



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, 2019-12.-23





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 2019年12月23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>u=0.1m/s</u> enters the channel from a micropore at the bottom.

Fig.1 Computational domain and geometry sizes (μm) 3/76

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, 140°
Up	Wall, 60°

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 H₂O Multiple components (多组分多相流)

Crown

Liquid water and air system, H₂O, N₂, O₂...

6

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)

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

$$P_{\mathbf{C}} = P_1 - P_2 = \frac{\sigma \cos\theta}{r}$$

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.

2. Different methods for multiphase flow

Macroscopic

 Volume of Fluid (VOF) 流体体积法
 VOSET

 Level Set (LS) 水平集法
 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 finitedifference 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 ... ☆ 99 被引用次数: 12539 相关文章 所有 17 个版本

Several methods have been previously used to approximate free boundaries in finitedifference 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

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 12/76

Schematic of 2D two-phase flow system

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 termConvection termConvection-diffusion type equationThe two phases are not soluble (互溶) , so there is nodiffusion term. When there is chemical reaction orphase change, source term is not zero.14/76

 $\mathbf{F} = 2\sigma k \frac{\rho \mathbf{v} \mathbf{C}_1}{(\rho_1 + \rho_g)}$

CSF model

The governing equations for multiphase phase flow using VOF

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

Surface tension force

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

u from NS equation

$$\frac{\partial \boldsymbol{C}_m}{\partial t} + \mathbf{u} \cdot \nabla \boldsymbol{C}_m = 0$$

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

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

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

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

D-NHT-EHT

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

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

interface mean curvature

$$k = \frac{1}{r} = \nabla \cdot \left(\frac{\nabla \tilde{C}_1}{|\nabla \tilde{C}_1|}\right)$$

pressure in the transition region is

$$P_{x} = P_{g} + \sigma k(C_{x} - C_{g}) = P_{g} + \sigma kC_{x}$$
$$\mathbf{F} \sim \nabla(P_{x} - P_{g}) = \nabla(\sigma k(C_{x} - C_{g}))$$
$$= \sigma k \nabla C$$

Suppose local k is constant.

$$\begin{array}{c} \mathbf{C} \\ 1 \\ 0.75 \\ 0.5 \\ 0.5 \\ 0 \end{array} \begin{array}{c} p_{\mathbf{a}} \\ p_{\mathbf{g}} \\ \mathbf{b} \end{array} \begin{array}{c} p_{\mathbf{b}} \\ P_{\mathbf{b}}$$

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

How to solve the VOF equation?

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?

CFD-NHT-EHT

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, 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 - Interface is reconstructed! Volume of fraction

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

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.

i-1 *i i*+1 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

is satisfied, or the Courant number <1.

The governing equations for multiphase phase flow using VOF ∇C_1

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

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

CSF model

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

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

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

Two-way coupled with each other.

2 Process of simulation

2.1 Launch ANSYS Fluent

💶 Fluent Laur	ncher			_		×
ANS	YS		F	luen	t Laun	cher
Dimension			Options 🗹 Double Precisi	on		
 3D Display Options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme 			Processing Options Serial Parallel 	\$		
Show Mor	e Options <u>DK</u>	fault	Cancel	<u>H</u> e	elp 🔻	

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

2.2 Read the mesh

$File \rightarrow Read {\rightarrow} Mesh$

	File	Mesh	Define	Solve	Adapt	Surface	Display	Report	Para	alle
		Read			>	Mesh		-		•
		Write			>	Case		F		
		Import			>	Data		-		-
		Export			>	Case &	Data		_	
		Export	to CFD-P	ost		PDF		ua	ity	
		Solution	n Files			ISAT Ta	able			
		Interpo	late			DTRM	Rays			
		FSI Ma	pping		>	View Fa	actors			
		Save Pi	cture			Profile.				
		Data Fi	le Quantit	ies		Scheme	e			
		Batch C	Options			Journal				
		Exit				3d1-1				
1	So	onitors Jution Initi	ialization		,	model1	l -	Jni	ts	
	Ca	alculation /	Activities			3d1				-
	Ru Deci	un Calculat ulte	tion	Help	0	18.10.2	5			
	Gr	aphics and	d Animation	s	_					

