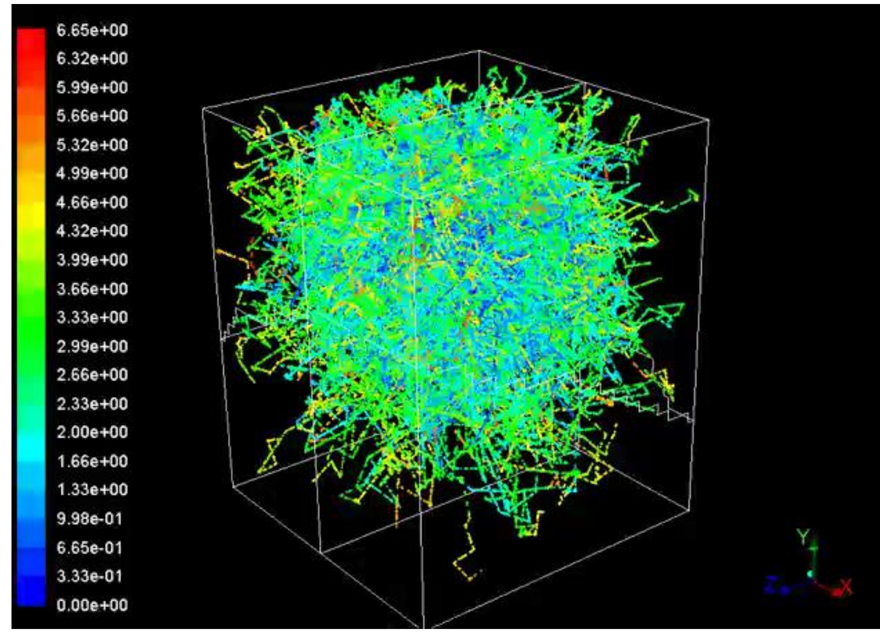
Multiple components （多组分多相）:

Small density: water and oil system

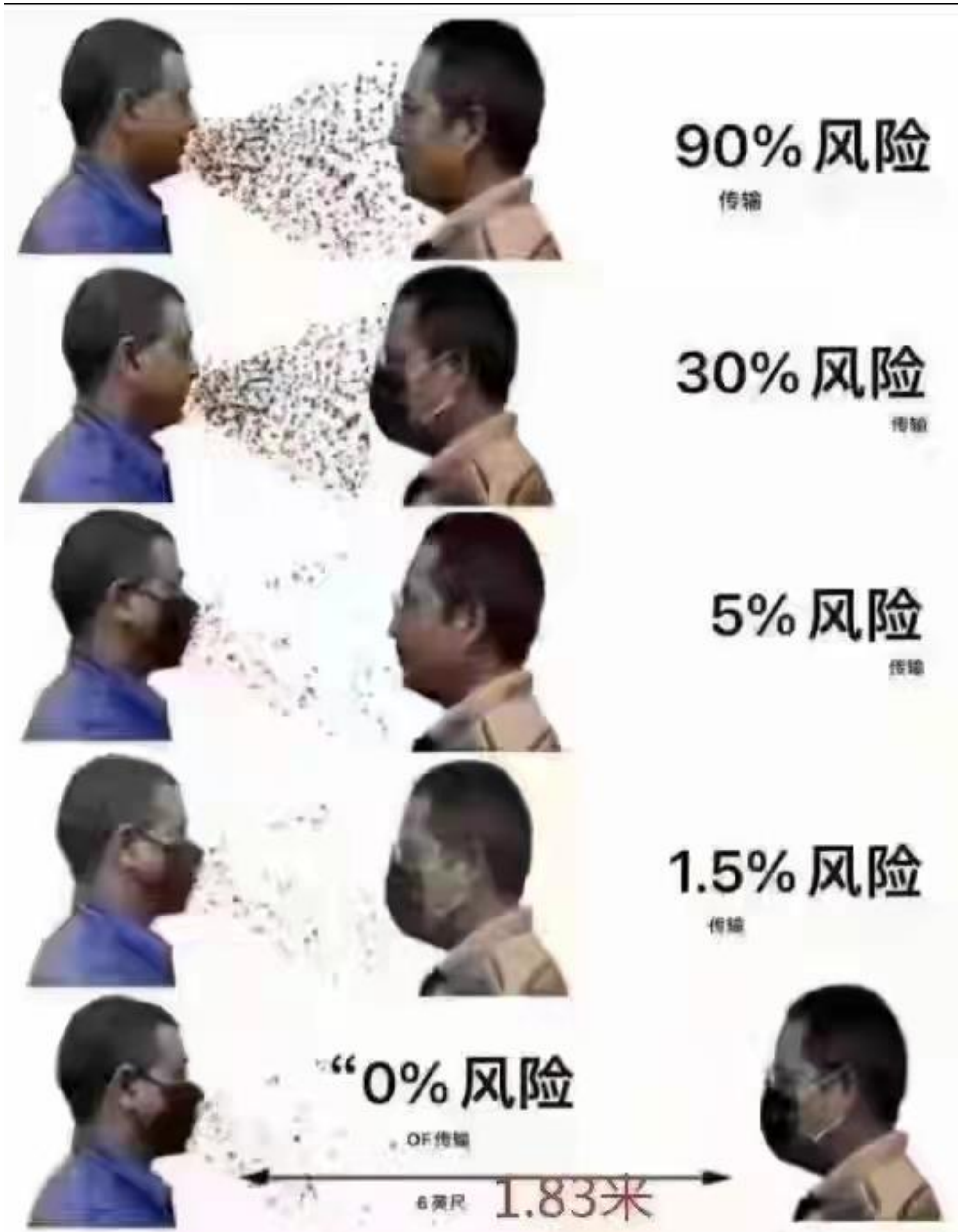
Large density ratio: water and air system

Crown

Multiphase flow during cough



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


90% 风险
传输
30% 风险
传输
5% 风险
传输
1.5% 风险
传输
"0% 风险
OF传输
6英尺 1.83米

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

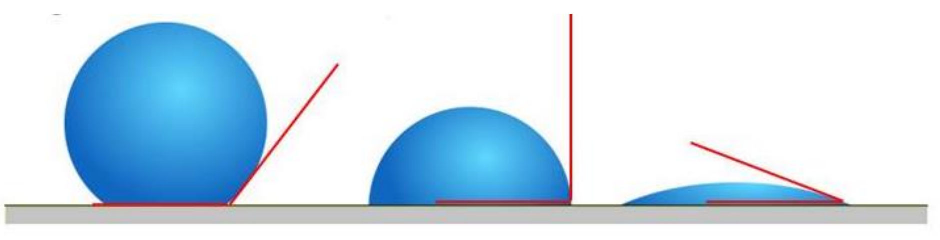
Water striders stay on top of water (水面上的水黾)

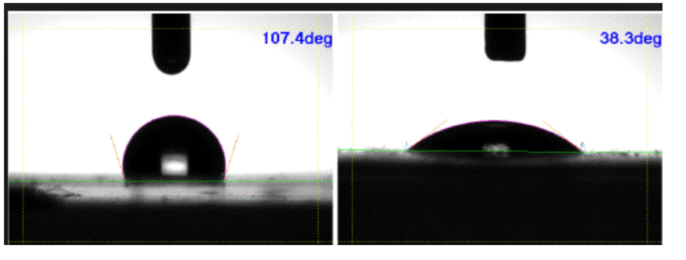
Lotus effect (荷叶效应)

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 （中性表面） with angle as 90.

θ>90° θ=90 ° θ<90 °

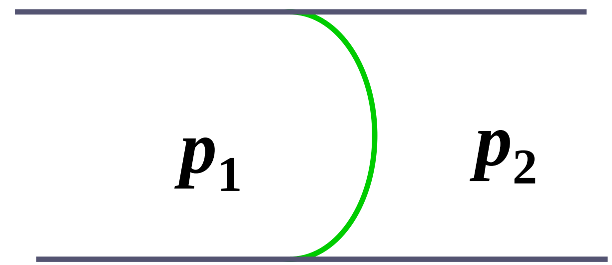




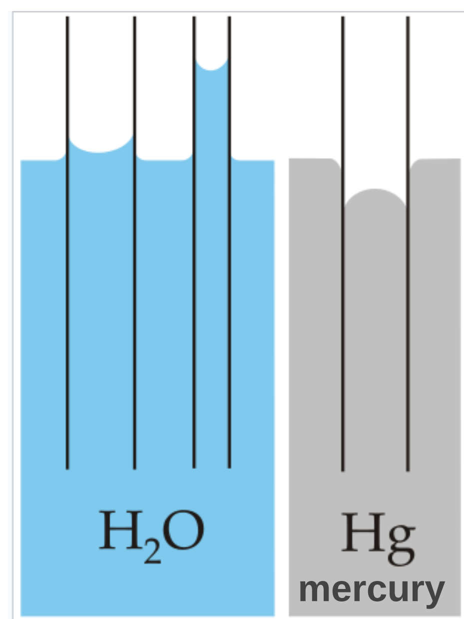
Capillary pressure

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

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）

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 ...

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

The volume fraction of each fluid in a computational cell

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

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

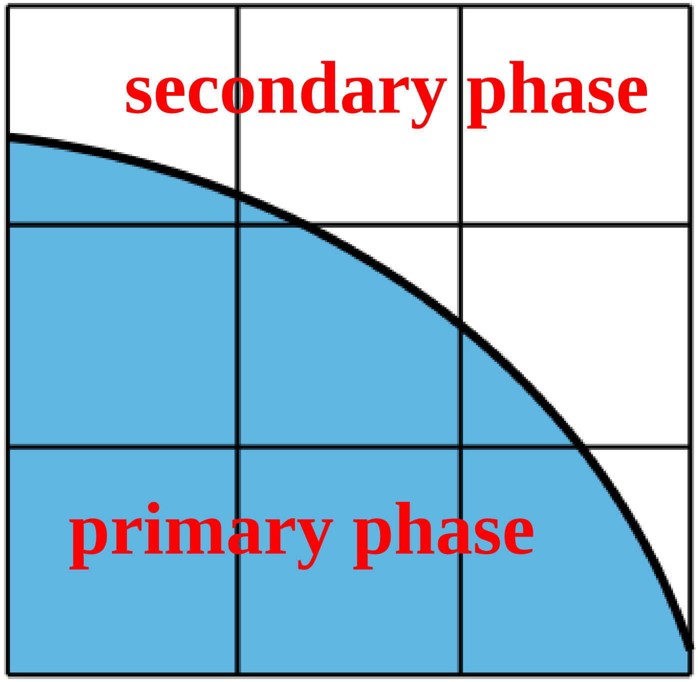
$C_1 = 1$ **The cell is filled with the primary phase**

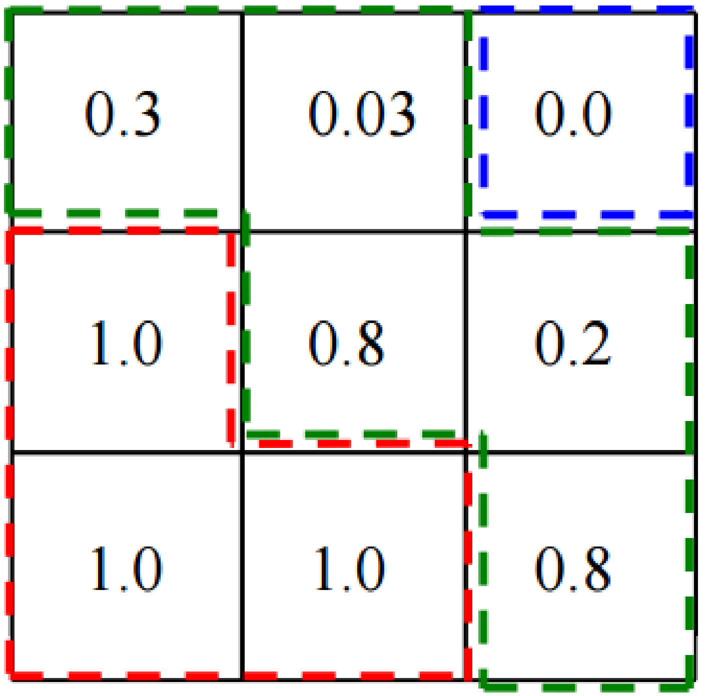
$C_1 = 0$ **The cell is free of primary phase**

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

Schematic of 2D two-phase flow system

C **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

Surface tension force

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

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

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

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

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

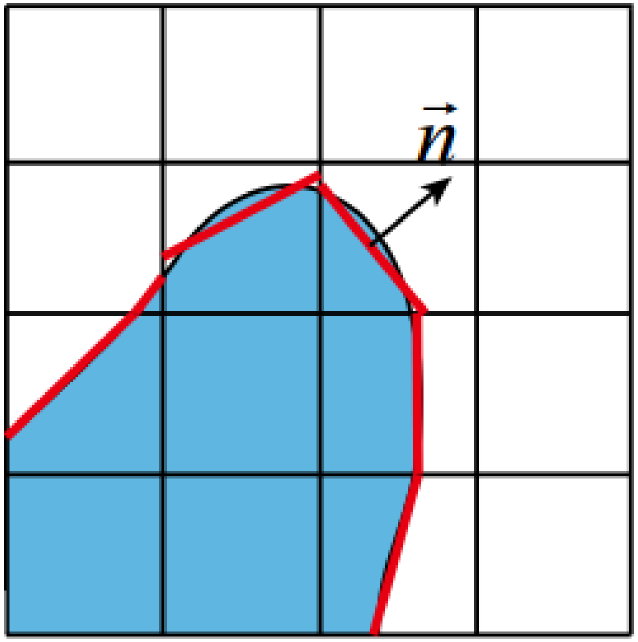
Two-way coupled with each other

Continuum surface force (CSF) model

The form of volumetric force (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.

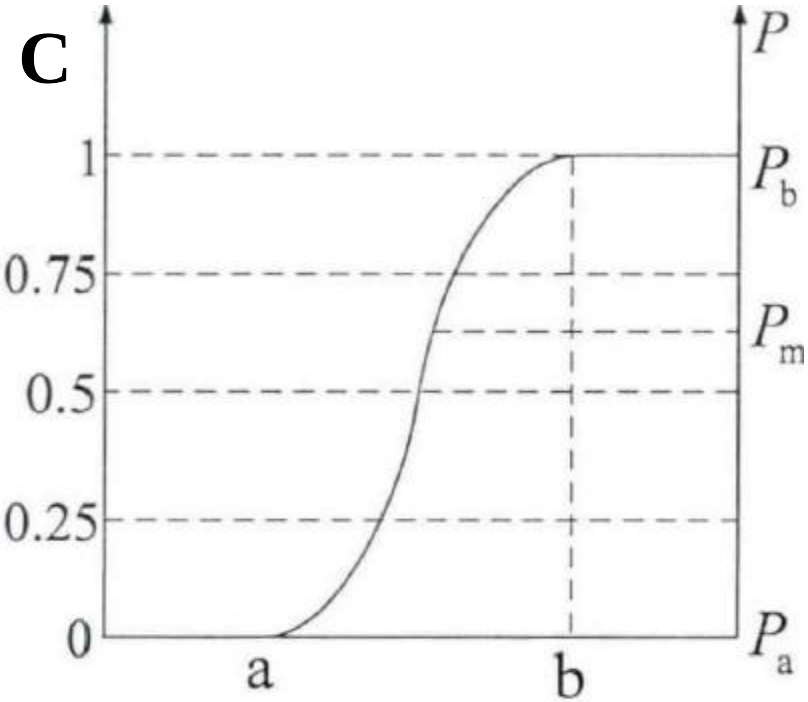
✓ Smooth C

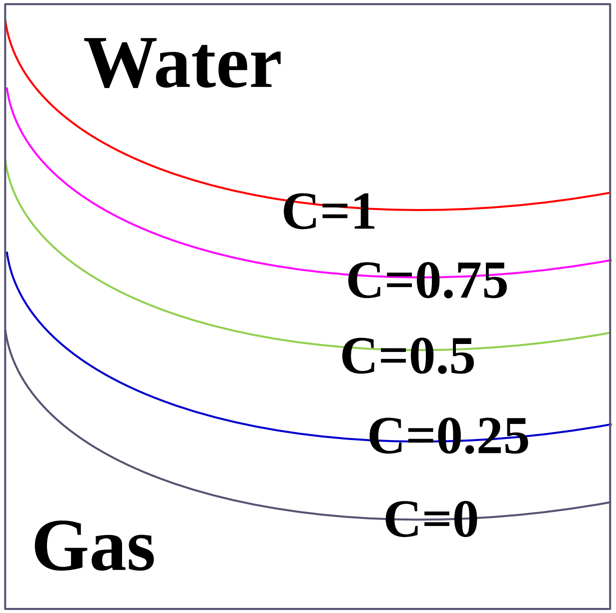
- **VOF in fact is a sharp-interface model.**
- **The thickness of the interface is zero.**
- **The fluid property is sharply changed from 1 to 0 across the interface.**



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.

