



# **Numerical Heat Transfer**

# Chapter 13 Application examples of Fluent for flow and heat transfer problem



# Instructor Li Chen, Wen-Quan Tao

CFD-NHT-EHT Center Key Laboratory of Thermo-Fluid Science & Engineering Xi'an Jiaotong University Xi'an, 2020-12-15







# 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 2020年12月15日,西安



CFD-NHT-EHT

# **Class intermediate**

13. A1 Single phase fluid flow and heat transfer in manifold microchannel (歧管微通道中流动换热)

13. A2 Flow and heat transfer in porous media (多孔介质流动换热)

13.A3 Multiphase flow using Volume of Fraction method (多相流VOF方法模拟)



# For each example, the general content of the lecture is as follows:

# 1: Using slides to explain in detail the general 10 steps for Fluent simulation! (PPT讲解)

- 1. Read mesh
- 3. Choose model
- **5. define zone condition**
- 7. Solution
- 9. Run the simulation

- 2. scale domain
- **4.define material**
- **6. define boundary condition**
- 8. Initialization
- **10. Post-processing**

2: Operating the Fluent software to simulate the example and post-process the results. (运行软件)





### **13. A1**

# Flow and heat transfer in manifold

# (**岐管**) microchannel

# (歧管微通道中流动换热)

# 1. What is microchannel?

2. What is manifold?





# What is "Microscale" ?

# 1. The continuum assumption (连续介质假设) does not stand.



# **Boltzmann equation**



# What is "Microscale" ?

2. The continuum assumption still stands, but the relative importance of affecting factors changes.

Fluid flow is controlled by different forces such as viscous force, gravitational force, surface tension force...

These force can be classified into two kinds: body force and surface tension force.

body forces: ~m<sup>3</sup> surface forces: ~m<sup>2</sup>

surface forces/body forces: ~m<sup>-1</sup>; surface force becomes stronger as length scale decreases.



# Multiphase heat transfer in microchannel



- 1. Body force such as gravity force can be neglected.
- 2. Pressure and surface tension force are dominant (主导).



Because of the integration (集成化) of electron compon ent (电子元件), the heat flux of a EC greatly increases, ev en reaches MW·m<sup>-2</sup> order of magnitude.

西安交通大學

Traditional cooling techniques cannot meet the cooling demand of such high heat flux.

Microchannel is promising technique for cooling.









Huawei Technologies Co., Ltd., It designs, develops, and sells telecommunications equipment and consumer electronics. 10/58



CFD-NHT-EHT

There are three most important key laboratories in Huawei, including Advance structural material Lab, Advance thermal technique lab and Noah's Ark Lab (诺亚方舟实验室, for AI) 。



Mate 20



 $W = L = 1 \text{ cm}, w_w = w_c = 57 \ \mu\text{m}, \ z = 365 \ \mu\text{m}$ 

**Proposed by Tuckerman and Pease in 1981 from Stanford Electronics Laboratories.** 

12/58





# **Traditional microchannel**



**Proposed by Tuckerman and Pease in 1981** 



# **Traditional microchannel**



 $W = L = 1 \text{ cm}, w_w = w_c = 57 \mu \text{m}, z = 365 \mu \text{m}$ 

- 1. From the inlet to the outlet, temperature increases.
- 2. Pressure drop is high.

西安交通大學

**3.** Inlet effect is not significant.



# What is "manifold" ?

西安交通大學



A kind of structure that can distribute fluid.



16/58

# Manifold microchannel



1. Inlet effect is strong.

西安交通大學

- 2. Pressure drop decreases.
- 3. Temperature distribution is more uniform.



CFD-NHT-EHT

# Traditional microchannel

# **Manifold microchannel**



- 1. Better cooling performance.
- 2. Lower pressure drop.
- 3. More uniform temperature distribution



CFD-NHT-EHT

#### International Journal of Heat and Mass Transfer 157 (2020) 119982



Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/hmt





#### Ming Peng, Li Chen\*, Wentao Ji, Wenquan Tao

Key Laboratory of Thermo-Fluid Science and Engineering of MOE, School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an, Shaanxi 710049, China

#### ARTICLE INFO

Article history: Received 4 March 2020 Revised 18 May 2020 Accepted 20 May 2020 Available online 18 June 2020

Keywords: Multi-jet microchannel Heat transfer Cooling performance Thermal resistance Pressure drop Numerical simulation