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

$General {\rightarrow} Mesh {\rightarrow} Check$

1: Mesh	~	
		ANSYS
		R14.5

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

General	Scale Mesh	×
Mesh Scale Check Report Quality Display	Domain Extents Xmin (m) -0.00033 Xmax (m) 0.00117 Ymin (m) 0 Ymax (m) 0.0003	Scaling Convert Units Specify Scaling Factors Mesh Was Created In Select>
Solver Type Velocity Formulation Pressure-Based Absolute Density-Based Relative Time Steady	View Length Unit In m ~	Scaling Factors X 0.000001 Y 0.000001 Scale Unscale
● Transient	Close Help	

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

CFD-NHT-EHT

2.5 Choose the solver

General→Solver

Solver		
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative]
Time Steady Transient		
✓ Gravity		Units
Gravitational Accelerat	ion	
X (m/s2) 0		
Y (m/s2) -9.8	·	
Z (m/s2)	P	
<u>.</u>		

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.

CFD-NHT-EHT CENTER

Models

Models

2.6 Choose the models

Solution Setup→Models

		bold don be dap	Muruphase - Volume of Flu	liu
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Models Models Multiphase - Volume of Fluid Energy - Off Viscous - Laminar Radiation - Off Viscous Model Inviscid Laminar Spalart-Allmaras (1 eqn) k-epsilon (2 eqn) k-omega (2 eqn)	General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Activities Run Calculation Results Graphics and Animations	Model Off Off Off Off Off Off Off Off Evelerian Wet Steam Coupled Level Set + VOF Level Set Volume Fractor For anceters Scheme Explicit Implicit	Options Options Open Channel Flow Open Channel Wave BC Zonal Discretization
	O Transition K-Ri-omega (3 eqn) O Transition SST (4 eqn) O Reynolds Stress (7 eqn) O Scale-Adaptive Simulation (SAS) O Detached Eddy Simulation (DES) O Large Eddy Simulation (LES)	Plots Reports	Volume Fraction Cutoff 1e-06 Courant Number 0.25 Default Body Force Formulation Implicit Body Force OK	Cancel Help

Meshina

Mesh Generation

Colution Cotur

Choose Volume of Fluid as Multiphase Model

Coupled Level set +VOF?

1: Mes

 \times

Coupled Level Set +VOF

Spatial gradient (interface curvature and surface tension force)

Combining the advantages of LS and VOF




2.6 Choose the models

Body force formulation

Multiphase - Volume of Elui	id
Epergy - Off	
🔁 Multiphase Model	×
Model Off Volume of Fluid Mixture Eulerian Wet Steam	Number of Eulerian Phases
Coupled Level Set + VOF	
Level Set	
Values Frankiss Basershare	O-Fara
volume Fraction Parameters	Options
Scheme	Open Channel Flow
Explicit	Open Channel Wave BC
Volume Fraction Cutoff 1e-06	
Courant Number	
0.25	
	5
Default	
Body Force Formulation	-
ОК	Cancel Help

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

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 Momentum equation

Multiphase flow is controlled by a set of forces.