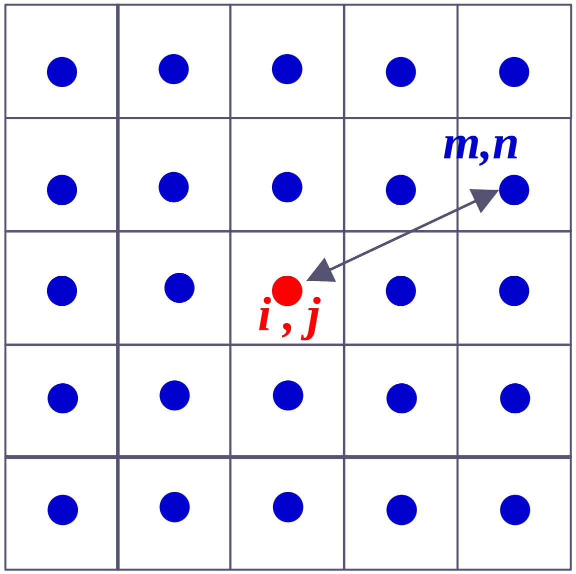




The following function is adopted to smooth C

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

Smoothed one



ε

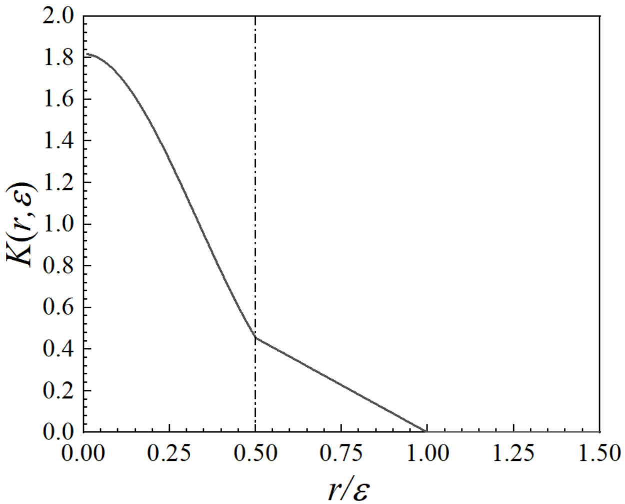
Control the thickness of the interface! 3Δ

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

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

K **Smooth integration kernel**

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



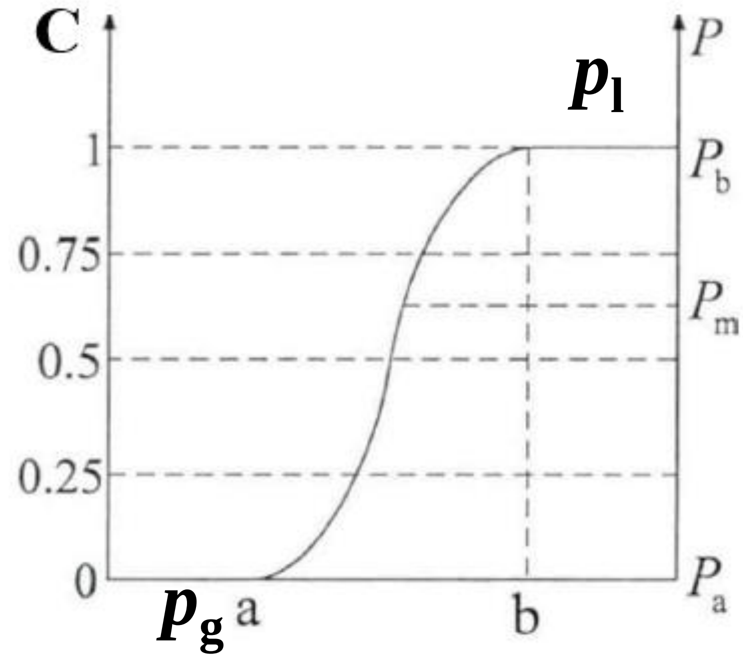
Smoothed *C*, namely $\tilde{C}$, is adopted to calculate force

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

interface mean curvature

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

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 introduced in NHT.

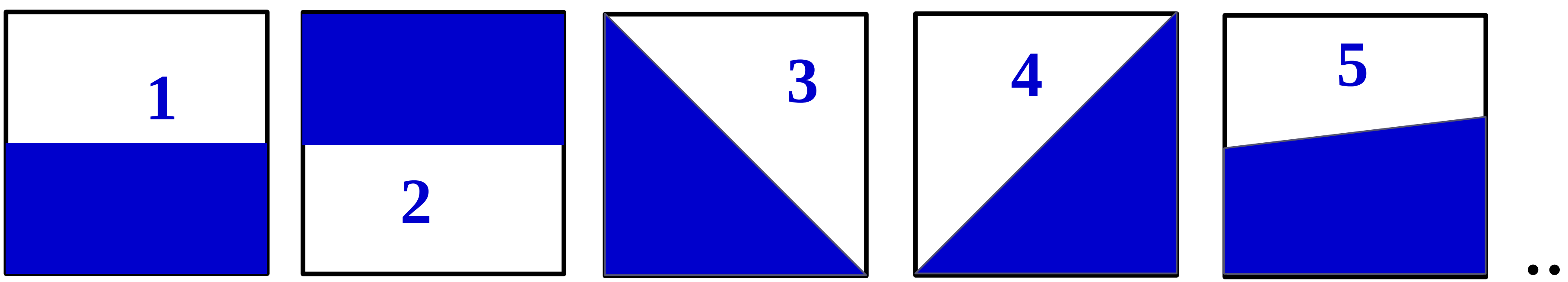
However, because *C* is not a continuous function. Such method may result in false diffusion, leading to gradually increasing thickness of the interface.

(2) . Reconstruction method

Step 1. Interface reconstruction

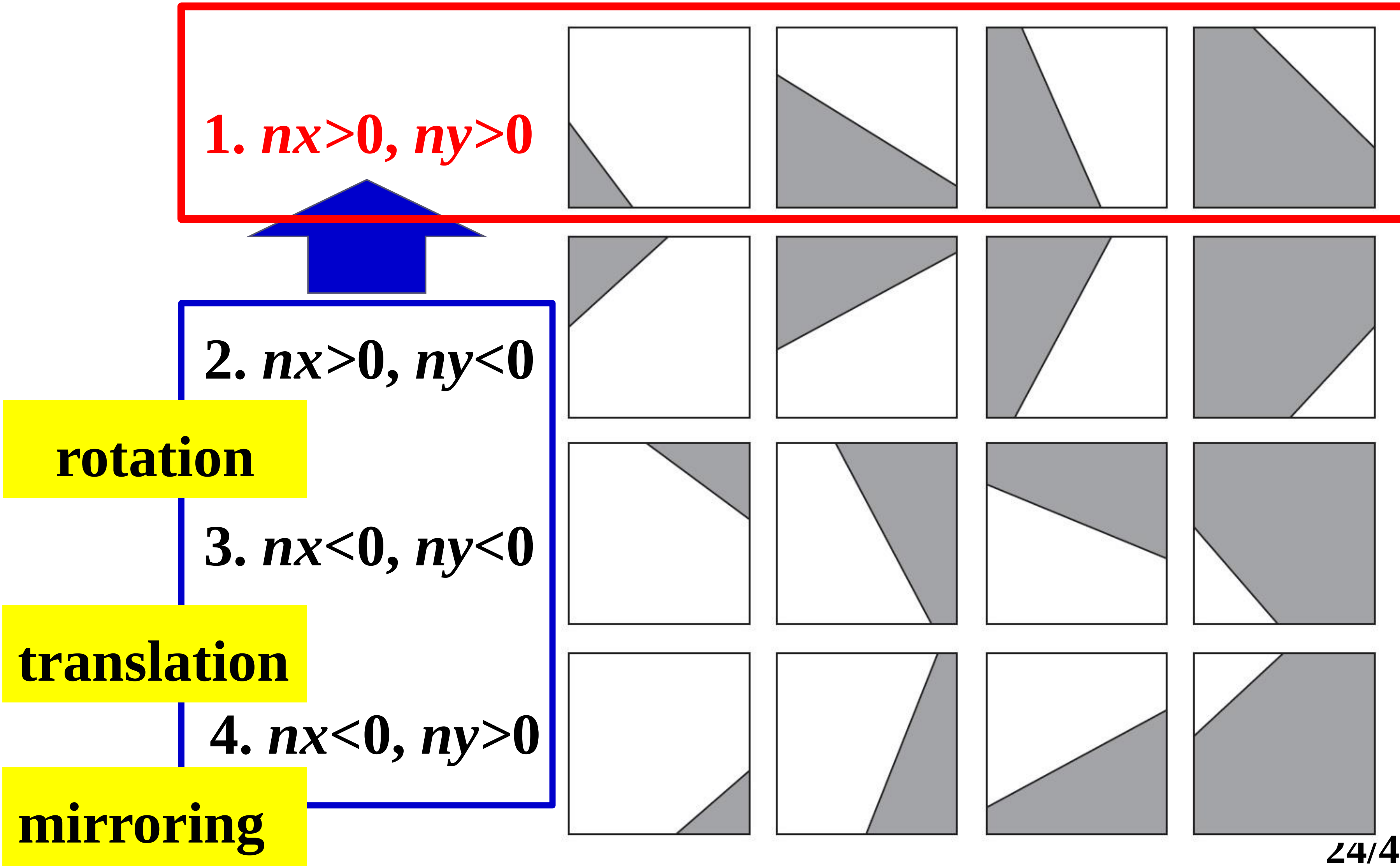
For a value of *C* in a computational cell, the pattern of interface should be determined first.

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

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

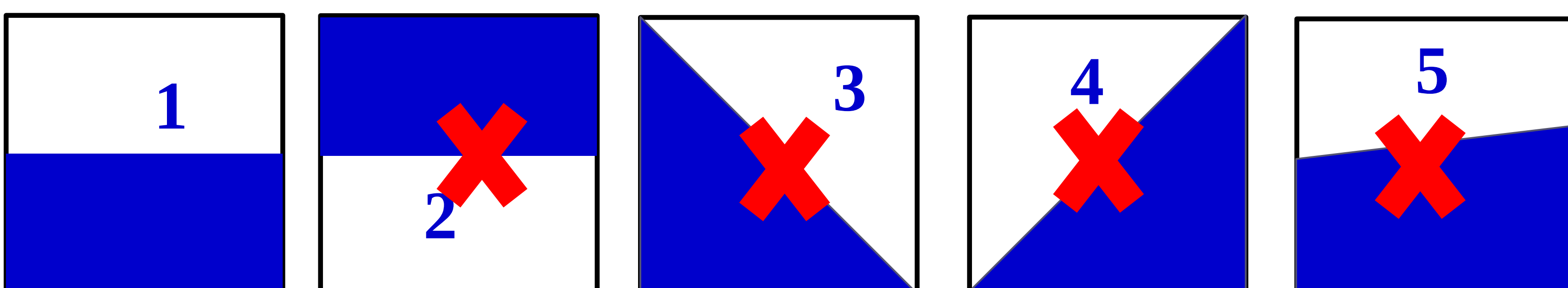
$$n_{i,j}^{y}=(\widetilde{C}_{i+1,j+1}+2\widetilde{C}_{i,j+1}+\widetilde{C}_{i-1,j+1}-\widetilde{C}_{i+1,j-1}-2\widetilde{C}_{i,j-1}-\widetilde{C}_{i-1,j-1})/\delta y$$

Interface normal direction

Volume of fraction

Interface is reconstructed!

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



Normal direction of the interface

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

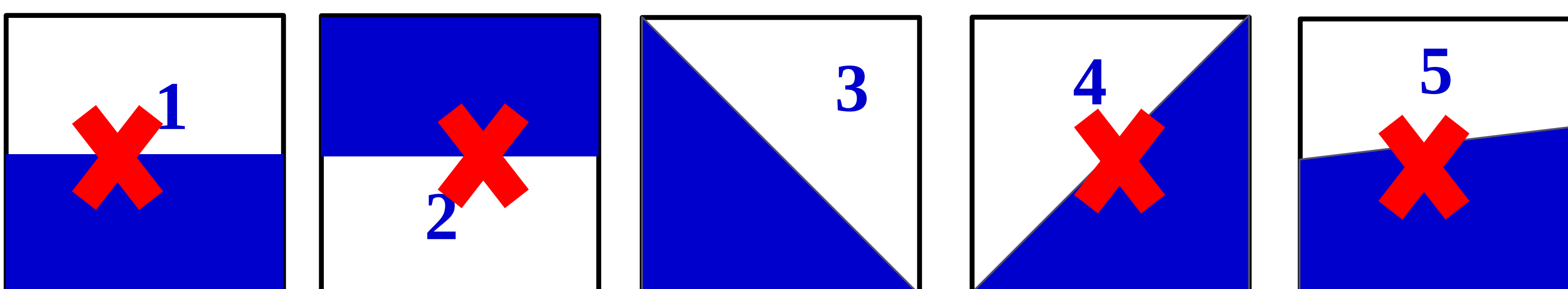
$$n^y_{i,j} = (\tilde{C}_{i+1,j+1} + 2\tilde{C}_{i,j+1} + \tilde{C}_{i-1,j+1} - \tilde{C}_{i+1,j-1} - 2\tilde{C}_{i,j-1} - \tilde{C}_{i-1,j-1}) / \delta y$$

Interface normal direction

Volume of fraction

Interface is reconstructed!

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

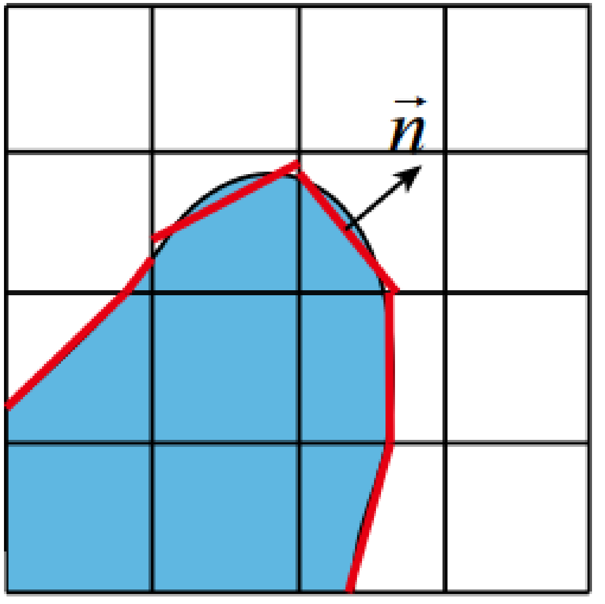


By the reconstruction scheme, the phase interface is determined in each computational cell.