### June 18th

#### ABSTRACT

A multi-jet microchannel (MJMC) heat sink with coolant flowing through alternative inlet and outlet jets in the direction normal to the heated surface is studied. Three dimensional flow and heat transfer processes in the MJMC are numerically simulated using the SIMPLE-type finite volume method (FVM). Compared with traditional microchannels, the MJMC combines the advantages of impinging jet flow and entrance effects of microchannels, and thus its cooling performance overwhelms showing less pumping power, lower thermal resistance and improved uniformity of temperature at the bottom surface. Effects of various geometrical parameters including jet numbers, channel aspect ratio, the fin width to channel width ratio and the width of the outlet on the performance of the MJMC are analyzed in detail. It is found that the MJMC with more jets, wider outlet and smaller fin width to channel width ratio offers better cooling performance. While the cooling performance exhibits a non-monotonic trend with the channel aspect ratio, the MJMC heat sink with 7 jets, aspect ratio of 6 and fin width to channel width ratio of 0.5 obtains the best cooling performance.





# Nature paper

# nature

Explore our content V Journal information V

nature > articles > article

**09 Sep.** 

Article Published: 09 September 2020

# Co-designing electronics with microfluidics for more sustainable cooling

Remco van Erp, Reza Soleimanzadeh, Luca Nela, Georgios Kampitsis & Elison Matioli 🖂

Nature 585, 211–216(2020) Cite this article

14k Accesses | 342 Altmetric | Metrics











# Example 13 A1. Known

Steady single phase fluid flow and heat transfer of water in a manifold microchannel, as shown in Fig. 1



#### Fig. 1 Schematic of the manifold microchannel channel



Inlet	Velocity inlet; 293.15K
Outlet	Pressure out: 1atm
Bottom	Heat flux(1×10 <sup>6</sup> W·m <sup>-2</sup> )
Up	Adiabatic wall
Side	Symmetry & adiabatic



Find: Temperature of bottom surface  $(T_b)$ , pressure drop  $(\Delta P)$  and heat transfer coefficient (h) under different Reynolds number (44, 88, 132, 176 and 220).

# Assumptions:

- (1) When *Kn* is less than 10<sup>-3</sup>, N-S Eqs still can be used;
- (2) Laminar, incompressible, Newtonian fluid;
- (3) Physical parameters are constant;
- (4) The gravity and viscous dissipation can be ignored;
- (5) The thermal radiation can be ignored.



Remark: develop reasonable physical model and write down the right governing equation, BC and IC is the first and most important step before using software Fluent. Fluent is just a tool for solving above problem ! Background of NHT helps you better use the tool.

# Governing equations:

Continuum equation  $\nabla u = 0$ 

**Momentum equation**  $\nabla(\rho uu) = -\nabla p + \eta \nabla^2 u$ 

**Energy equation**  $\nabla(\rho c_p u T_f) = \nabla \lambda_f \nabla T_f \quad 0 = \nabla \lambda_s \nabla T_s$ 

西安交通大学

# **Start the Fluent software**

C Fluent Launcher	
<b>ANSYS</b>	Fluent Launcher
Dimension 2D 3D	Options Double Precision  Meshing Mode
Display Options           Image: Display Mesh After Reading           Image: Display Mesh After Reading <th>Processing Options Serial Parallel</th>	Processing Options Serial Parallel
● Show More Options	
<u>D</u> K <u>D</u> efault	

- 1. Choose 3-Dimension
- 2. Choose display options
- **3.** Choose Serial processing

option or parallel to choose

different number of processes

#### **Note: Double precision or Single precision**

Sometimes the single precision version of Fluent is sufficient. For example, for heat transfer problem, if the thermal conductivity between different components are high, it is recommended to use Double Precision Version. 25/68





## **Step 1: Read and check the mesh**

- The mesh is generated by pre-processing software such as ICEM and GAMBIT. The document is with suffix (后缀名) ".msh"
- This step is similar to the Grid subroutine (UGRID, Setup1) in our general teaching code.