CFD-NHT-EHT

2.7 Define the materials

Solution Setup→ Materials→Create/Edit Material Create water-liquid

Mashina	Materials	Lreate/Edit Materials		
Mesh Generation	Material	Name	Matazial Tura	Order Materials by
Solution Setup	Matenais Fluid	water-liquid	fluid	Name
General	water-liquid	Chemical Formula		Chemical Formula
Maddle	Solid	h2o<>	Fluent Fluid Materials	Fluent Database
Phases	aluminum	L	water-liquid (n2o<1>)	
Cell Zone Conditions			Mixture	v
Boundary Conditions			i non re	
Dynamic Mesh		🛃 Fluent Database Materials	×	
Reference Values		Fluent Fluid Materials	Material Type	
Solution		vinyl-silylidene (h2cchsih)	fluid V	
Solution Methods		vinyl-trichlorosilane (sicl3ch2ch)	Order Materials by	
Monitors		water-liquid (h2o <l>)</l>	Name	
Solution Initialization		wood-volatiles (wood, vol)	Chemical Formula	
Calculation Activities Run Calculation		1		
Results		`	,	
Graphics and Animations		Copy Materials from Case Delete		
Plots		Properties		
Reports	Create/Edit Delete	Density (ka/m3)	^	
•			constant v View	
			998.2	
	нер	Cp (Specific Heat) (j/kg-k)	1.1.	
			constant View	
			4182	
		Thermal Conductivity (w/m-k)	View	
		Ī	verv	
			0.6	
		Viscosity (kg/m-s)	constant View	
			0.001002	
			0.001003	
		,	· ·	
		New Edit	Save Copy Close Help	
	l	coTe		1

- 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

	Dhanan	Meshing	Phases	1; Mesn
Meshing Mesh Generation	Phases	Mesh Generation	Chases	
Solution Setup General	phase-1 - Primary Phase phase-2 - Secondary Phase	Solution Setup General	air - Primary Phase phrase-2 - Secondary Phase	
Models Materials	Primary Phase	Models Materials	Secondary Phase	\times
Phases Cell Zone Conditions	Name	 Cell Zone Conditions Boundary Conditions 	Name]
Mesh Interfaces Dynamic Mesh	Phase Materia air V Edit	Mesh Interfaces Dynamic Mesh Reference Values	Phase Material water-liquid ~ Edit	
Reference Values Solution	OK Cancel Help	Solution Solution Methods	OK Cancel Help	
Solution Methods Solution Controls Monitors		Solution Controls Monitors		
Solution Initialization Calculation Activities		Calculation Activities Run Calculation	Edit Interaction ID 3	

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





2.8 Define the phases

Define surface tension force

Meshing	Phases	1: Mesh v
Mesh Generation	Phases	
Solution Setup General Models Materials Phases Cell Zone Conditions	air - Primary Phase water - Secondary Phase	
Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values		
Solution		
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Puro Calculation		
Results	Edit	
Phase Interaction		
Drag Lift Wall Lub Surface Tension Force Model O Continuum Surface F	Adhesion Options	lisions Slip Heat Mass Reaction Surface Tension Decretization I
Surface Tension Coefficien	nts (n/m)	
water	air constant	Edit
		v
	OK	Cancel Help

- 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.



CFD-NHT-EHT

2.9 Define cell zone conditions

Solution Setup→ **Cell Zone Condition** →**Operating Conditions**

Meshing	Cell Zone Conditions
Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Boundary Conditions Dynamic Mesh Reference Values Solution Solution Methods Solution Controls Monitors Solution Initialization	Cell Zone Conditions Zone Fluid
Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Phase Type ID mixture fluid 9 Edit Copy Profiles Parameters Operating Conditions
Coperating Condition Pressure Operating Press 101325 Reference Pressure Locat X (m) -0.00033 Y (m) 0.00015 Z (m) 0	ns X Gravity Gravity Gravity Gravitational Acceleration X (m/s2) 0 P Y (m/s2) -9.8 Z (m/s2) 0 P Variable-Density Parameters Specified Operating Density Operating Density (kg/m3) 1.225 P
	OK Cancel Help

Operating pressure In Fluent, operating pressure is the same as reference pressure. Input location and

value of operating

pressure.