Piecewise linear interface calculation (PLIC)

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

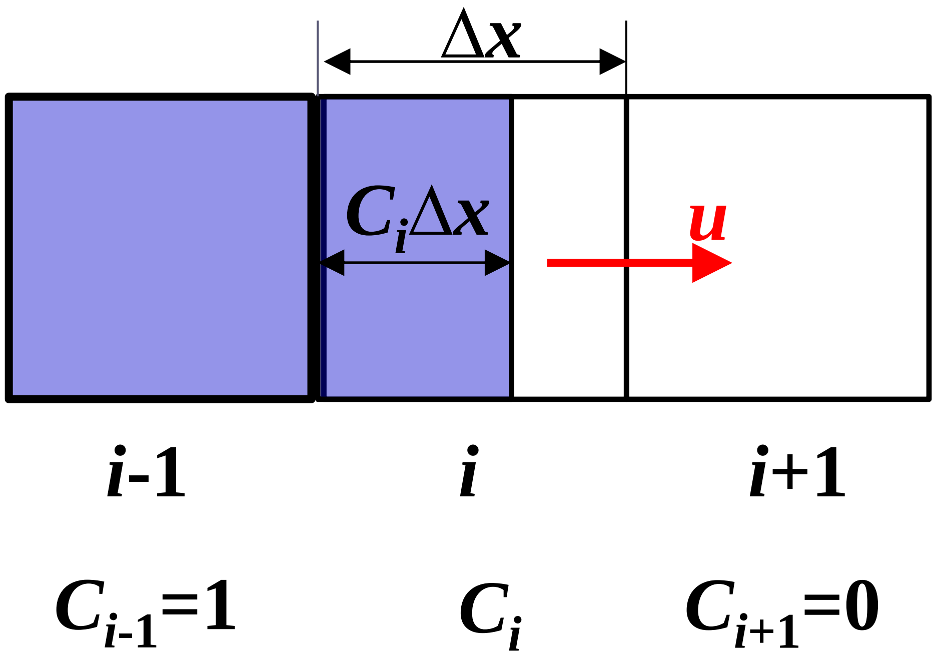
D.L. Youngs, Time-dependent multi-material flow with large fluid distortion, Numerical methods for Fluid Dynamics, 1982, 24(2), 273-285

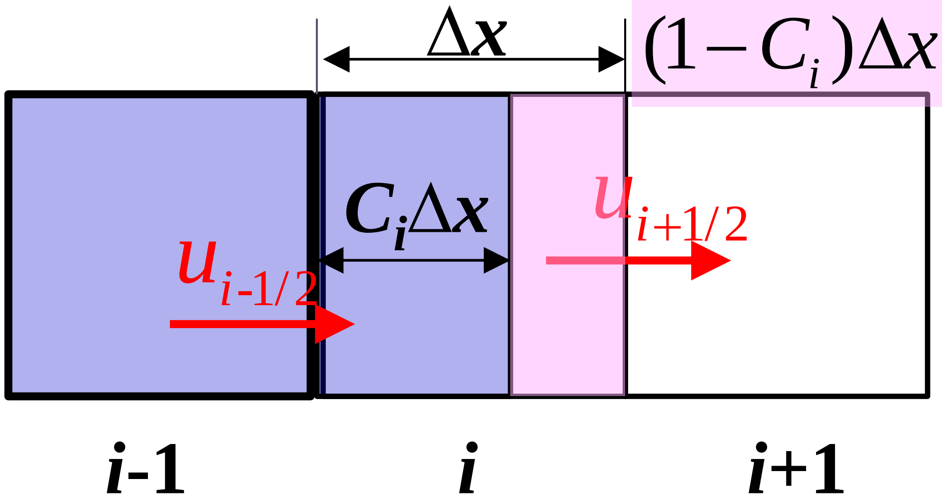


Step 2. Interface advection in a given velocity field

Based on the reconstructed phase interface, calculate the variation of *C* according to local velocity 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

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

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

$$\frac{\partial C_{\mathrm{l}}}{\partial t}+\mathbf{u}\cdot\nabla C_{\mathrm{l}}=\boxed{\frac{S_{\mathrm{l}}}{\rho}}$$

$$S_{\mathrm{v}}=-S_{\mathrm{l}}=\alpha_{\mathrm{g}}\rho_{\mathrm{g}}\frac{T-T_{\mathrm{sat}}}{T_{\mathrm{sat}}}$$

for condensation (T_{sat} > T)

$$\frac{\partial C_{\mathrm{v}}}{\partial t}+\mathbf{u}\cdot\nabla C_{\mathrm{v}}=\boxed{\frac{S_{\mathrm{v}}}{\rho}}$$

$$S_{\mathrm{v}}=-S_{\mathrm{l}}=\alpha_{\mathrm{l}}\rho_{\mathrm{l}}\frac{T-T_{\mathrm{sat}}}{T_{\mathrm{sat}}}$$

for evaporation (T_{sat} < T)

W.H. Lee, A pressure iteration scheme for two-phase flow modelling, in: T.N. Veziroglu (Ed.), Multiphase Transport Fundamentals, Reactor Safety, Applications, Hemisphere, Washington, DC, 1980, pp. 61–82.

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

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

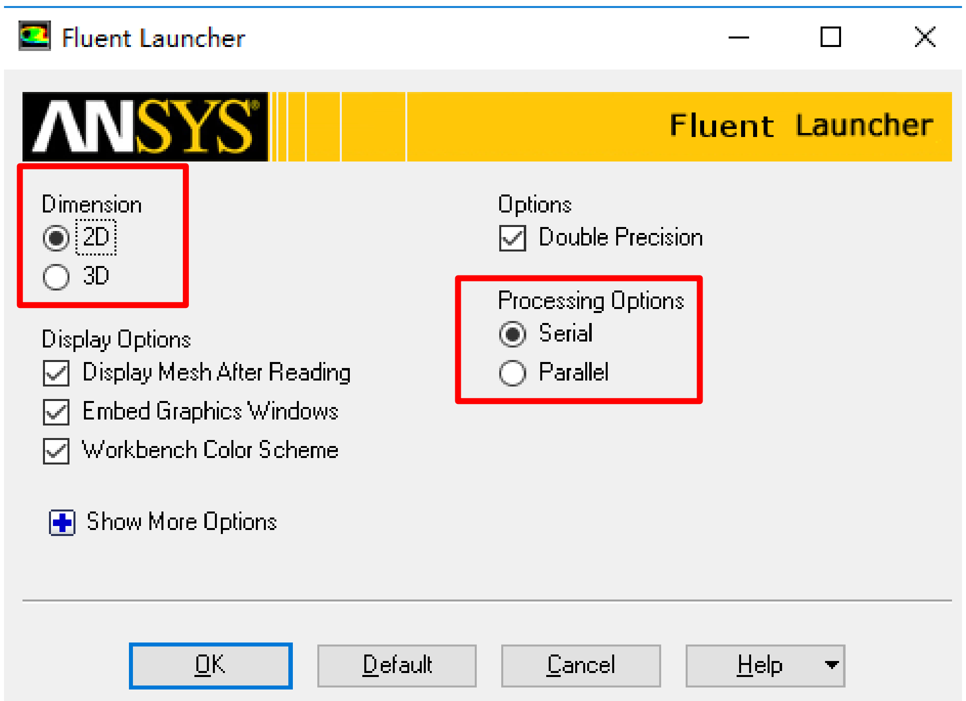
$$\frac{\partial(\rho C_p T)}{\partial t}+(\mathbf{u}\cdot\nabla)(\rho C_p T)=\nabla(\lambda\nabla T)+\boxed{S_l h}$$

$$\frac{\partial C_{\mathrm{l}}}{\partial t}+\mathbf{u}\cdot\nabla C_{\mathrm{l}}=\boxed{\frac{S_{\mathrm{l}}}{\rho}}$$

$$\frac{\partial C_{\mathrm{v}}}{\partial t}+\mathbf{u}\cdot\nabla C_{\mathrm{v}}=\boxed{\frac{S_{\mathrm{v}}}{\rho}}$$

2 Process of simulation

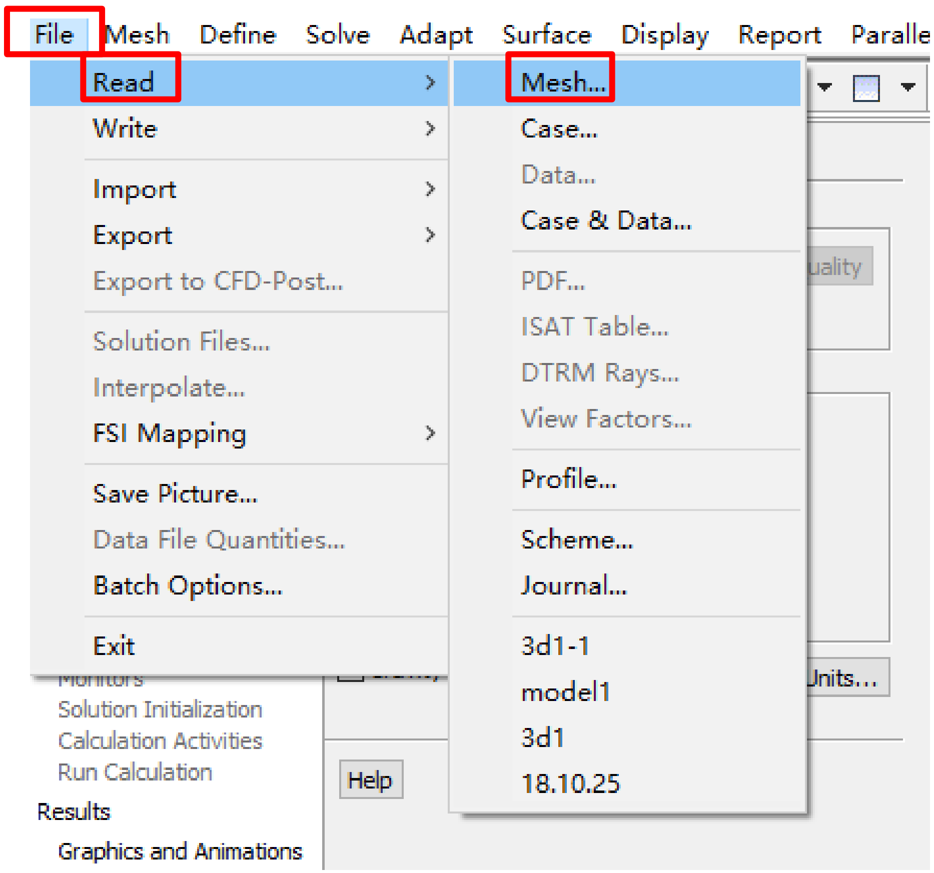
2.1 Launch ANSYS Fluent



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

2.2 Read the mesh

File → Read→Mesh

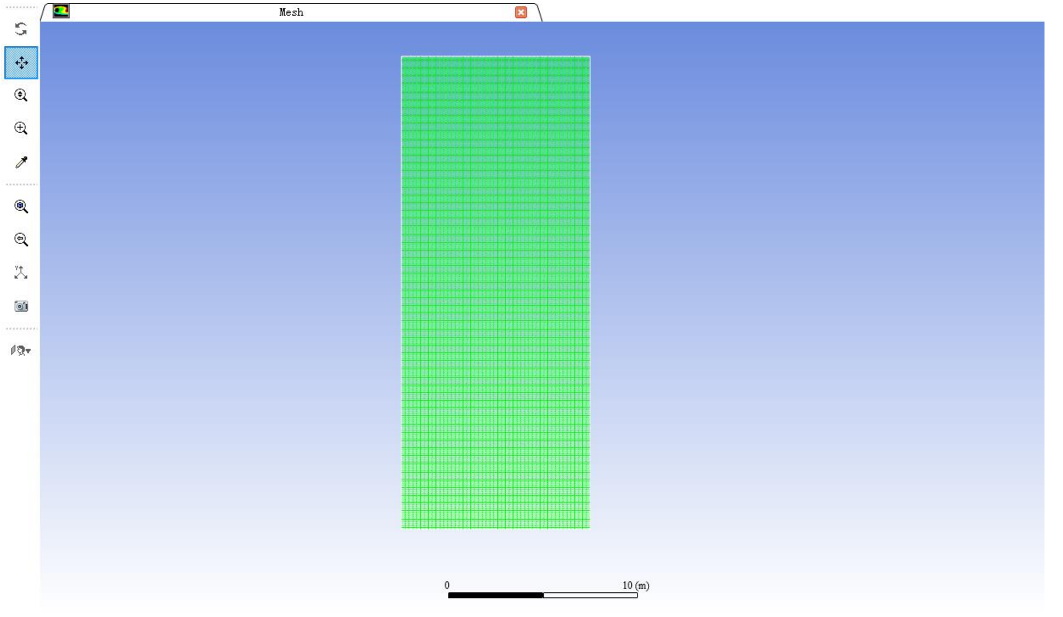



```
Building...
     mesh
     materials,
     interface,
     domains,
     zones,
        water
        gdl
        wall
        air-in
        air-out
        int_fluid
        fluid
Done.

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

2.3 Check the mesh

General→Mesh→Check

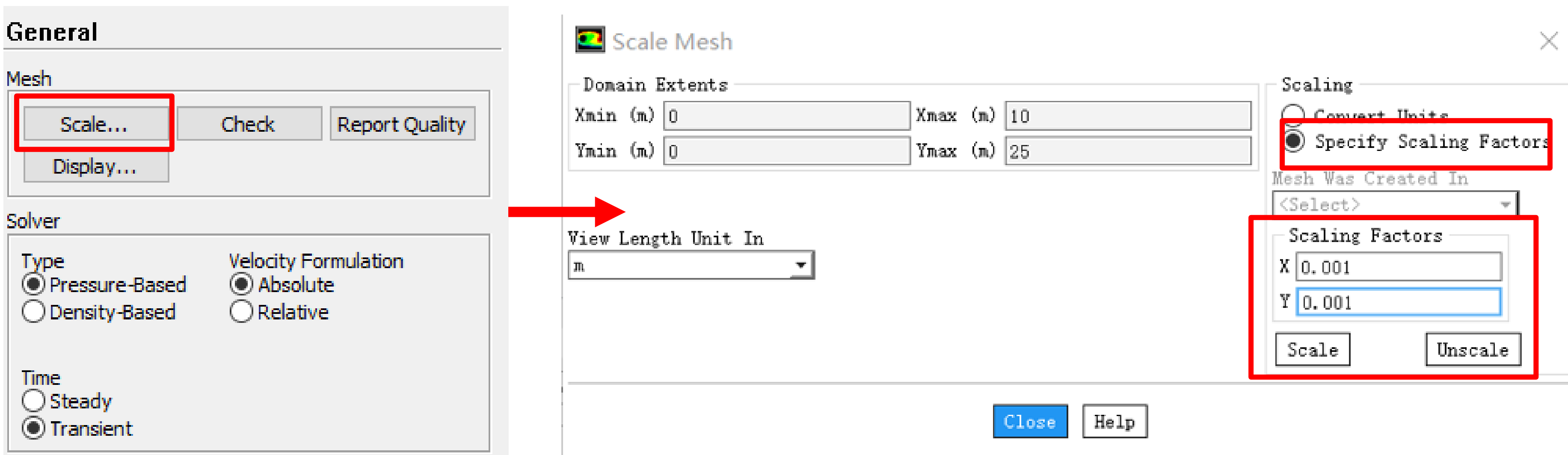



```
Mesh Check

 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

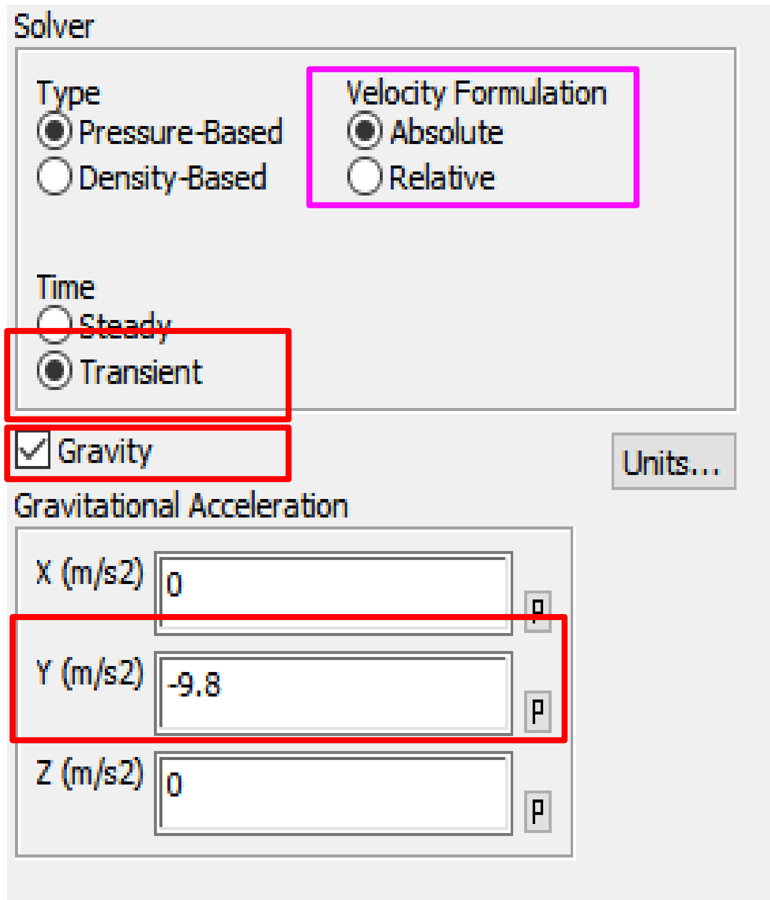
General→Mesh→Scale



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

2.5 Choose the solver

General→Solver



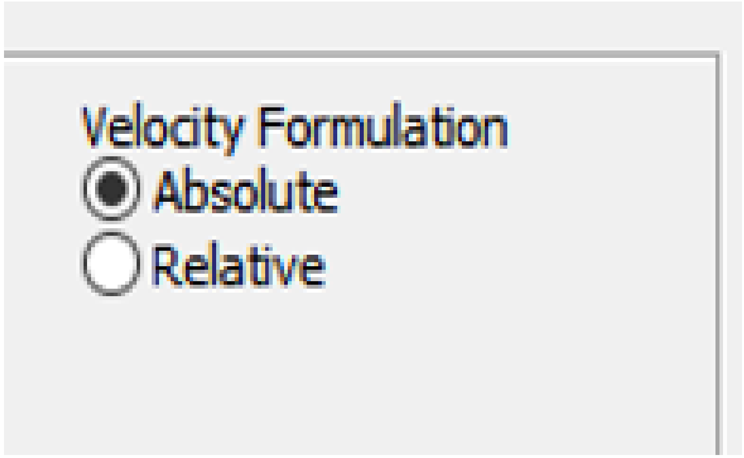
- Choose **Transient**

  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.