2	Fluent@DESKTOP-20	C2BC	DSO [2d,	pbns, lam]		
File	Mesh Define So	lve	Adapt	Surface Display	File $\rightarrow$ R	ead→Mesh
	Read		$\rightarrow$ (	Mesh	<b>T</b>	Building
	Write		>	Case		mesh Note: Separating wall gove 15 into gover 15 and 2
	Import Export Export to CFD-Post		>	Data Case & Data PDF ISAT Table	uality	Note: Separating wall zone 16 into zones 16 and 3. symm -> symm (15) and symm:002 (2) Slitting wall zone 18 into a coupled wall. materials, interface, domains,
	Interpolate FSI Mapping		>	DTRM Rays View Factors		zones, coupled-shadow wall:003 symm:002 out
	Save Picture Data File Quantities Batch Options			Protile Scheme Journal		in coupled btm wall symm int_solid
	Exit			wall		int_fluid solid fluid
So Ca Ru	lution Initialization Iculation Activities In Calculation	Gra	avity		Units	Done. <b>26/68</b>



# **Step 1: Read and check the mesh**

# Mesh→Check

西安交通大學

Check the quality and topological information of the mesh



# Sometimes the check will be failed if the quality is not good or there is a problem with the mesh.

Face area statistics: WARNING: invalid or face with too small area exists. minimum face area (m2): 0.000000e+00 maximum face area (m2): 5.081937e-03

WARNING: Mesh check failed.

WARNING: The mesh contains high aspect ratio quadrilateral, hexahedral, or polyhedral cells.





# Step 2: Scale the domain size (缩放)

## General→Scale

# Make sure the unit is right.

General				
Mesh		Scale Mesh		×
Scale	Check Report Quality	( Domain Extents		Scaling
Display		(Xmin (mm) -1.11022e-16	Xmax (mm) 5	• Convert Units
		Ymin (mm) -1.38778e-17	Ymax (mm) 0.6	Specify Scaling Factors
Solver		• Zmin (mm) -6.93889e-18	Zmax (mm) 0.15	Mesh Was Created In
Type <ul> <li>Pressure-Based</li> <li>Density-Based</li> </ul> Time <ul> <li>Steady</li> <li>Transient</li> </ul>	Velocity Formulation <ul> <li>Absolute</li> <li>Relative</li> </ul> 2D Space <ul> <li>Planar</li> <li>Axisymmetric</li> <li>Axisymmetric Swirl</li> </ul>	View Length Unit In		Scaling ractors           X         0.001           Y         0.001           Z         0.001           Scale         Unscale
Gravity	Units		Close Help	
Help				

You can scale the domain size use "Convert Units" or " Specify Scaling Factors" command.





**<u>Remark:</u>** Fluent thought you create the mesh in units of <u>m</u>. However, if your mesh is created in a different unit, such as <u>cm</u>, you must use Convert Units Command to scale the mesh into the right size. The values will be multiplied by the Scaling Factor.

ICEM: 1 mm -> Fluent: 1m -> Scale: mm, factor: 1/1000

Scale Mesh	X	Scale Mesh		<u> </u>
		Domain Extents		Scaling
Xmin (m)         -2.20847e-24         Xmax (m)         5	© Convert Units	min (mm) -2.20847e-24	Xmax (mm) 5	Convert Units     Specify Scaling Factors
Ymin (m) -1.38778e-17 Ymax (m) 0.6	© Specify Scaling Factors Mesh Was Created In	Ymin (mm) -1.38778e-17	Ymax (mm) 0.6	Meth Was Created In
Tmin (m) -6.93889e-18 Zmax (m) 0.15	Scaling Factors	Zmin (mm) -6.93889e-18	Zmax (mm) 0.15	scaling Factors
View Length United	× 1	View Length On La		× 1
m 🗸	Y 1	mm		Y 1
	Z 1			Z 1
	Scale Unscale			Scale Unscale
Close Help			Close Help	



CFD-NHT-EHT

# **Step 3: Choose the physicochemical model**

Based on the governing equations you are going to solve, select the related models in Fluent.

**<u>Remark:</u>** Understand the problem you are going to solve, and write down the right governing equations is the first and most important step for numerical simulation. Without background of "Fluid mechanics", "Heat Transfer" and "Numerical heat transfer", it is hard to complete this step for fluid flow and heat transfer problem.





Fluent is just a tool!



# **Step 3: Choose the physicochemical model**

# To select the model, the command is as follows:





CFD-NHT-EHT

### **Remark:** In our general teaching code



30/73





## **Step 4: Define the material properties**

Define the properties required for modeling! For fluid flow and heat transfer problem studied here,  $\rho$ ,  $c_p$  and  $\lambda$ should be defined.

**Solution Setup**→**Materials** 

In Fluent, the default fluid is air and the default solid is Al.

Click the Create/Edit button to add silicon and liquid water in our case.