CFD-NHT-EHT

2.9 Define cell zone conditions

Solution Setup→ **Cell Zone Condition** →**Operating Conditions**

	Cell Zone	Conditions	
Mesh Generation	Zone		
Solution Setup	fluid		
General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values			
Solution			
Solution Methods Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation	Phase	Type ID	
Results	mixture	✓ fluid ✓ 9	
Graphics and Animations			
Plots	Edit	Copy Profiles	
Reports	Parameters.	Operating Conditions	
Operating Conditio	ns		\times
Derating Conditio	ns	Gravity	×
Operating Conditio Pressure Operating Press 101325 Reference Pressure Locat X (m) -0.00033 Y (m) 0.00015 C ()	ns ure (pascal) P tion F	Gravity Gravity Gravitational Acceleration X (m/s2) 0 Y (m/s2) -9.8 Z (m/s2) 0 I	
Operating Conditio Pressure Operating Press 101325 Reference Pressure Locat X (m) -0.00033 Y (m) 0.00015 Z (m) 0	ns sure (pascal) tion F	Gravity Gravity Gravitational Acceleration X (m/s2) 0 Y (m/s2) -9.8 Z (m/s2) 0 Variable-Density Parameters Specified Operating Density Operating Density (kg/m3) 1.225	

Specified operating density

Set the operating density to be the

density of the lightest phase.

Here input the density of the air.

Variable-Density Parameters	
Specified Operating Density	
Operating Density (kg/m3)	
1.225 P	
ncel Help	
	43/7



Ъ	CFD-NHT-EHT
	CENTER

2.10 Define the boundary conditions

Solution Setup → **Boundary Condition**

Phase Type mixture velocity-inlet velocity-inlet	ID 12
Velocity Inlet	
Zone Name air-in	Phase mixture
Momentum Thermal Radiation Species DPM Multiphase	UDS
Velocity Specification Method Magnitude, Normal to	Boundary
Reference Frame Absolute	
Velocity Magnitude (m/s)	constant
Supersonic/Initial Gauge Pressure (pascal)	constant
OK Cancel Help	

Phase	Type	ID
water 👻	velocity-inlet •	12

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.

Velocity Inlet	×
Zone Name air-in	Phase water
Momentum Thermal Radiation Species DPM Multiphase Volume Fraction 0 constant	UDS
OK Cancel Help	



ЪЪ.	CFD-NHT-EHT
	CENTER

2.10 Define the boundary conditions

For the top and bottom surface, Define contact angle

ne Name		Phase		
dl		mixture		
jacent Cell Zone				
uid				
10mentum Thermal Radi	ation Species DPM Multip	phase UDS Wall Film	l.	
Vall Motion Motio	n			
Stationary Wall Moving Wall	Relative to Adjacent Cell Zone			
haar Condition				
ilear condition	7			
No Clip				
 No Slip Specified Shear 				
 No Slip Specified Shear Specularity Coefficient 				
 No Slip Specified Shear Specularity Coefficient Marangoni Stress 				
 No Slip Specified Shear Specularity Coefficient Marangoni Stress Vall Roughness 			1	
No Slip Specified Shear Specularity Coefficient Marangoni Stress Wall Roughness Roughness Height (m)	constar	ıt v		
No Slip Specified Shear Specularity Coefficient Marangoni Stress Vall Roughness Roughness Height (m)		it v]		
No Slip Specified Shear Specularity Coefficient Marangoni Stress Vall Roughness Roughness Height (m) Roughness Constant 0.5	constan	it v		
No Slip Specified Shear Specified Shear Marangoni Stress Wall Roughness Roughness Height (m) Roughness Constant 0.5 Wall Adhesion	constan	it v]		
No Slip Specified Shear Specified	constar	nt v] nt v]		
No Slip Specified Shear Specularity Coefficient Marangoni Stress Wall Roughness Roughness Height (m) Roughness Constant O.5 Wall Adhesion Contact Angles (deg)	constar	nt v) nt v)		