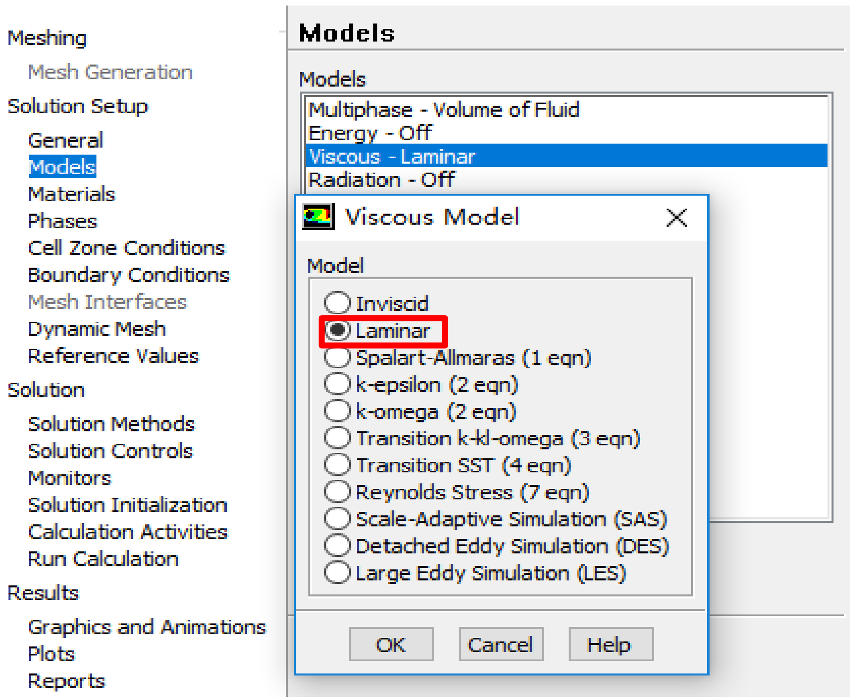
The velocity formulation resulting in most of the flow domain having the smallest velocities is recommended, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution.

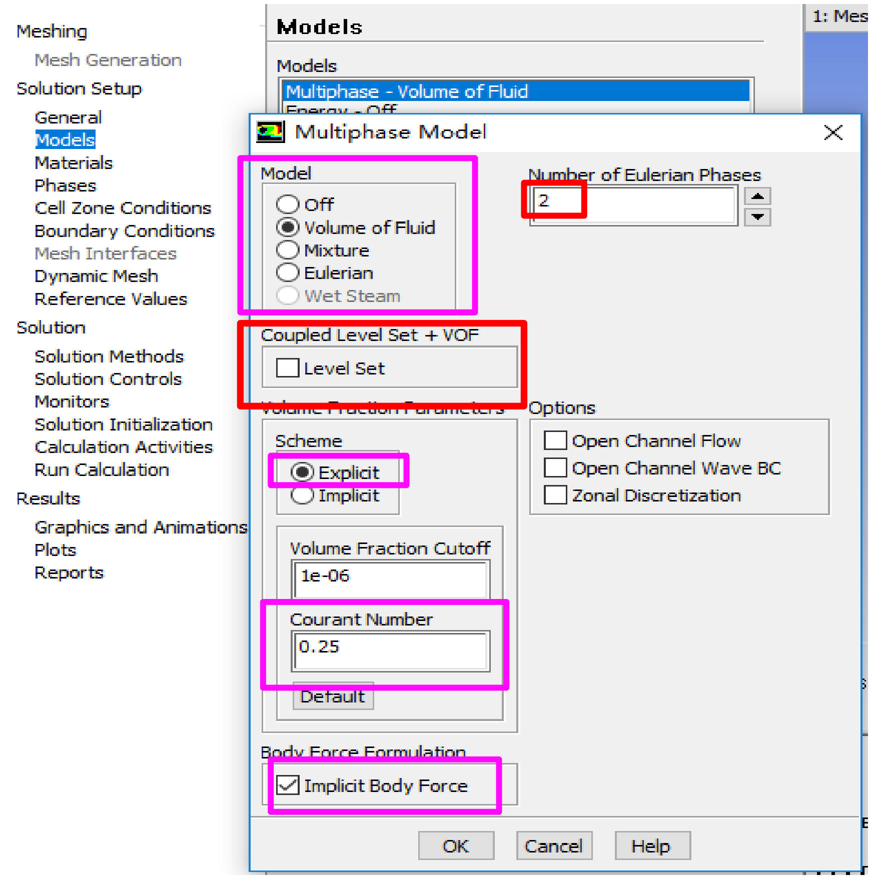


- **The absolute velocity formulation is preferred in applications where the flow in most of the domain is not moving (for example, a fan in a large room).**
- **The relative velocity formulation is appropriate when most of the fluid in the domain is moving, as in the case of a large impeller in a mixing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

2.6 Choose the models

Solution Setup→Models

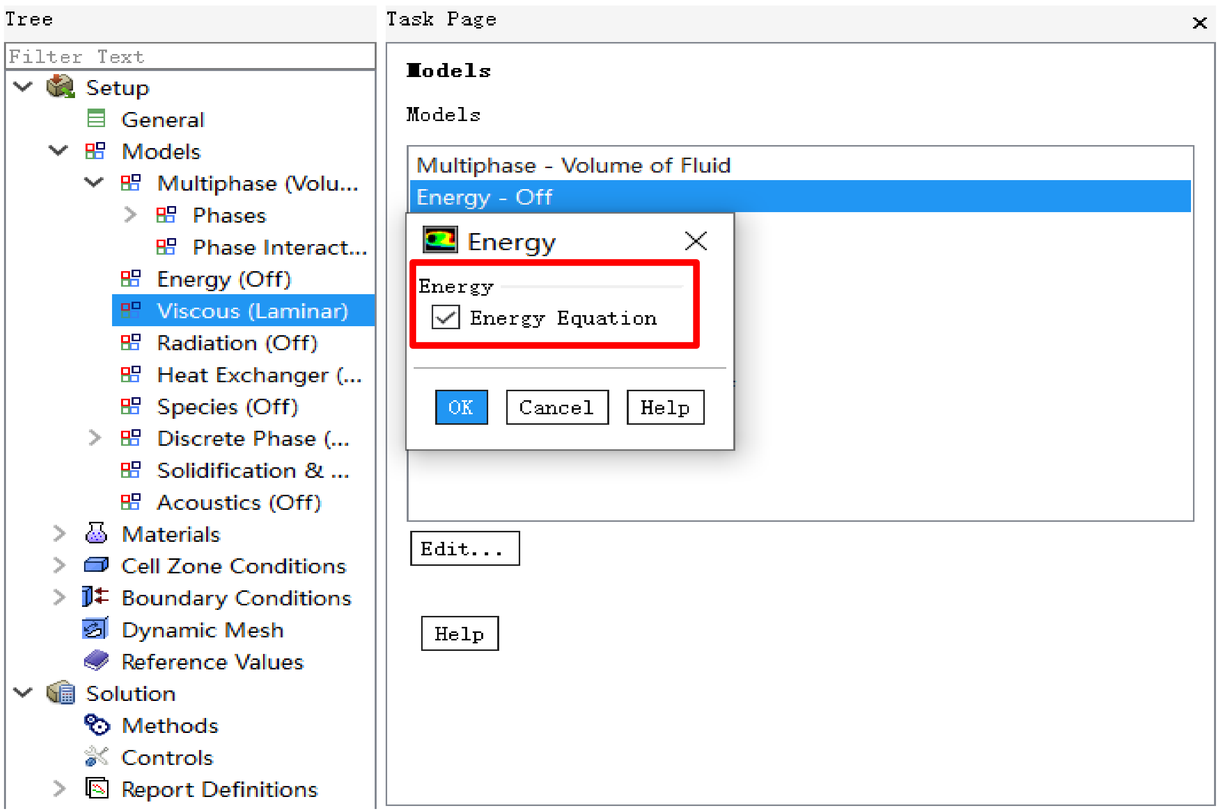




■ **Choose Volume of Fluid as Multiphase Model**

2.6 Choose the models

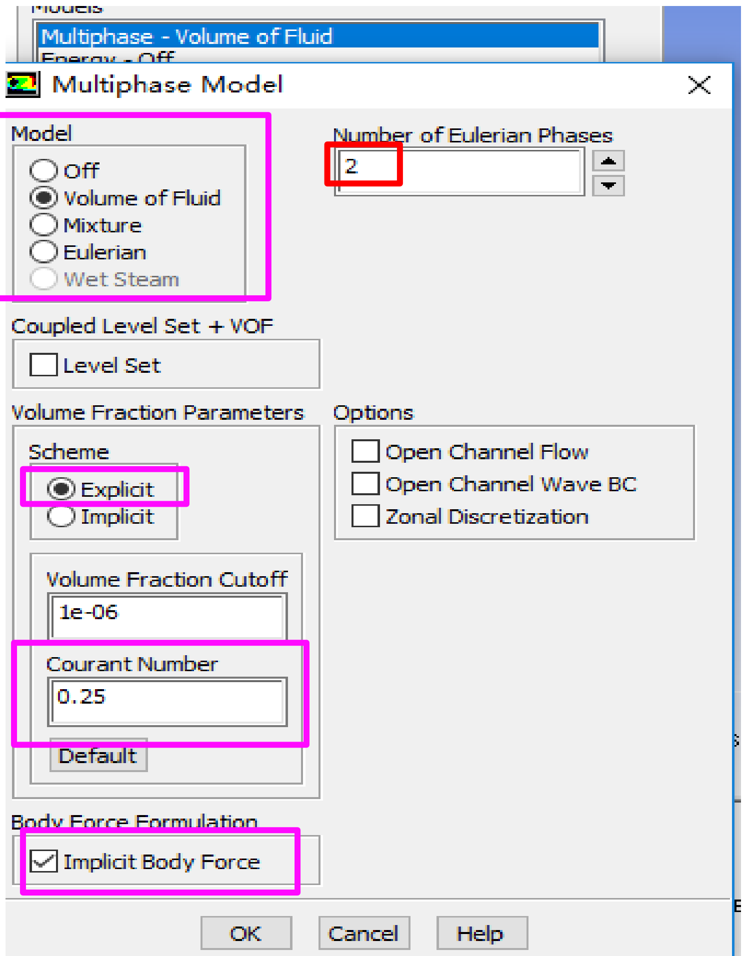
Solution Setup→Models



- 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.

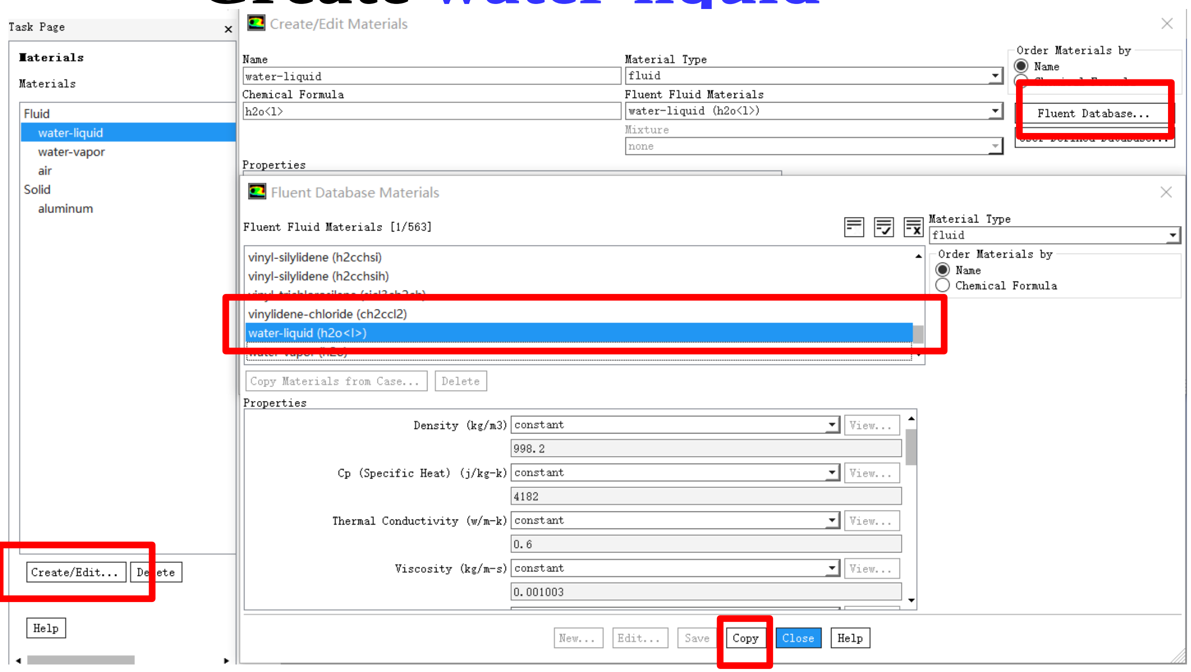
$$\overbrace{\frac{\partial(\rho\mathbf{u})}{\partial t}+\nabla\cdot(\rho\mathbf{u}\mathbf{u})}^{\text{Inertial force}} = \underbrace{-\nabla p}_{\text{Pressure}} + \overbrace{\nabla\cdot[\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^{\mathrm{T}})]}^{\text{Viscous force}} + \overset{\text{Body force}}{\boxed{\underbrace{\rho\mathbf{g}}_{\text{gravity}}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