#### Create/Edit Materials



×

Name		Material Type	Order Materials by
water-liquid		fluid	Name
' Chemical Form	nula		Chemical Formula
h2o <l></l>		Fluent Fluid Materials	Fluent Database
1		Mixture	User-Defined Database
		none	
Properties			
	Density (kg/m3)	constant  Edit 998.2	
Cp (Spe	cific Heat) (j/kg-k)	constant  Edit	
Thermal Cor	nductivity (w/m-k)	constant  Edit	
Viscosity (kg/m-s)		0.6 constant	
		0.001003	
		Change/Create Delete Close Help	



# However, it will happen that the material you need is not in the database. You can input it manually.

	Create/Edit Materials		×
Materials Materials	Name silicon	Material Type	Order Materials by
Fluid air water-liquid Solid aluminum silicon	Chemical Formula	FLUENT Solid Materials Silicon (si) Mixture	Chemical Formula  FLUENT Database User-Defined Database
	Properties Density (kg/m3) Constant 2330	Edit	
	Cp (Specific Heat) (j/kg-k) constant 712	▼ Edit	
	Thermal Conductivity (w/m-k) constant	▼ Edit	
Create/Edit Delete		▼	
	Change/Cre	eate Delete Close Help	]



# **Our general Code:**

# 12. GAMSOR

(1) Determine  $\Gamma_{\phi}$  for different variables:






### **Step 5: Define zone condition**

### Solution Setup→Cell Zone Condition

Cell Zone Cor	ditions	
Zone		
fluid		
Solid		
Phase	Type	
mixture		18
Edit	Copy Profiles	]
Parameters	Operating Conditions	]
Display Mesh		
Porous Formulation	n	
Superficial Velocit	city	
	у	

Each zone has its ID.

Each zone should be assigned a type, either fluid or solid.

Phase is not activated here. It can be edited under other cases, for example multiphase (多相流) flow model is activated. See Example A3.

Click Edit to define the zone condition of each zone.

😰 西安交通大學



**Porous media is** treated as a type of fluid zone, in which parameters related to orous media should given such be as porosity, permeability (渗透率), etc. We will discuss it in Example A2.







Frame motion and Mesh motion is used if the solid or the frame is moving.

Source term in need as a constant value or by user defined with .c file compiled if you need.

If T of the zone is fixed, you can select the Fixed value button.

P Fluid			
Zone Name			
fluid			
Matarial Nama			
water-liquid	▼ Edit		
Frame Motion 3D Fan Z	Source Terms		
Mesh Motion	Fixed Values		
Porous Zone			
Reference Frame Mesh Mot	tion P Jus Zone 3D Fa	an Zone Embedded LES Reaction	Source Te
Conical			
	Undet	- Free Place Teel	
	Updat	e From Plane Tool	
Direction-1 Vector			
X	1	constant 👻	E
	[		
	0	constant 🔻	
z	0	constant 👻	
Direction-2 Vector			_
x	0	constant -	
	<u> </u>	- Constaint	
Y	1	constant 🗸	
z	0	constant 🗸	
	1		

### **Step 6: Define the boundary condition**

Boundary condition definition is one of the most important and difficult step during Fluent simulation. General boundary conditions in Fluent can be divided into two kinds:

**1. BC at inlet and outlet:** pressure, velocity, mass flow rate, outflow...

**2. BC at wall:** wall, periodic, symmetric...

**<u>Remark:</u>** Interior cell zone and interior interface will also shown in the BC Window.



CFD-NHT-EHT

# For example, **Coupled-shadow** is listed here. It is the interface between fluid and solid zones.

#### It is treated as coupled, conjugate condition (流固耦合) **Boundary Conditions** 💶 Wall Problem Setup General Zone Zone Name Models coupled Dtm Materials coupled sounled-shadow Phases Adjacent Cell Zone Cell Zone Conditions llin fluid int fluid Boundary Conditions int solid Mesh Interfaces Shadow Face Zone out Dynamic Mesh coupled-shadow svmm Reference Values svmm:002 wall Solution Momentum Thermal Radiation Species DPM Multiphase UDS Wall Fi wall:003 Solution Methods Thermal Conditions Solution Controls Monitors Heat Flux Solution Initialization Heat Generation Rate (w/m3) 0 Coupled Calculation Activities Run Calculation atterial Name Results silicon Edit... Graphics and Animations Type ID Plots mixture wall Reports Edit... Copy... Profiles... Parameters... Operating Conditions... Display Mesh... Periodic Conditions... Highlight Zone Cancel 41/68 OK