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

Wall	Ad	hesion
A M POINT	1004	I INGGINALI

Contact Angles (deg)			
water	air	140	constant 👻



46/76

2.10 Define the boundary conditions

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

Phase Type I	Velocity Inlet	
mixture 🔻 velocity-inlet 👻	Zone Name	Phase
Edit Copy Profiles	water	mixture
Parameters Operating Conditions	Momentum Thermal Radiation Species DPM Multiphase	UDS
Display Mesn	Velocity Specification Method Magnitude, Normal to B	oundary 👻
Help	Reference Frame Absolute	•
	Velocity Magnitude (m/s) 0.1	constant 👻
	Supersonic/Initial Gauge Pressure (pascal)	constant 💌
	OK Cancel Help	
Phase Type	I 💽 Velocity Inlet	— X —
water velocity-inle	t Zone Name	Phase
Edit Copy Pr	ofiles water	water
Parameters Operating Condition Display Mesh Periodic Condition	itions Momentum Thermal Radiation Species DPM ons Volume Fraction 1 constant	Multiphase UDS
Help	OK Cancel He	þ





2.11 Choose the solution methods

Solution \rightarrow Solution Methods

Meshing

Mesh Generation

Solution Setup

General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values

Solution

Solution Methods

Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation

Results

Graphics and Animations Plots Reports

Scheme		
PISO		
Skewness Correction		
1		
Neighbor Correction		
1		
C Skawpage Naighb	er Ceueline	
[♥] 5kewness-weight	or Coupling	
Spatial Discretization		
Gradient		
Green-Gauss Cell Ba	ased	
Pressure		
Body Force Weighte	ed .	
Momentum		
Second Order Upwi	nd	
Volume Fraction		
Geo-Reconstruct		
ransient Formulation		
First Order Implicit		~
Non-Iterative Time	Advancemen	it
Frozen Flux Formul	ation	
_ High Order Term Re	axation C	ptions

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.



ЪЪ.	CFD-NHT-EHT
	CENTER

2.11 Choose the solution methods

Solution \rightarrow Solution Methods

Meshing

Mesh Generation

Solution Setup

General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values

Solution

Solution Methods

Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation

Results

Graphics and Animations Plots Reports

	ty coup	/iiiing		
Scheme				
PISO				`
Skewness Cor	rection			
1				
Neighbor Corr	ection			
1				
	Neiabb	or Couplin	n	
		or coupin	9	
patial Discretiz	ation			
Gradient				
Green-Gauss	s Cell Ba	ased		
Pressure				
Body Force \	Neighte	ed		
Momentum				
Second Orde	er Upwir	nd		
Volume Fracti	on			
Geo-Recons	truct			
Transient Form	ulation			
First Order Imp	olicit			\sim
Non-Iterativ	e Time	Advancer	nent	
Frozen Flux	Formula	ation		
Libela Onder 7	i orm u c	lavation	Onlines	

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)
- 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

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





2.11 Choose the solution methods

Solution \rightarrow Solution Methods

Meshing

Mesh Generation

Solution Setup

General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values

Solution

Solution Methods

Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation

Results

Graphics and Animations Plots Reports

Scheme				
PISO				
Skewness Corr	ection			
1				
Neighbor Corre	ection			
1				
		C I		
Skewness-	leighbo	r Couplin	g	
Spatial Discretiza	ation			
Gradient				
Green-Gauss	Cell Bas	sed		
Pressure				
Body Force W	/eighteo	ł		
Momentum				
Second Order	Upwin	d		
Volume Fractio	'n			
Geo-Reconstr	uct			
ransient Formu	ation			
First Order Imp	icit			\sim
Non-Iterative	Time A	dvancer	nent	
Frozen Flux F	Formula	tion		
_ High Order 1	erm Rei	axation	Options	

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): <u>recommended</u>.
- ✓ For porous media: not recommended!
- **5. PRESTO!** (Pressure Staggering Option) scheme For problem with high pressure gradient.