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

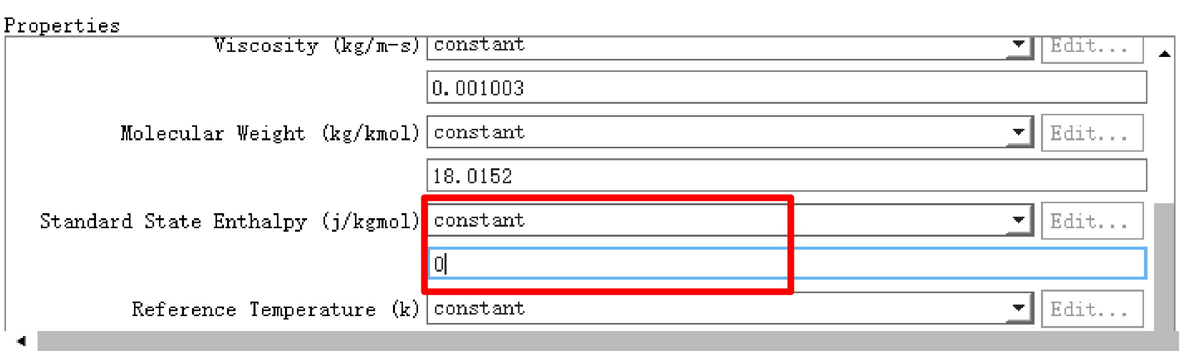
2.7 Define the materials

Solution Setup→ Materials→Create/Edit Material

- **Create water-liquid**



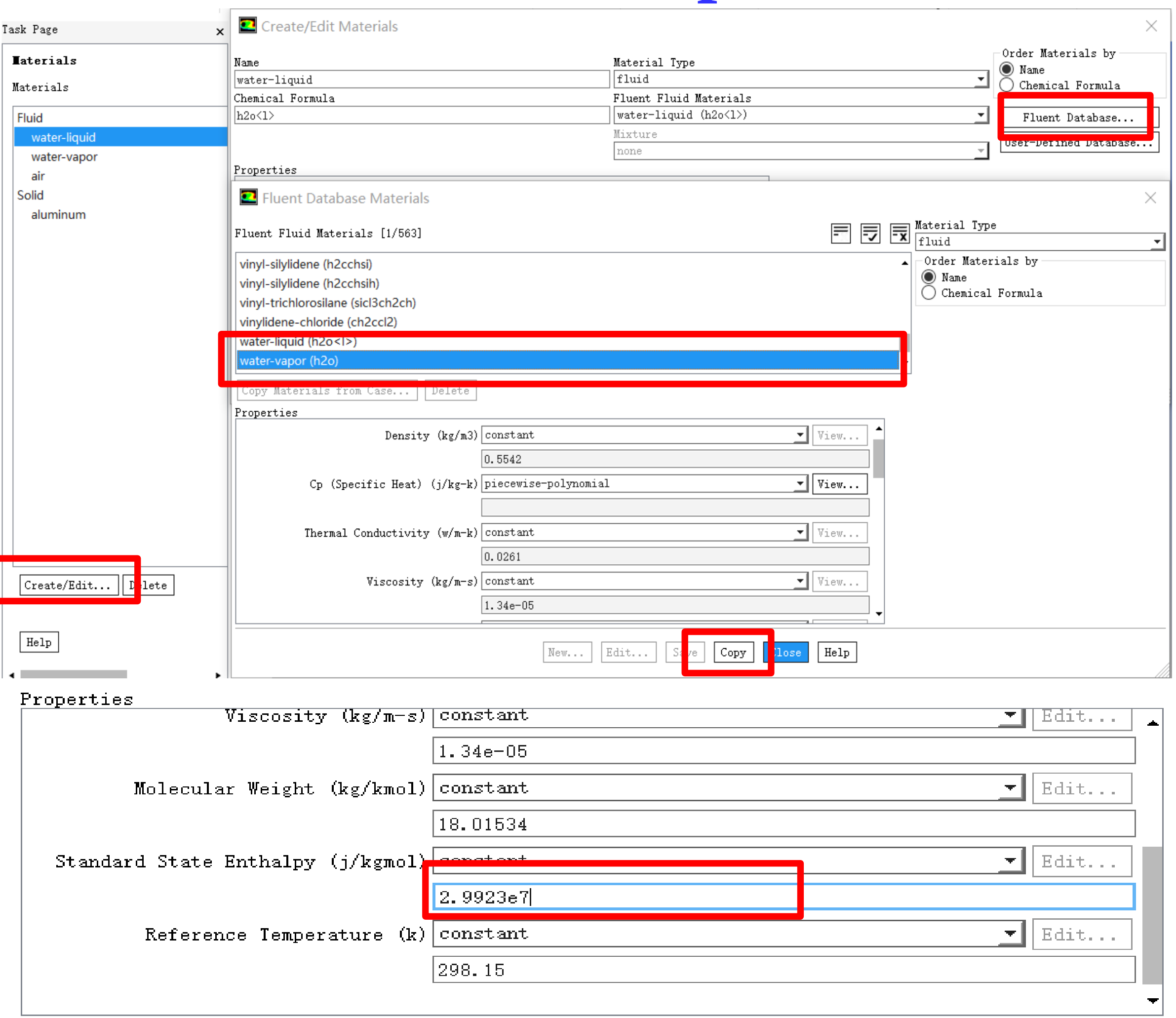
1. Click **Fluent Database**
2. Choose **water-liquid**
3. Set **Standard State Enthalpy** to **0**
4. Click **Copy**



2.7 Define the materials

Solution Setup→ Materials→Create/Edit Material

- Create water-vapor



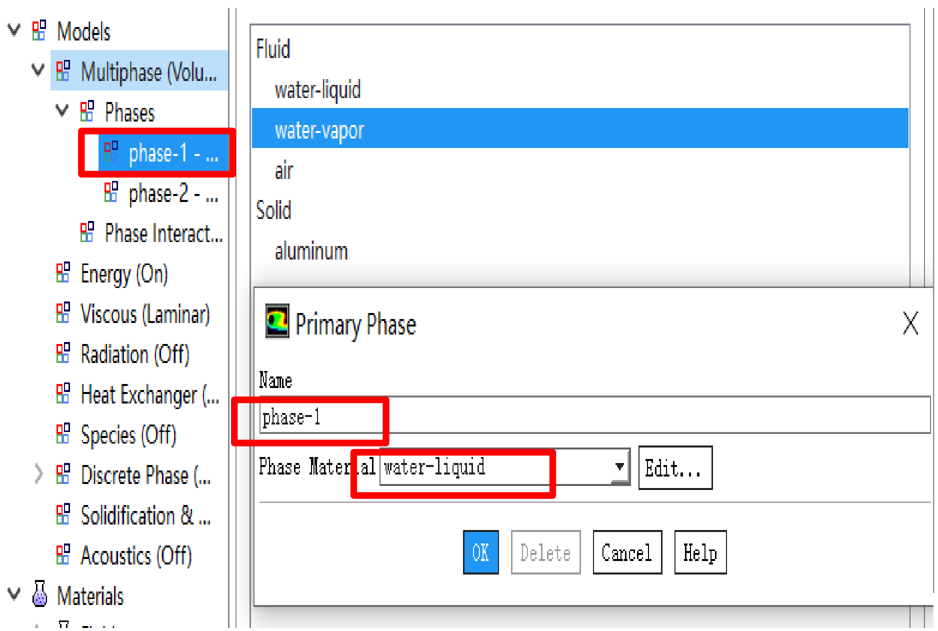
1. Click Fluent Database
2. Choose water-vapor
3. Set Standard State Enthalpy to 2.9923e7
4. Click Copy

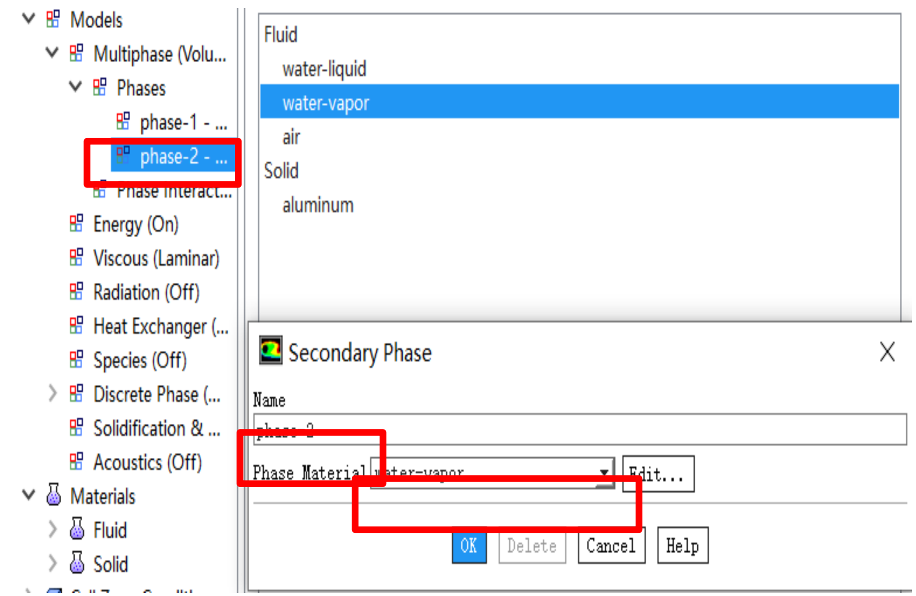
The difference between the Standard State Enthalpy of two phases is the latent heat of phase change.

2.8 Define the phases

Solution Setup→ Phases

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

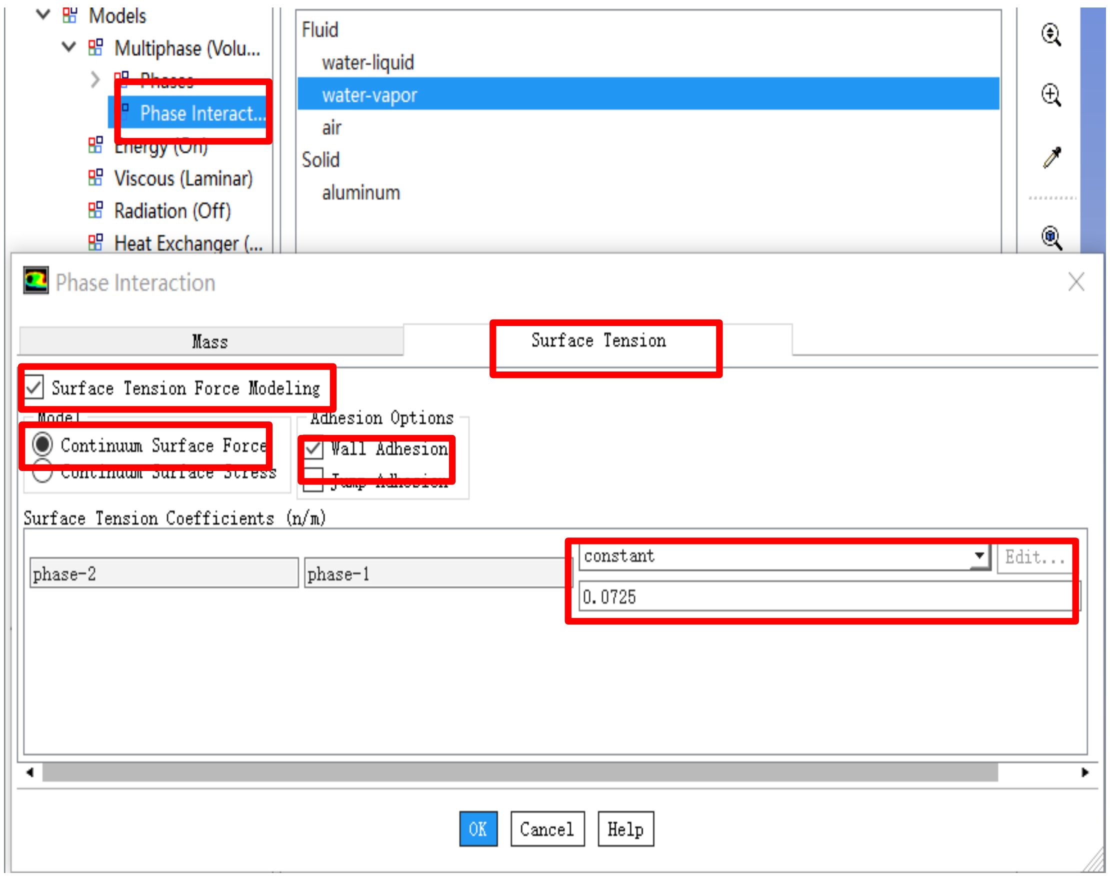




Primary phase is usually set as the one dominated in the computational domain.

2.8 Define the phases

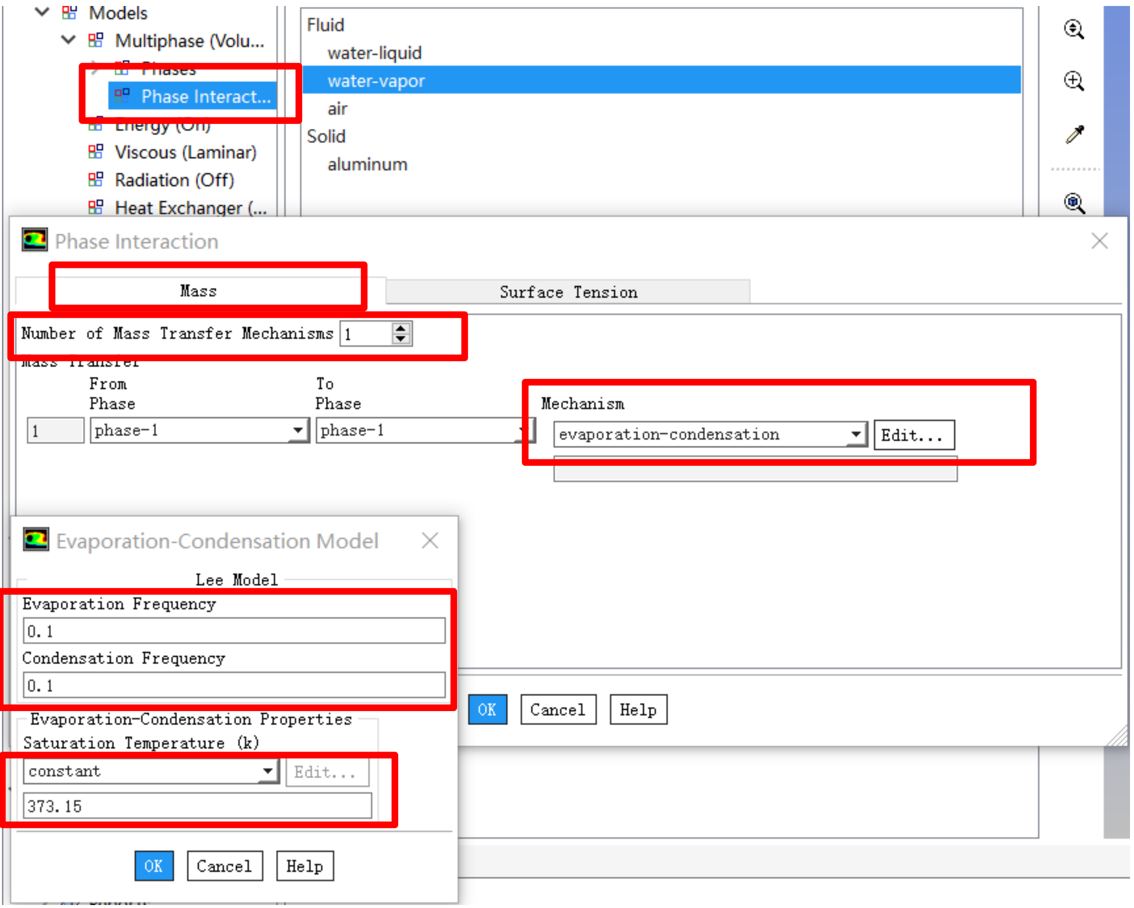
Define surface tension force



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**

2.8 Define the phases

phase-change model: Lee Model

$$\frac{\partial C_{\mathrm{l}}}{\partial t} + \mathbf{u} \cdot \nabla C_{\mathrm{l}} = S_{\mathrm{l}}$$

$$\frac{\partial C_{\mathrm{v}}}{\partial t} + \mathbf{u} \cdot \nabla C_{\mathrm{v}} = S_{\mathrm{v}}$$

$$S_{\mathrm{v}} = -S_{\mathrm{l}} = \alpha_{\mathrm{g}} \rho_{\mathrm{g}} \frac{T - T_{\mathrm{sat}}}{T_{\mathrm{sat}}}$$

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

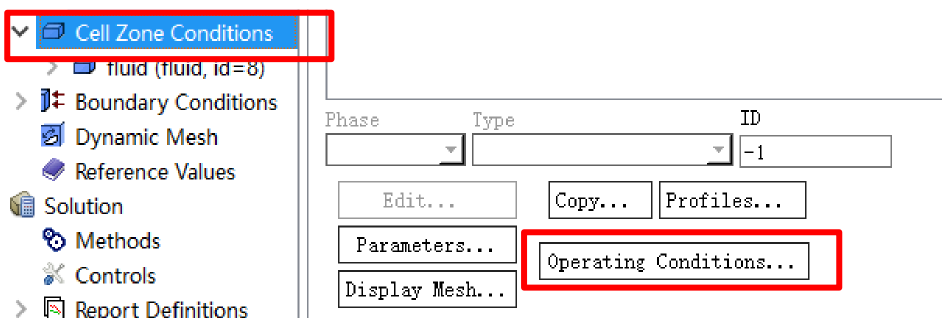
$$S_{\mathrm{v}} = -S_{\mathrm{l}} = \alpha_{\mathrm{l}} \rho_{\mathrm{l}} \frac{T - T_{\mathrm{sat}}}{T_{\mathrm{sat}}}$$