CFD-NHT-EHT

### **Other BCs are as follows:**

### For fluid inlet: velocity inlet

Boundary Conditions		
Zone		
btm		
coupled hadow		
in		
int_fluid		
out		
symm		
wall		
wall:003		
Phase	Туре	ID
mixture 👻	velocity-inlet 👻	19
	axis	L
Edit	exhaust-fan	
Parameters Op	p intake-fan	
Display Mesh	interface	
Highlight Zone	outflow	
	outlet-vent	
$\square$	pressure-tar-field	
Help	pressure-outlet	
	symmetry	
L	wall	
	wall	

Velocity Inle	et		
Zone Name			
In			
Momentum Th	ermal Radiation Species	DPM Multiphase U	DS
Ve	locity Specification Method	Magnitude, Normal to Bour	ndary 🔹
	Reference Frame	Absolute	•
	Velocity Magnitude (m/s)	0.3	constant 👻
Supersonic/Initial Gauge Pressure (pascal)			
OK Cancel Help			

velocity inter	
one Name	
Nomentum Thermal Radiation Species DPM Multiphase UDS	
emperature (k) 203 15	
	10





### **Other BCs are as follows:**

### For fluid outlet: pressure outlet

	Pressure Outlet
Boundary Conditions	Zone Name
Zone btm coupled coupled-shadow in int_fluid int_solid out cymm symm:002 wall wall:003	Jout         Momentum       Thermal       Radiation       Species       DPM       Multiphase       UDS         Gauge Pressure (pascal)       0       constant       •         Backflow Direction Specification Method       Normal to Boundary       •         Radial Equilibrium Pressure Distribution       Average Pressure Specification       •         Target Mass Flow Rate       OK       Cancel       Help
	Proscure Outlet

Phase mixture	Type D pressure-outlet -
Edit	Copy Profiles
Parameters	Operating Conditions
Display Mesh	Periodic Conditions
Highlight Zone	

Pressure Outlet
Zone Name out
Momentum Thermal Radiation Species DPM Multiphase UDS
Backflow Total Temperature (k) 300 constant
OK Cancel Help





**Seven kinds of Pressure in Fluent** 

- 1. Atmospheric pressure (大气压)
- 2. Gauge pressure (表压): the difference between the true pressure and the Atmospheric pressure.
- 3. Absolute pressure (真实压力): the true pressure
  - = Atmospheric pressure + Gauge pressure
- 4. Operating pressure (操作压力) : the same as the reference pressure (参考压力) in our teaching code



### **Pressure in Fluent**

Absolute pressure (真实压力): the true pressure

- = **Reference Pressure** + **Relative Pressure**
- 5. Static pressure (静压): the difference between true pressure and operating pressure.
- The same as relative pressure.
- 6. Dynamic pressure (动压): calculated by  $0.5\rho U^2$
- is related to the velocity.
- 7. Total pressure (动压):
  - = Static pressure + dynamic pressure





46/68

### **Other BCs are as follows:**

#### For bottom surface: constant heat flux

	💶 Wall	
Boundary Conditions	Zone Name	
Zone	btm	
btm	Adjacent Cell Zone	
coupled-shadow	solid	
in int fluid		
int_solid	Homentain   mermail Radiation   Species   DPM   Hultiphase	
symm	This page is not applicable under current settings.	
symm:002 wall		
wall:003	See Wall	×
	Zone Name	
	btm	Take care of the
	Adjacent Cell Zone	• • • •
	solid	unit of heat flux
Phase Type ID	Momentum Thermal Radiation Species DPM Multiphase UD	S   Wall Film
mixture vall 17	Thermal Conditions	
Edit Copy Profiles	Heat Flux     Heat Flux	000000 constant 👻
Parameters Operating Conditions	Convection	Wall Thickness (m)
Display Mesh	© Radiation	
Highlight Zone	Mixed Heat Generation Rate (w/m3)	constant 👻
	Material Name	Shell Conduction
	Edit	
	ОК Сапсе	Help



CFD-NHT-EHT

### **Other BCs are as follows:**

For left and right solid and fluid surfaces: symmetry

The left and right boundary
for solid and fluid are set as
symmetry. Because the
calculation domain is a
typical part extracted from
the total district, which can
represent the heat transfer
and fluid flow characteristics.