ЪЪ.	CFD-NHT-EHT
	CENTER

2.11 Choose the solution methods

Solution \rightarrow Solution Methods

Meshing

Mesh Generation

Solution Setup

General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values

Solution

Solution Methods

Solution Controls Monitors Solution Initialization Calculation Activities Run Calculation

Results

Graphics and Animations Plots Reports

Scheme			
PISO			`
Skewness Correct	tion		
1			
Neighbor Correcti	ion		
1			
Skewness-Nei	abbor Couplin	0	
Skewness-weig	gribbi Coupiin	iy	
patial Discretizatio	n		
Gradient			
Green-Gauss Ce	ll Based		
Pressure			
Body Force Weig	ghted		
Momentum			
Second Order U	pwind		
Volume Fraction			
Geo-Reconstruc	t		
ransient Formulati	ion		
First Order Implicit	t		\sim
Non-Iterative Ti	ime Advancen	nent	
Frozen Flux For	mulation		
High Order Tern	n Relaxation	Options	

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 <u>most accurate</u> and is <u>applicable for</u> general unstructured meshes.

Modified HRIC, Compressive, and CICSAM schemes <u>are less computationally expensive</u> than the Geo-Reconstruct scheme, the interface between phases will <u>not be as sharp</u> as the geometric reconstruction interpolation scheme. 54/76





2.12 Define the monitors

$\textbf{Solution} \rightarrow \textbf{Monitors}$

Define the Residuals Monitor and write 0.0001 in

the Absolute Criteria box

Meshing	Monitors		1: Mesh	~		
Mesh Generation	Residuals, Statistic and Force Monitors					
Solution Setup	Residuals - Print, Plot					
General Models	Residual Monitors					×
Materials	Options	Equations				
Phases	Print to Console	Residual	Monitor	Check Convergen	e Absolute Criteria	A
Cell Zone Conditions Boundary Conditions	Plot	continuity		\checkmark	0.0001	
Mesh Interfaces Dynamic Mesh	Window	x-velocity			0.0001	
Reference Values	Curves Axes	v-velocity			0.0001	
Solution	Iterations to Plot	[[,] [,] [,] [,] [,]		Č.		\sim .
Solution Methods	1000	Residual Va	lues		Convergence Cr	iterion
Solution Controls Monitors		Norma	lize	Iterations	absolute	~
Calculation Activities	Iterations to Store			-		
Run Calculation	1000	Scale 🗹				
Results		Compu	ute Local Scale			
Graphics and Animations						
Plots	OK Plot	Rer	ormalize (Cancel H	lelp	
Reports						



CFD-NHT-EHT

2.12 Define the monitors

Create the Surface Monitor

Meshing	Monitors		1: Mesh	Meshina	Monitors		1: Mesh
Mesh Generation	Residuals, Statistic a	nd Force Monitors		Mesh Generation	Desiduale Challen	and Farme Manitana	-
Solution Setup	Residuals - Print, Pl	ot		Solution Setup	Residuals, Stausuc	and Force Monitors	
General	Statistic - Off			General	Statistic - Off		
Models				Models			
Materials				Materials			
Cell Zone Conditions				Phases Cell Zone Conditions			
Boundary Conditions	Create 🔻 Edit	Delete		Boundary Conditions	Create 👻 Edit	Delete	
Mesh Interfaces	Surface Monitors			Mesh Interfaces	Surface Monitors		
Reference Values				Reference Values	pressure-in - Area	-Weighted Average, Static Pressu	1
Solution				Solution			
Solution Methods				Solution Methods			
Solution Controls				Solution Controls			
Solution Initialization	Create Edit	Delete		Solution Initialization			
Calculation Activities	Create	Delete		Calculation Activities	Create Edi	. Delete	
Surface Monitor			×	Surface Monitor			×
Name		Report Type				Descent Trees	
pressure-in		Area-Weighted Average	\sim	pressure-out		Area-Weighted Average	~
		Field Variable				Field Variable	
Print to Console		Pressure	~			Pressure	~
Plot		Static Pressure	\sim	Print to Console		Static Pressure	~
Window		Surfaces		Window		Surfaces	
2 Curves	. Axes	air-in air-out		3	A.v.o.a	air-in	
Write		gdl		Curves.	Алев	air-out	
File Name		int_fluid		Write		nt_fluid	
surf-mon-1.out		water		File Name		wall	
V Avia				Sur Hior-2.out		and the second sec	
Iteration	\sim			X Axis			
Get Data Every				Iteration	~		
1 Iteration	~			Get Data Every			
				1 Iteration	~		
		New Surface 🕶		-		New Surface -	
	ОК	Cancel Help			ОК	Cancel Help	