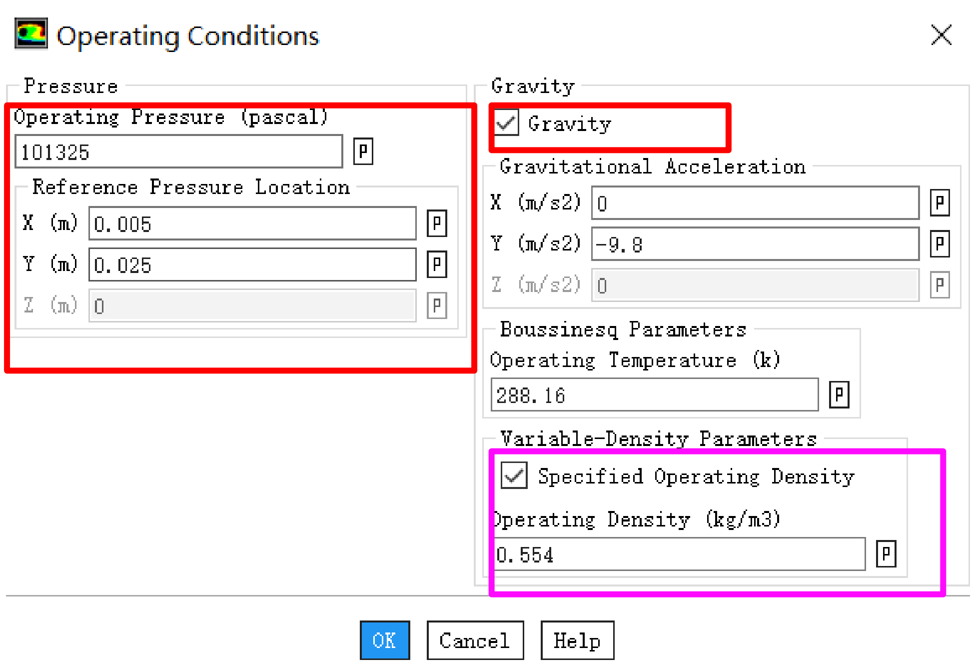
for evaporation ($T_{\mathrm{sat}} < \mathrm{T}$)

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

2.9 Define cell zone conditions

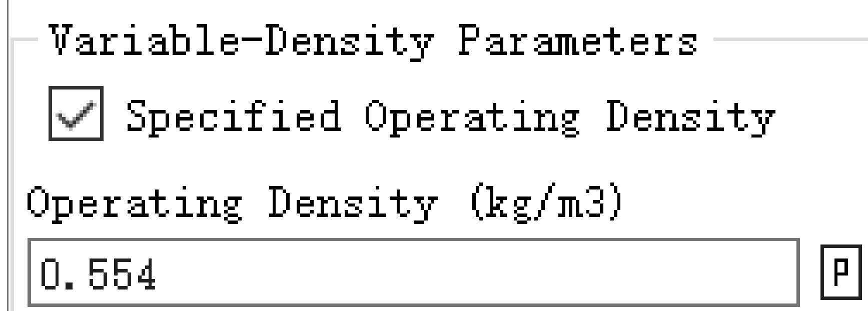
Solution Setup→ Cell Zone Condition →Operating Conditions





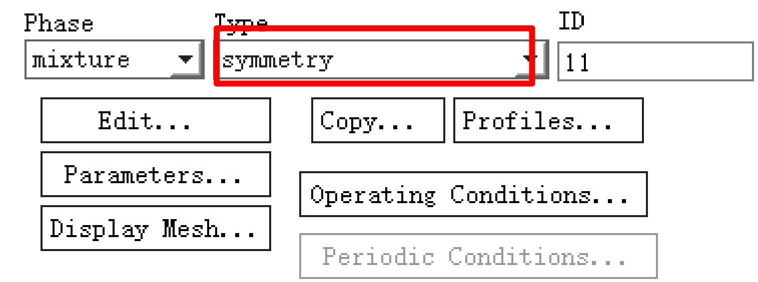
Specified operating density

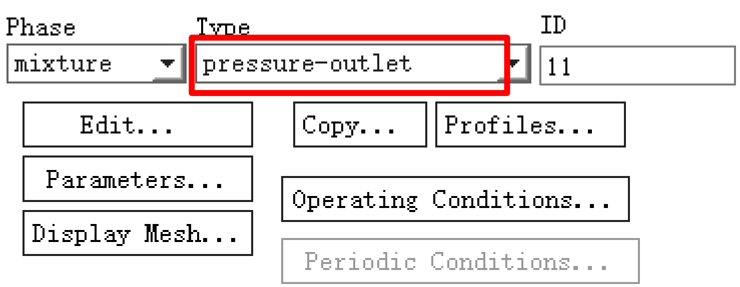
Set the operating density to be the density of the lightest phase. Here input the density of the water-vapor.



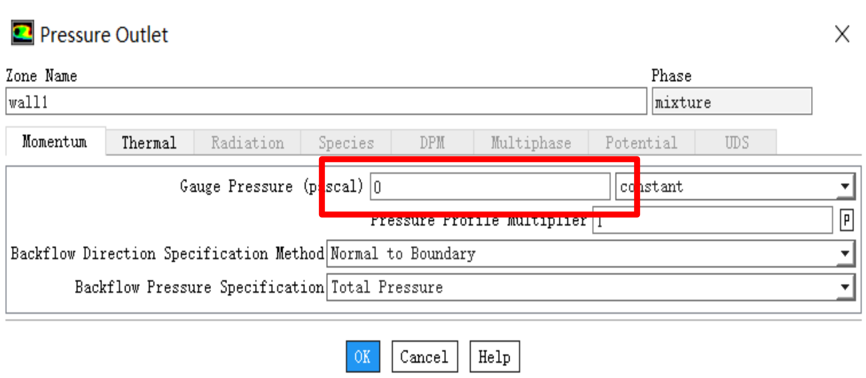
2.10 Define the boundary conditions

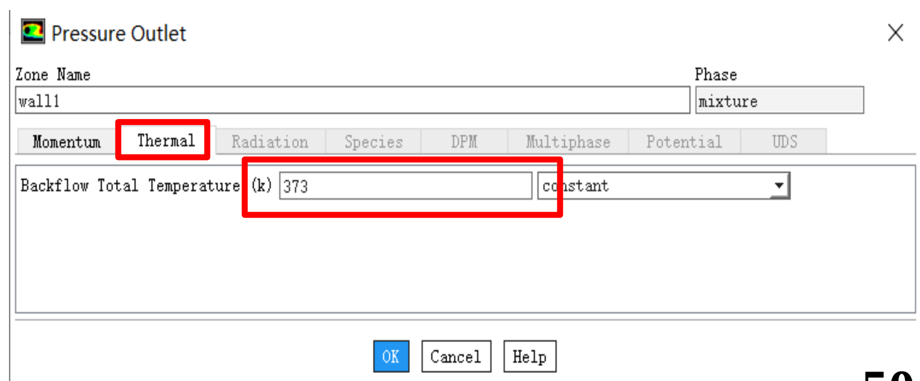
Solution Setup→ Boundary Condition





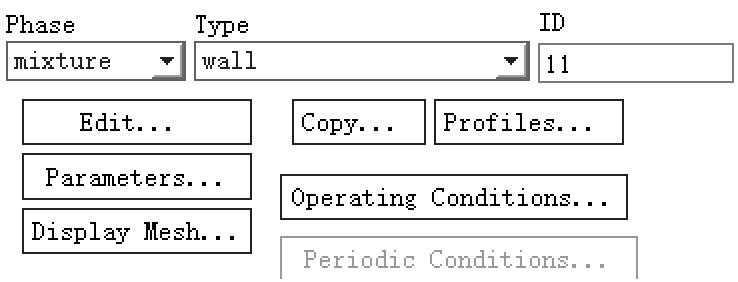
- **For the boundary on both sides, define symmetry condition.**
- **For the outlet, define pressure outlet**
- **Set backflow temperature**

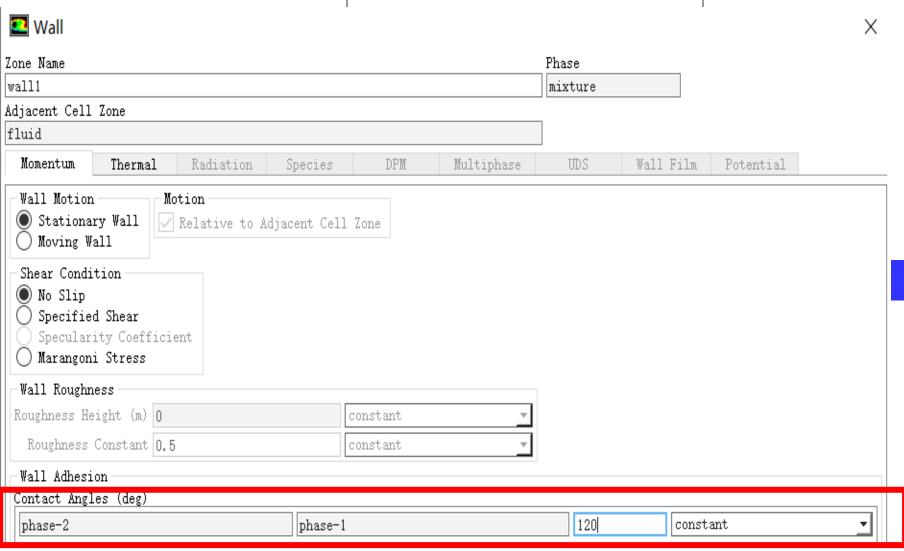




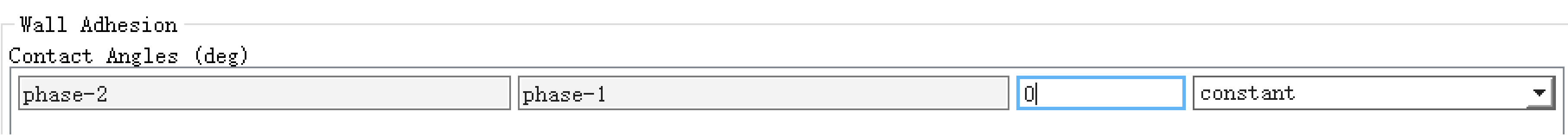
2.10 Define the boundary conditions

For the bottom surface, Define contact angle



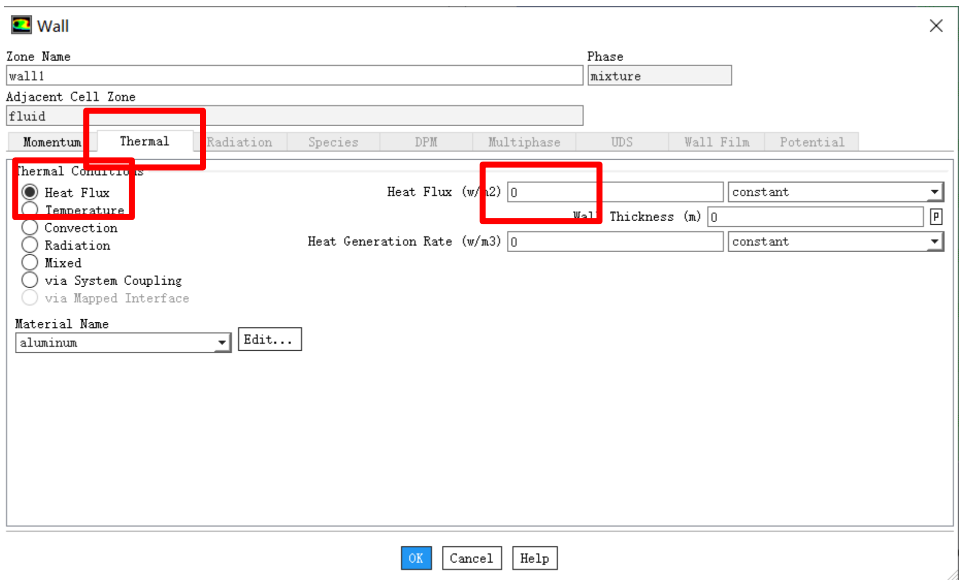


- Choose **wall** as **Type**
- Input value of the **contact angle 120º for adiabatic wall, 10º for high temperature wall.**
- **The angle is measured by vapor here.**

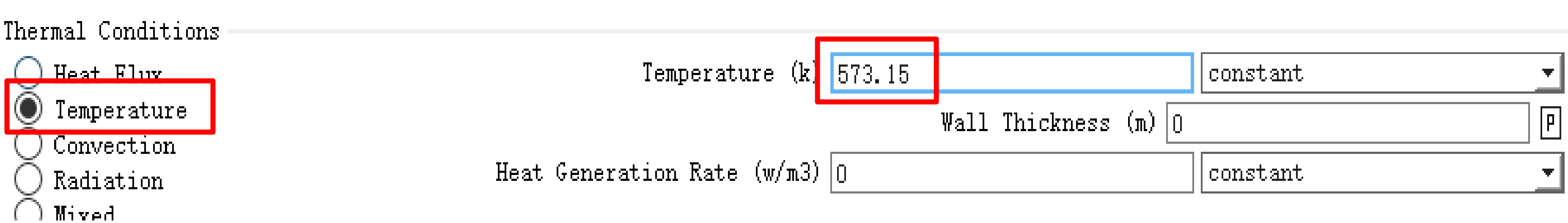


2.10 Define the boundary conditions

For the bottom surface, Define heat boundary condition

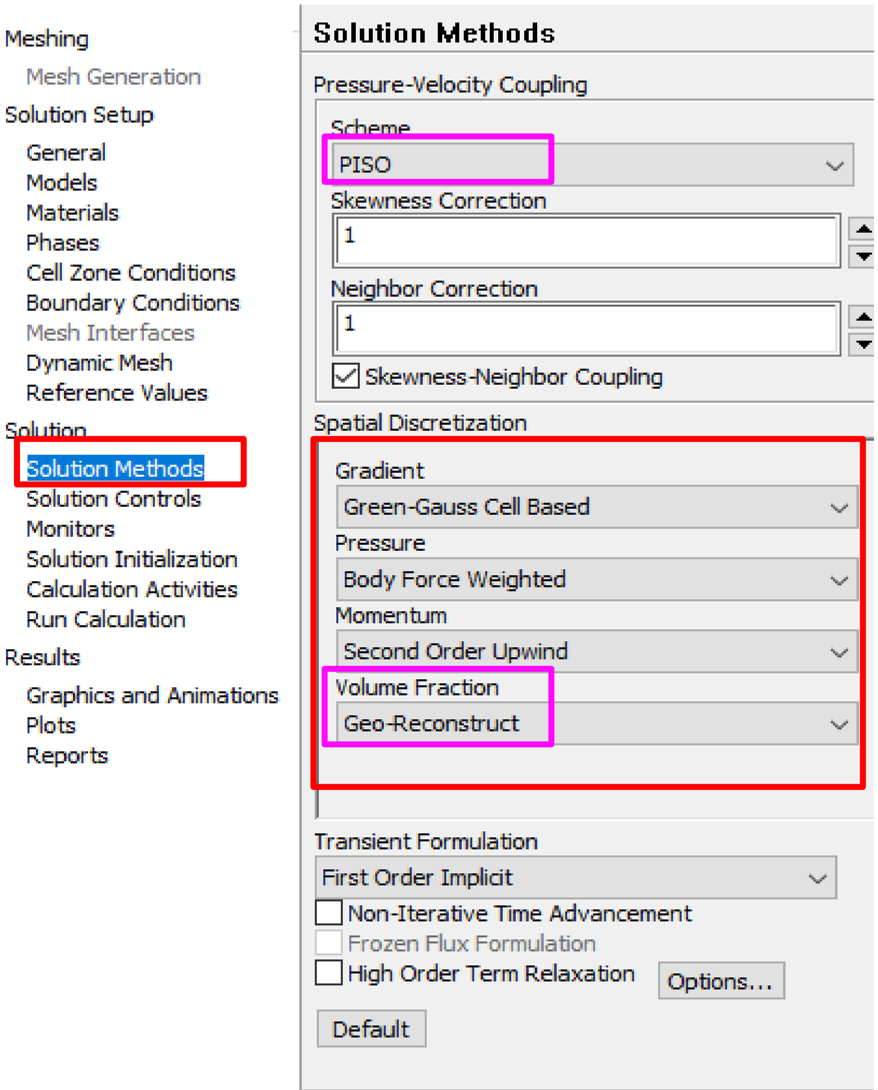


- For the adiabatic wall, set heat flux to **zero.**
- For the high temperature wall, set temperature to **573.15 K.**