Boundary Cor	Iditions
Zone	
btm coupled coupled-shadow	
in	
int_fluid int_solid	
out	
symm symmi002	
symm.002	
wall:003	
Phace	
Fridse Latation	
mixture	symmetry 15
Edit	Copy Profiles
Parameters	Operating Conditions
Display Mesh	Periodic Conditions
Highlight Zone	



# **Other BCs are as follows:**

### For top surface, end surface: adiabatic and non-

### slipping wall

Boundary Conditions	🞴 Wall	
Zone	Zone Name	🖳 Wall
Zone btm coupled coupled-shadow in int_fluid int_solid out symm:002 wall wall:003	up_wall_f         Adjacent Cell Zone         fluid         Momentum       Thermal         Radiation         Wall Motion       Motion         Image: Stationary Wall       Image: Relation of the state of the	Zone Name          Up_wall_f         Adjacent Cell Zone         fluid         Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film         Thermal Conditions         Image: Heat Flux         Image: Temperature Convection         Image: Radiation Radiation         Image: Radiation Rate (w/m3)
Phase Type mixture wall Edit Copy Profiles. Parameters Operating Conditions Display Mesh Periodic Conditions	Marangoni Stress Wall Roughness Roughness Height (m) Roughness Constant 0.5	Via System Coupling via Mapped Interface  Material Name aluminum Edit  Adiabatic wall
		OK Cancel Help





### **Step 7: Solution setup: algorithm and scheme**

Remark:In Fluent, fortheSIMPLEseriesalgorithms, onlySIMPLEandSIMPLECareincluded.

Review:What is thedifferencebetweenSIMPLE, SIMPLEC andSIMPLER?

Meshing	Solution Methods
Mesh Generation	Pressure-Velocity Coupling
Solution Setup General Models	Scheme SIMPLE
Materials Phases	Spatial Discretization
Cell Zone Conditions Boundary Conditions	Least Squares Cell Based
Dynamic Mesh Reference Values	Second Order
Solution Solution Methods Solution Controls Monitors	Momentum Second Order Upwind Energy Second Order Upwind
Solution Initialization Calculation Activities Run Calculation Results	Transient Formulation
Graphics and Animations Plots Reports	Non-Iterative Time Advancement  Frozen Flux Formulation  Pseudo Transient
	High Order Term Relaxation Options      Default
	Help



- 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.

### **Green-Gauss Theory:**

The averaged gradient over a control domain is:

$$<\nabla\phi>=rac{1}{V_C}\int\limits_{V_C}\nabla\phi dV$$



Using the Gauss integration theory (高斯定理), the volume integral (体积分) is transformed into a surface integral (面积分):

$$\langle \nabla \phi \rangle = \frac{1}{V_C} \int_{V_C} \nabla \phi dV = \frac{1}{V_C} \oint \phi \cdot \mathbf{n} dS$$

In the presence of discrete faces, the above equation can be written as:

$$\langle \nabla \phi_{\text{centroid}} \rangle V_C = \sum \phi_f \cdot \mathbf{S}$$
  
 $\phi_f$   
 $\phi_f$   
 $\phi_f$   
 $\phi_f$ 





$$\nabla \phi_{\text{centroid}} V_C = \sum \phi_f \cdot \mathbf{n} S$$

The problem of calculating gradient is transferred into the following equation: How to determine  $\phi_f$  at the face?

1. Green-Gauss Cell-Based (格林-高斯基于单元法)

Calculate  $\phi_f$  using cell centroid values  $\phi_f$  (网格中心点).

$$\phi_f = \frac{\phi_{C0} + \phi_{C1}}{2}$$

3 历安交通大学

2. Green-Gauss Node-Based (格林-高斯基于节点法) Calculate  $\phi_f$  by the average of the node values. (面顶 点的代数平均值)  $\phi_n$ 

Nf: number of nodes on the face,  $\Phi_n$ : node value.  $\Phi_n$ , is calculated by weighted average of the cell values surrounding the nodes  $\Phi_{Ci}$ .

 $\phi_f = \frac{1}{N} \sum \phi_n \qquad \phi_n = \sum_{i}^{N_{\text{cells}}(n)} \phi_{c_i} w_{c_i,n}$ 

**<u>Review:</u>** the node-based method is more accurate than the cell-based method.



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 basic idea is as follows. Consider two cell centroid
C<sub>0</sub> and C<sub>i</sub>, and their distance vector as δr. Then, the following equation

$$\phi_{Ci} = \phi_{C0} + (\nabla \phi) \cdot (\mathbf{r}_{Ci} - \mathbf{r}_{C0})$$

is exact only when the solution field is linear! In other words, there is no second-order term for Taylor expansion of  $\phi$ !