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





2.13 Initialize

Solution \rightarrow Solution Initialization

Meshing	Sole
Mesh Generation	Initia
Solution Setup	0
General	Ĭ
Models Materials	Com
Phases	air-i
Cell Zone Conditions Boundary Conditions	Refe
Mesh Interfaces Dynamic Mesh	l
Solution	Initia
Solution Methods Solution Controls Monitors	Ga
Solution Initialization Calculation Activities Run Calculation	5
Results	YV
Graphics and Animations Plots	
Reports	wa 0

-	Solution Initialization				
	Initialization Methods O Hybrid Initialization Standard Initialization				
L	Compute from				
	air-in	~			
	Reference Frame				
	Relative to Cell Zone Absolute				
	Initial Values				
	Gauge Pressure (pascal)	1			
	0				
	X Velocity (m/s)				
	5				
	Y Velocity (m/s)				
	0				
	water Volume Fraction				
	0				

- Choose Standard Initialization
 Choose air-in
 - Click Initialize



2.14 Run calculation

Solution \rightarrow Run Calculation

Meshing	Run Calculation
Mesh Generation Solution Setup General Models Materials	Check Case Preview Mesh Motion Time Stepping Method Time Step Size (s) Fixed 1e-07 Settings Number of Time Steps
Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values	Options
olution Solution Methods Solution Controls Monitors Solution Initialization	Sampling Interval Image: Sampling Options Time Sampled (s)
Results	Max Iterations/Time Step Reporting Interval 20 1
Graphics and Animations Plots Reports	Profile Update Interval 1 Data File Quantities Acoustic Signals
	Calculate

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 ∆t. 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=1.0529e-2

T=1.0729e-2

Fig.2 water behavior











Pressure drop

Saturation







66/76

Computer-aided project (3) of NHT-2019, XJTU

Known: There are two kinds of solid particles A and B. The diameter of A and B are different. A and/or B will be packed into a microchannel to form porous zone to enhance heat transfer. Half of the microchannel will be occupied by the porous zone.







Parameter	Н	L	$oldsymbol{ ho}_{ ext{water}}$	$\eta_{ m water}$	C _{p water}	$\lambda_{ ext{water}}$	Е
Value (Unit)	0.2 (cm)	2 (cm)	1000.0 (kg·m ⁻³)	998×10 ⁻⁶ (kg·m ⁻¹ ·s ⁻¹)	4182 (J·kg ⁻¹ ·K ⁻¹)	0.59 (W·m ⁻¹ ·K ⁻¹)	0.5
Parameter	d _A	d _B	Re				
Value (Unit)	100 (μm)	50 (μm)	0.1-50				

Assumptions:

constant physical properties, steady, laminar flow

	Boundary	Condition		
	x=0	<i>v</i> _x = <i>U</i> _{in} , T=300 K		
Roundary	x=L	<i>p</i> =0 Pa , T _{backflow} =300 K		
conditions	y=H	$v_{\rm x} = v_{\rm y} = 0$ $q = 500 {\rm Wm^{-2}}$		
	Porous region	Thermal- equilibrium model		



CFD-NHT-EHT

Possible position of porous zone





CFD-NHT-EHT

Possible position of porous zone



More patterns should be considered...





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, and try to find the desirable structures of porous media to obtain high *PN*. Analyze the *Nu*, pressure drop (ΔP) and PN at different *Re* and permeability. Thermal-equilibrium model is adopted for porous medium region. KC equation is adopted to calculate the permeability.

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





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....







Group photo at the front of the main building