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

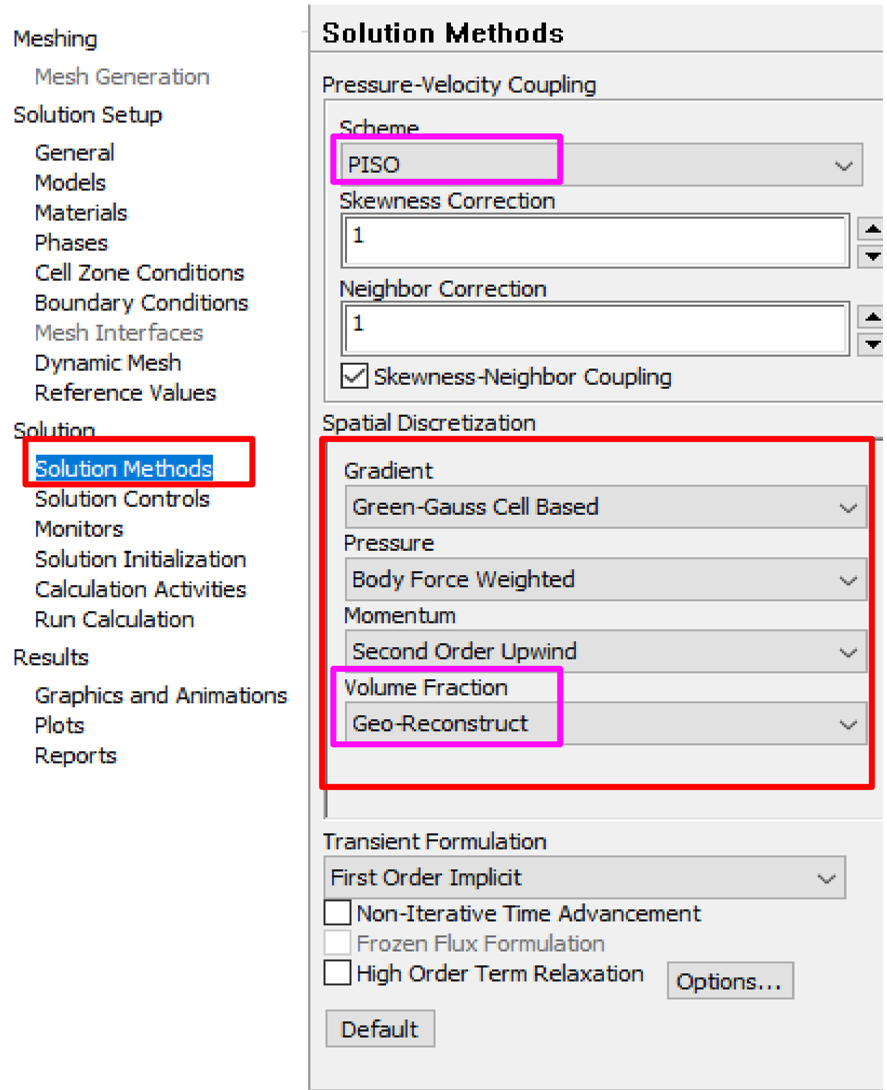
Correction step: solve pressure correction, obtain u and v that satisfying the Mass Conservation Equation.

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

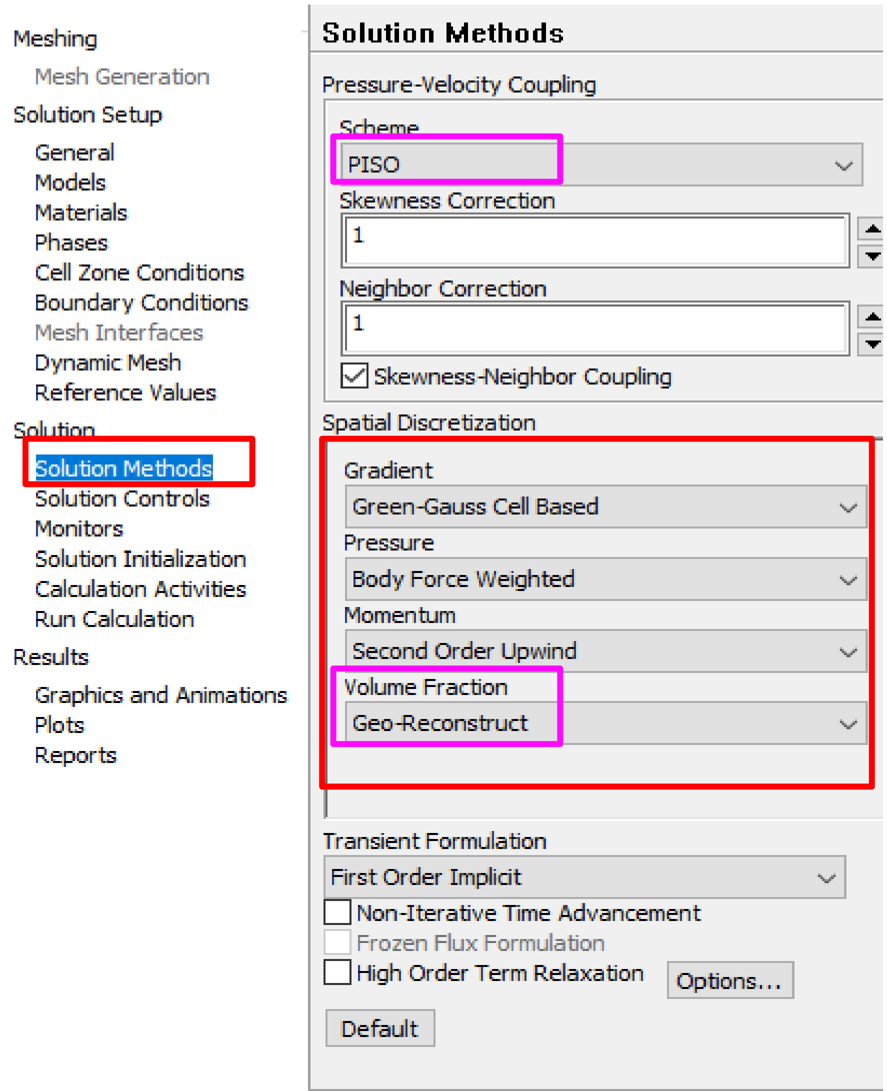
$$<\nabla\phi>=\frac{1}{V_C}\int_{V_C}\nabla\phi dV=\frac{1}{V_C}\int\phi\cdot\mathbf{n}dS=\sum\phi_f\cdot\mathbf{n}S$$

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.

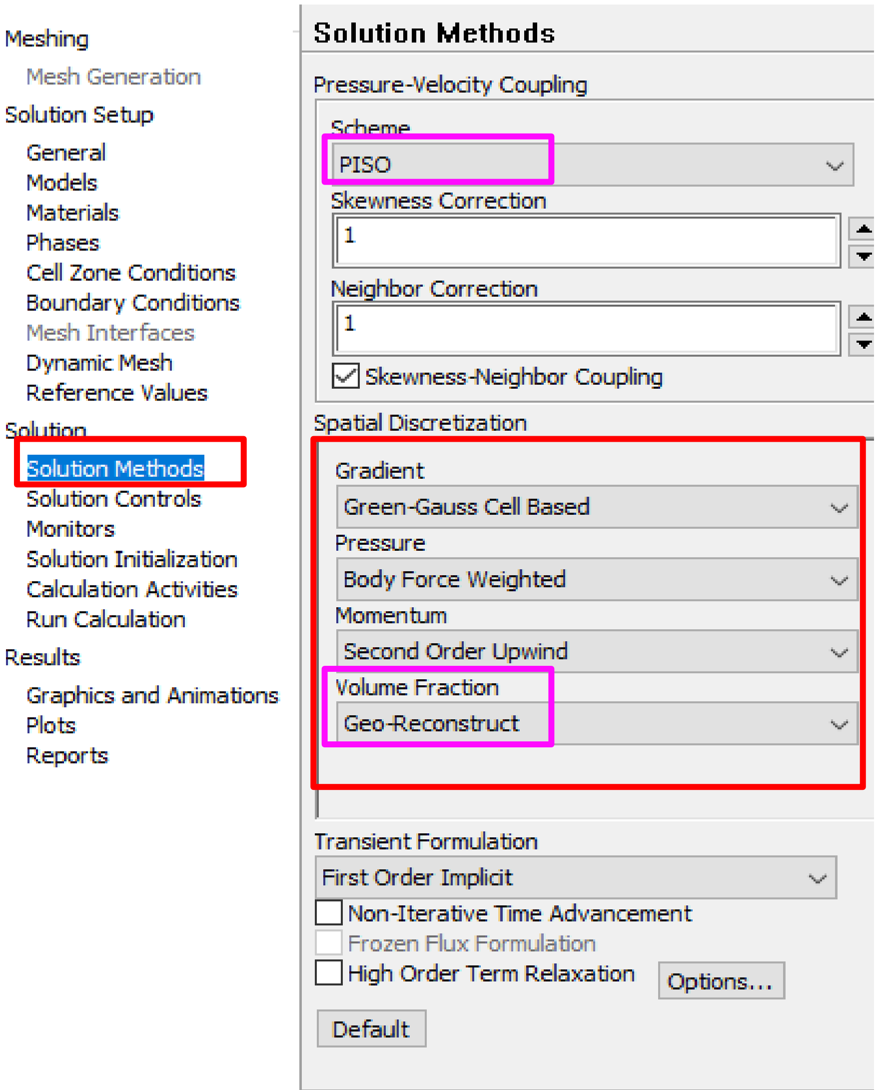
- ✓ Multiphase flow such as VOF (Volume of Fluid，体积函数法) or LS (Level Set, 水平集): recommended.
- ✓ For porous media: not recommended!

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

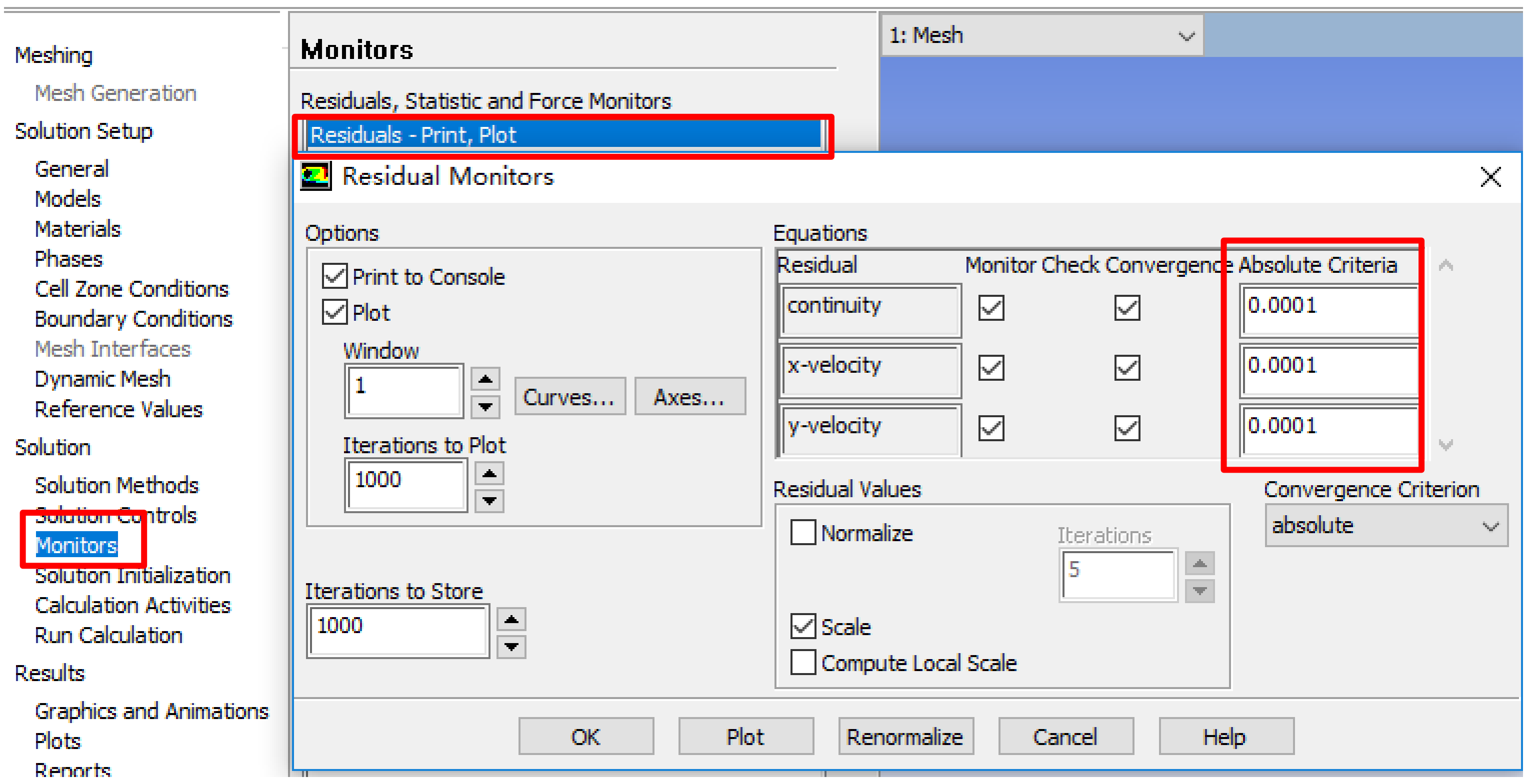
The geometric reconstruction interpolation scheme recommended when time-accurate transient behaviors of the multiphase are required. In other words, it can accurately predict the sharp interface. This scheme is the *most accurate* and is *applicable for general unstructured meshes*.