### For a cell centroid $C_0$ with N neighboring nodes $C_i$ ,

$$\Phi_{Ci} = \phi_{Ci} - \left[\phi_{C0} + (\nabla \phi) \cdot (\mathbf{r}_{Ci} - \mathbf{r}_{C0})\right]$$
  
True value Calculated value

Making summation of all these  $\Phi_{Ci}$  with a weighting factor  $w_i$ 

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



CFD-NHT-EHT

## Therefore, to calculate the gradient $\nabla \phi$ is to find the one leading to the minimum $\xi$ !

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

### This is the idea of Least-Squares method.

Remark: On irregular (不规则) unstructured meshes, the accuracy of the least-squares gradient method is comparable to that of the node-based gradient. However, it is more computational efficient compared with the

node-based gradient.

# **Pressure calculation: to calculate the pressure value at the interface using centroid value.**

Meshing	Solution Methods	
Mesh Generation	Pressure-Velocity Coupling	
Solution Setup	Scheme	PCentroid
General Models	SIMPLE	
Materials	Spatial Discretization	
Phases	Gradient	
Cell Zone Conditions Boundary Conditions		$ \rho_{f}$
Mesh Interfaces	Pressure	
Dynamic Mesh	Second Order	
Reference Values	Momentum	
Solution	Second Order Upwind	
Solution Methods	Energy	
Monitors	Second Order Upwind 👻	
Solution Initialization		
Calculation Activities	·	
Run Calculation	Transient Formulation	Pressure
Results		Second Order
Graphics and Animations Plots	Non-Iterative Time Advancement	
Reports	Frozen Flux Formulation Reseudo Transient	Second Order
	High Order Term Relaxation Ontions	Standard
		PRESTO!
	Default	Linear
		Body Force Weighted
	Help	





### 1. Linear scheme

Computes the face pressure use the average of the pressure values in the adjacent cells.

$$P_f = \frac{P_{C0} + P_{C1}}{2}$$

### 2. Standard scheme

Interpolate the pressure using momentum equation coefficient.

$$P_{f} = \frac{\frac{P_{c0}}{a_{P,c0}} + \frac{P_{c1}}{a_{P,c1}}}{\frac{1}{a_{P,c0}} + \frac{1}{a_{P,c1}}}$$





### **3. Second Order**

Calculate the pressure value using a central difference scheme

$$P_f \approx \frac{P_{C0} + \nabla P_{C0} \mathbf{r}_{C0} + P_{C1} + \nabla P_{C1} \mathbf{r}_{C1}}{2}$$

### 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.



### For convective term scheme, we are very familiar!

Momentum	Mashina	Solution Methods
Second Order Upwind 🚽	Mesh Generation	Pressure-Velocity Coupling
First Order Upwind Second Order Upwind Power Law QUICK Third-Order MUSCL	Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Methods	Pressure-Velocity Coupling Scheme SIMPLE  Spatial Discretization Gradient Least Squares Cell Based Pressure Second Order Momentum Second Order Upwind Energy
	Monitors Solution Initialization Calculation Activities	Second Order Upwind
Energy	Run Calculation	Transient Formulation
Second Order Upwind	Results Graphics and Animations	Non-Iterative Time Advancement
First Order Upwind	Reports	Frozen Flux Formulation Pseudo Transient
Second Order Upwind		High Order Term Relaxation Options
Power Law		Default
QUICK		
Third-Order MUSCL		Help



### **Step 7: Solution setup: relaxation**

Under-relaxation is adopted to control the change rate of simulated variables in subsequent iterations.

The relaxation factor α for each variable has been optimized for the largest possible.

Solution Controls	
Under-Relaxation Factors	
Pressure	*
0.3	
Density	
1	
Body Forces	
1	
Momentum	
0.7	
Energy	
1	_
Default	
Equations	

In some cases, if your simulation is not converged, and you are sure there is no problem with other setting, you can try to reduce  $\alpha$ !



CFD-NHT-EHT

#### **Step 7: Solution setup: monitors**

Similar to "Print" function in our teaching code, you can use Monitors in Fluent to setup a certain number of variables to monitor the iteration process of the simulation.

The Residuals are the most important values to be monitored. You can set the related values.

Residual Monitors					×
Options	Equations	Monitor (	berk Convergen	re Absolute Criteria	
V Print to Console	continuity			0.001	
Window	x-velocity			0.001	
Iterations to Plot	y-velocity			0.001	
	energy	<b>V</b>		1e-06	Ŧ
	Residual Values			Convergence	Criterion
Iterations to Store	Normalize		Iterations	absolute	<b></b>
	Scale				
	Compute Loca	al Scale			
OK Plot	Renormaliz	e C	ancel H	lelp	





 $\mathbf{v}_{\mathbf{v}}$ 

### **Step 8: Initialization**

tialization Methods
<ul> <li>Hybrid Initialization</li> <li>Standard Initialization</li> </ul>
More Settings
Patch
Reset DPM Sources Reset

Hybrid Initialization	X
General Settings Turbulence Settings Species	Settings
Number of Iterations 10	
Explicit Under-Relaxation Factor	
Scalar Equation-0	
Scalar Equation-1	

The default selection is Hybrid initialization (混合初始化).

The initial pressure and velocity field you give usually are not consistent, in other words, not meet the NS equation.

In SIMPLER algorithm, we solved an additional Poisson equation for pressure based on given velocity.





The Hybrid initialization method is similar that Poisson equation is solved to initialize the velocity and pressure equation. You can set the number of iterations to make sure the initial velocity and pressure are consistent.

Hybrid Initialization	x
General Settings Turbulence Settings Species Setti	ngs
Number of Iterations 10	
Explicit Under-Relaxation Factor	
Scalar Equation-0	
Scalar Equation-1	



CFD-NHT-EHT

### Or you can simply chose Standard initialization method.

Click Compute from, the drop-down list will show, and you can select an region.

Compute from	
	<b>-</b>
all-zones	
in	
out	
wall	
btm	
coupled	
wall:003	
coupled-shadow	

Initialization Methods          Hybrid Initialization         Standard Initialization         Compute from         Reference Frame	
Hybrid Initialization     Standard Initialization Compute from Reference Frame	
Compute from Reference Frame	
Reference Frame	
Reference Frame	•
<ul> <li>Relative to Cell Zone</li> <li>Absolute</li> </ul>	
Initial Values	_
Gauge Pressure (pascal)	
0	
X Velocity (m/s)	1
0	
Y Velocity (m/s)	
0	
Temperature (k)	
300	
	l
Initialize Reset Patch	





# The eight steps for preparing a Fluent simulation have been completed!

- 1. Read mesh
- 3. Choose model
- **5. define zone condition**
- 7. Solution step
- 9. Run the simulation.

- 2. scale domain
- **4.define material**
- 6. define boundary condition
- 8. Initialization
- **10. Post-process**

### **Step 9: Run the simulation**

What should you do in this step? Just stare at the monitor to hope that the residual curves are going down for a steady problem.



### **Diverged?** Go back to Steps 1 to 8.





### **Review:** The 10 steps for a Fluent simulation:

- 1. Read and check the mesh: mesh quality.
- 2. Scale domain: make sure the domain size is right.
- **3.** Choose model: write down the right governing equation is very important.
- 4. Define material: the solid and fluid related to your problem.
- 5. Define zone condition: material of each zone and source term
- 6. Define boundary condition: very important
- 7. Solution step: algorithm and scheme. Have a background of NHT.
- 8. Initialization: initial condition
- 9. Run the simulation: monitor the residual curves and certain variable.
- **10. Post-process: analyze the results.**





**Re=44** 



Streamline and velocity distribution







### **Step 10: Post-process: Data reduction**

The Reynolds number (*Re*) is expressed as follow:

$$Re = \frac{\rho u_m D_h}{\mu}$$

$$D_h = \frac{2H_c W_c}{H_c + W_c}$$

<i>u</i> (m/s)	0.3	0.6	0.9	1.2	1.5
Re	44	88	132	176	220
					69/6





### **Friction factor**

$$f = \frac{2D_h \Delta P}{L_t \rho u_m^2}$$

### Heat transfer coefficient

$$h_{ave} = \frac{q_w A_s}{A_{con}(T_{w,ave} - T_{f,ave})}$$

$$q_w A_s = h A_{con} \Delta T_m = C_p M (T'' - T')$$

### **Average Nusselt number**

$$Nu_{ave} = \frac{h_{ave}D_h}{\lambda_f}$$



@ 西步交通大學

Re	44	88	132	176	220
h (Wm <sup>-2</sup> K <sup>-1</sup> )	13999.6	18453.6	22140	25323.2	28336
Nu	3.5	4.6	5.5	6.3	7.1
Δ <b>P</b> ( <b>P</b> a)	354	892.8	1590	2436.6	3428
<i>T<sub>W</sub></i> (K)	334.45	321.98	316.5	313.3	311.1

1 历安交通大学