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

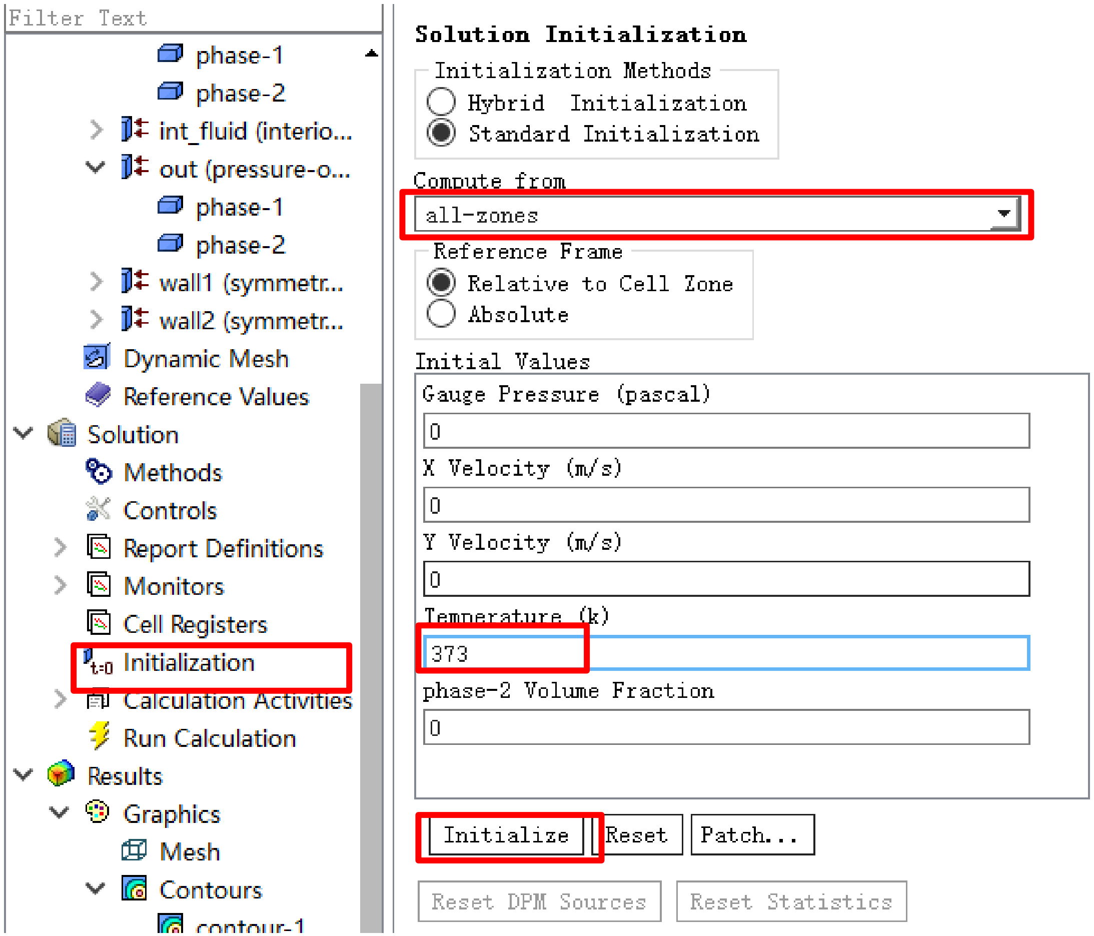
Solution → Monitors

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



2.13 Initialization

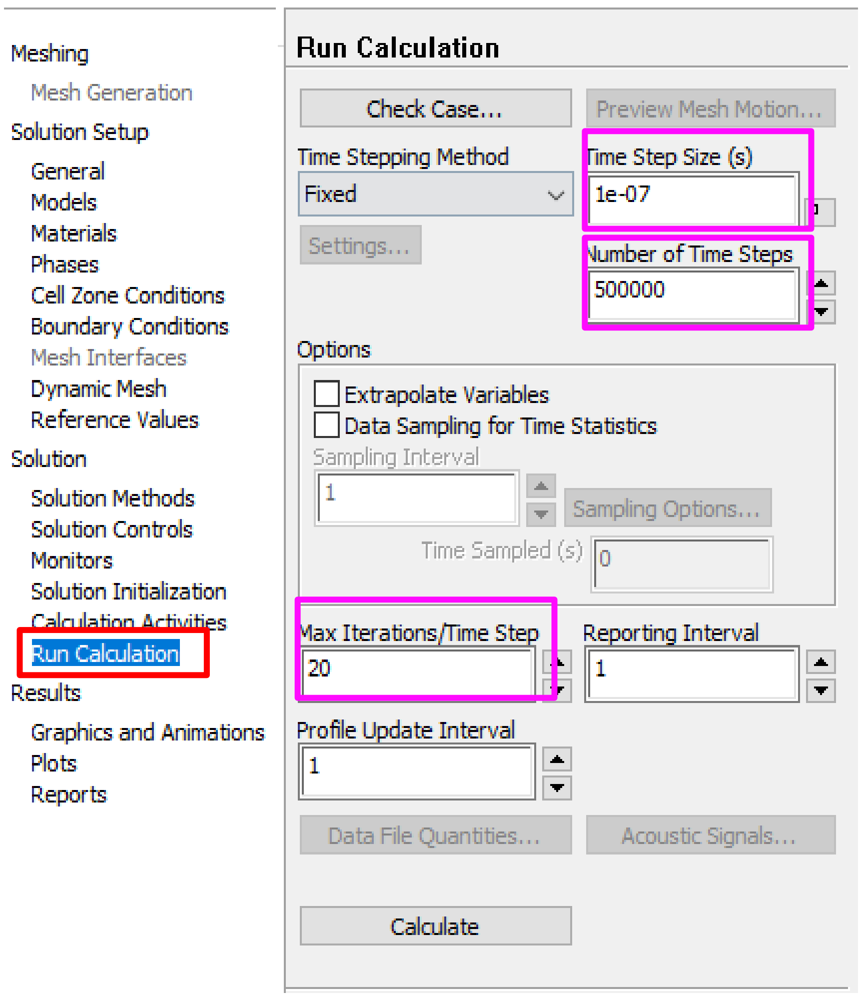
Solution → Solution Initialization



- Choose **Standard Initialization**
- Choose **all-zones**
- Write **373**
- Click **Initialize**

2.14 Run calculation

Solution → Run Calculation

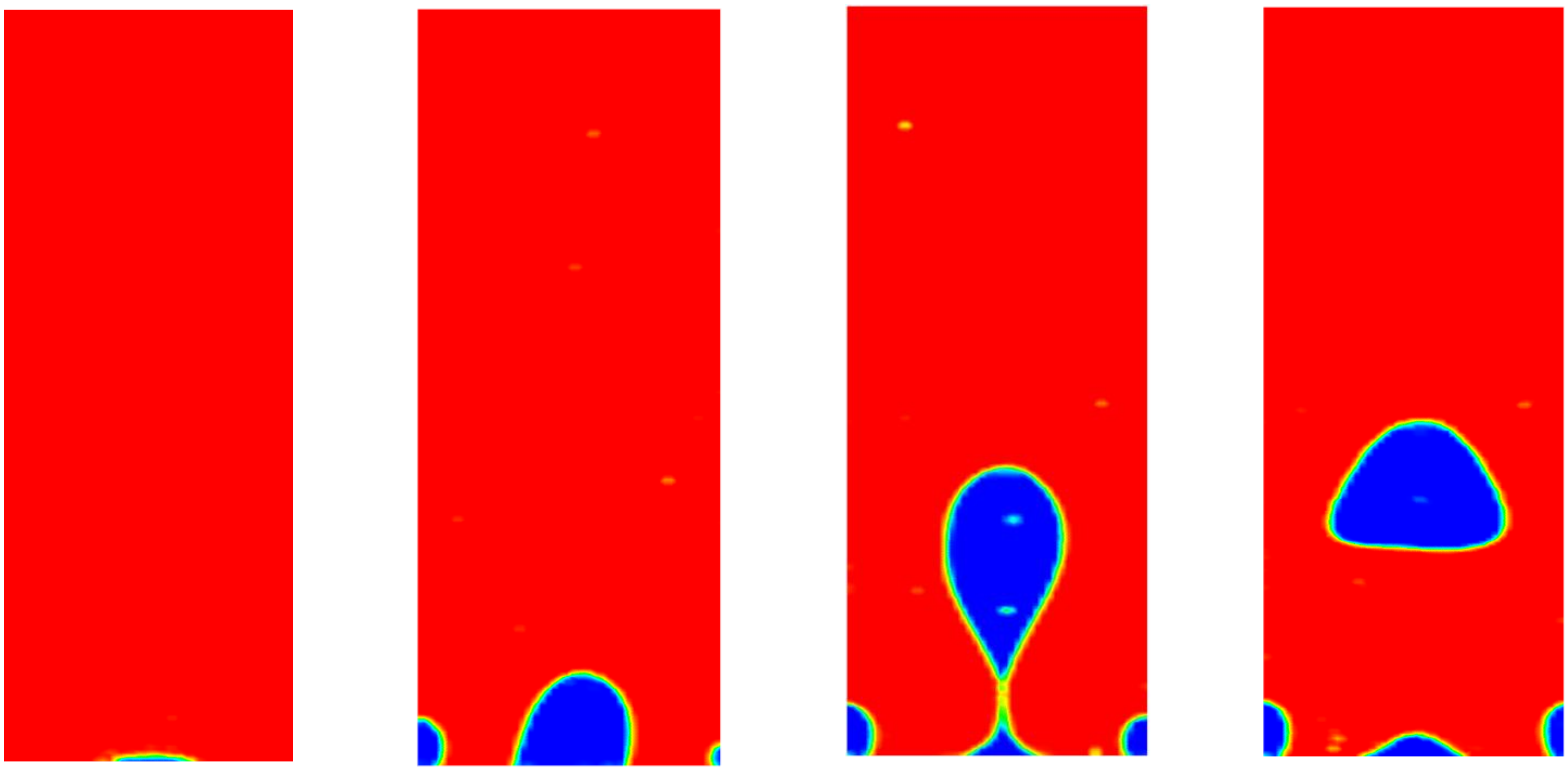


- Write **Number** in Time Step Size box
- Write **Number** in Number of Time Step box according to situations
- Write **Number** in Max Iterations according to situations

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



t=0 s *t*=1.5 s *t*=8.5 s *t*=8.7 s

Fig.2 vapor behavior

3 Results

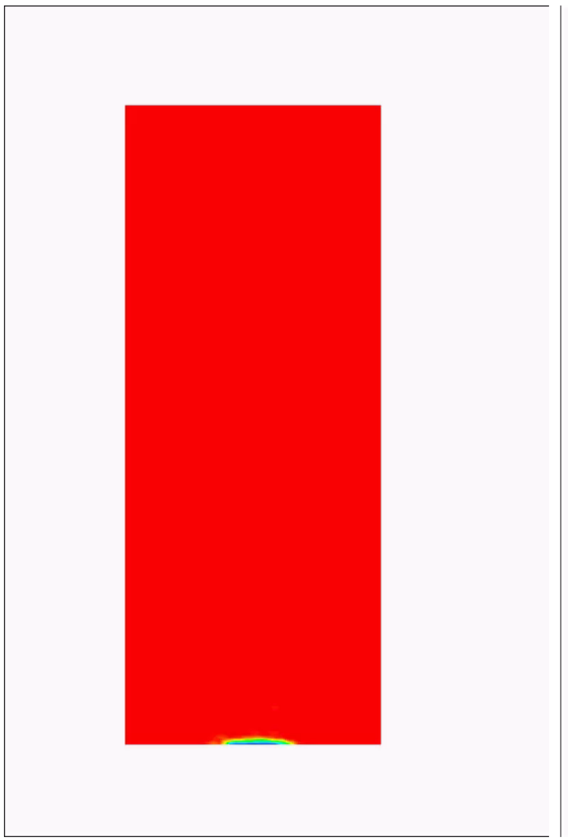


Fig.3 Boiling process

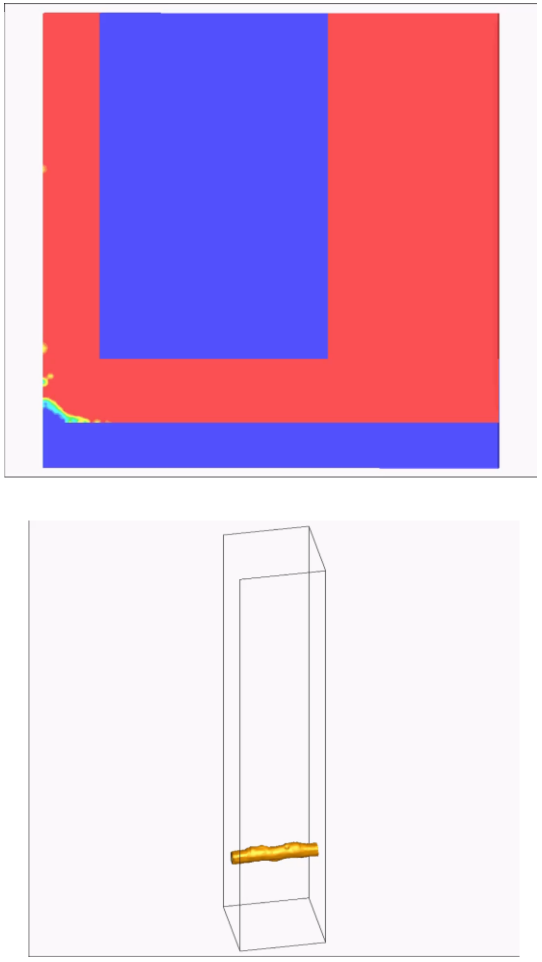
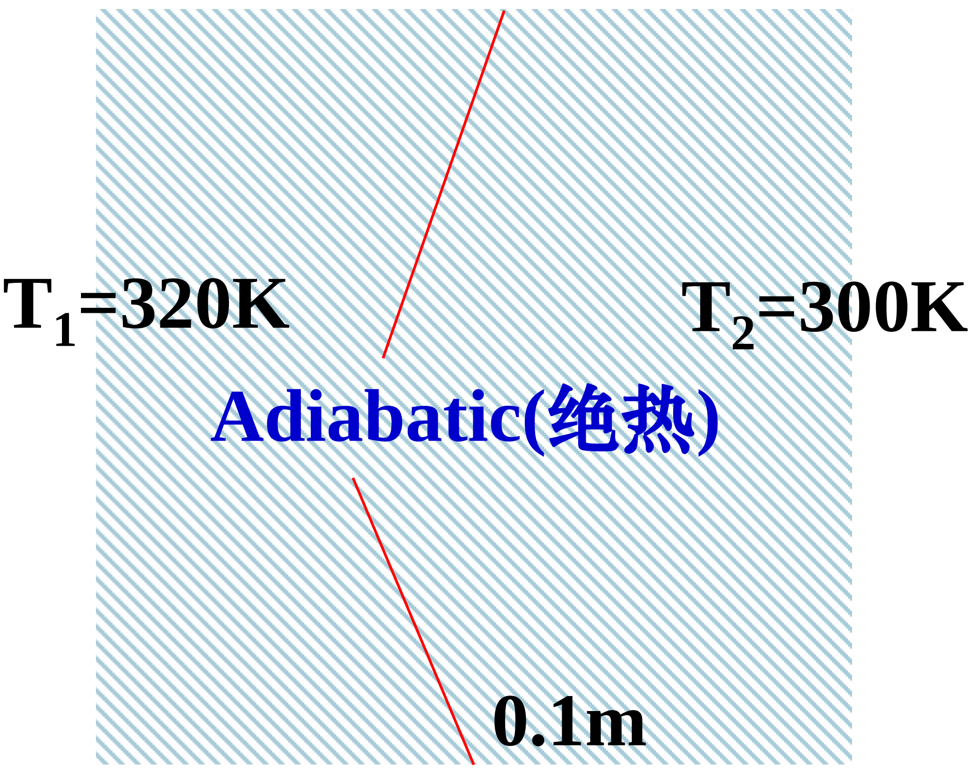


Fig.4 Other boiling results from FLUENT

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

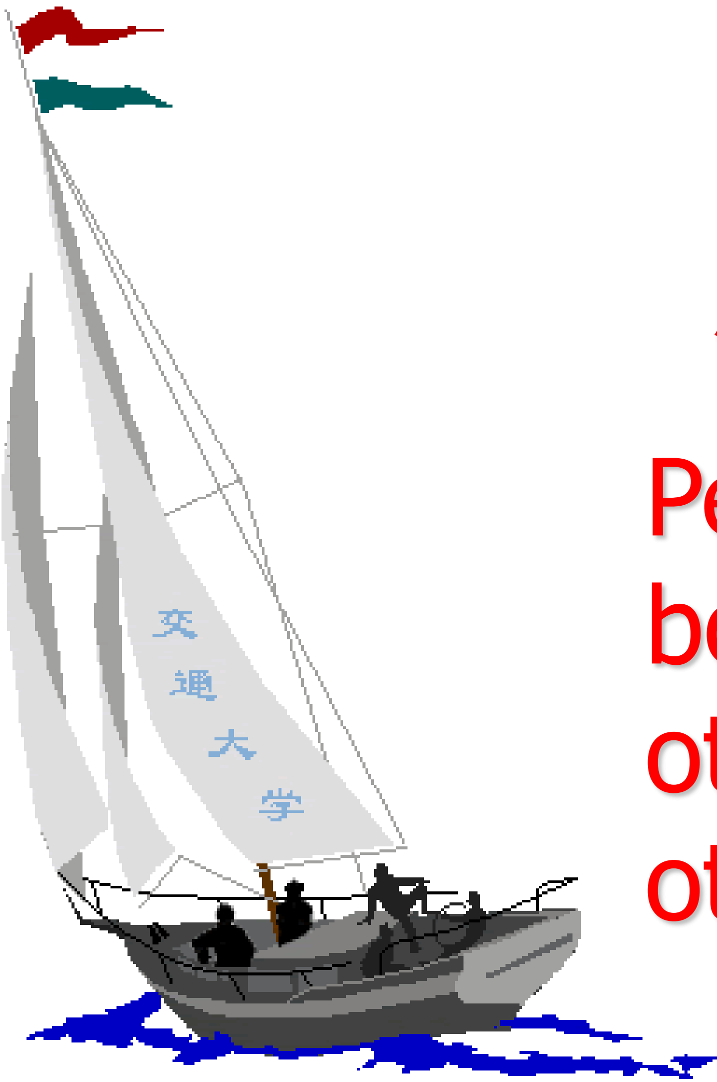
Known：Natural convection of air in a square filled with porous media. The left wall is with high temperature, while the right wall is with low temperature. The bottom and top walls are adiabatic.



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.

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….