



Numerical Heat Transfer

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



Instructor Li Chen, WenQuan Tao

CFD-NHT-EHT Center Key Laboratory of Thermo-Fluid Science & Engineering Xi'an Jiaotong University Xi'an, 2019-12-16







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16日,西安



CFD-NHT-EHT

Class intermediate

 13. A1 Flow and heat transfer in microchannels with secondary channels

 (具有二次通道的微通道中流动换热)

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)
- 3: Drawing inferences for each example (举一反三)_{4/68}



13_A1: Flow and heat transfer in microchannels with secondary channels

Background:

- Because of the integration(集成化) of electron component (电 子元件), the heat flux of a EC greatly increases, even reaches MW·m⁻² order of magnitude.
- Traditional cooling techniques cannot meet the cooling demand of such high heat flux.

Microchannel is proposed for this purpose.





What is "Microscale" ?

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

Depending on Kn number, the flow may be in continuum, slip, transition or even free molecular flow region.

The NS equation should be modified or even is not applicable.

2. The relative importance of affecting factors changes.

Fluid flow is controlled by body forces and surface forces

body forces: ~m³ surface forces: ~m²

surface forces/body forces: ~m⁻¹; surface force becomes stronger as length scale decreases.



λ

Flow regime

H.-S. Tsien, 1946

西安交通大學

Knudsen: Kn= λ /L







Known

Steady single phase fluid flow and heat transfer of water in a copper microchannel. There are secondary channels in the domain, as displayed in Fig. 1.







Boundary conditions

	Heat transfer	Fluid flow
Inlet	F:300K; S: adiabatic	F:Velocity inlet; S:wall
Outlet	adiabatic	F:1atm; S:wall
Bottom	Heat flux(1×10 ⁶ W·m ⁻²)	Wall
Up	Adiabatic	Wall
Side	Symmetry	Symmetry





Find: average Nusselt number (Nu_{ave}), average temperature of bottom surface (T_b) and resistance factor (f) at different *Re* numbers (100, 200, 300, 400, 500).

Assumptions:

- (1) When *Kn* is less than 10⁻³, 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: construct the reasonable physical model and write down the right governing equation, BC and IC is the first and most important step before using 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 $\nabla (\rho u u) = -\nabla p + \eta \nabla^2 u$

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

I Fluent Launcher	
ANSYS	Fluent Launcher
Dimension 2D 3D	Options Double Precision Meshing Mode
Display Options Display Mesh After Reading Embed Graphics Windows Workbench Cafor Scheme	Processing Options Serial Parallel
軠 Show More Options	
<u> </u>	<u>C</u> ancel <u>H</u> elp ▼

- 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

For most cases 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. 12/68





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.

						mesh	
	Fluent@DESKTOP-2C2B			_		Slitting wall zone 29 into a coupled wall	-
File	Mesh Define Solve	Adapt	Surface Display	File \rightarrow	Read→Mesh	materials, interface,	
	Read	\rightarrow (Mesh	T	·	domains,	
	Write	>	Case			zones, inter_surface_sf-shadow	
	•		Data			down_wall_s	
	Import	>				up_wall_s	
	Export	>	Case & Data			up_wall_f	
	Export to CFD-Post		PDF	uality		inter_surface_sf	
						wall_left_f	
	Solution Files		ISAT Table			wall_left_s wall_right_f	
	Interpolate		DTRM Rays			wall_right_s	
			View Factors			out s	
	FSI Mapping	>				out_f	
	Save Picture		Profile			in_s	
						in_f	
	Data File Quantities		Scheme			int_fluid	
	Batch Options		Journal			int_solid	
						fluid solid	
	Exit		wall			Done.	
	ution Initialization						
		ravity				Preparing mesh for display	1
	n Calculation	aviey		Units		Done.	13/68
100							0



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

General	
Mesin	
Scale Display	Check Report Quality
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute Relative
Time	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

Make sure the unit is right.

Scale Me	esh				×
Bomain Exten	its				Scaling
Xmin (m) 0		Xmax (m)	0.225		 Convert Units Specify Scaling Factors
Ymin (m) -(0.015	Ymax (m)	0.015		Mesh Was Created In
Zmin (m)		Zmax (m)	0.04		Scaling Factors
View Length L	Jnit In				× 0.001
m	▼				Y 0.001
					Z 0.001
					Scale Unscale
		C	lose	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	×
Scale Mesh	Domain Extents	Scaling
Domain Extents S	Xmin (m) -2.77556e-21 Xmax (m) 0.00225	 Convert Units Specify Scaling Factors
Xmin (m) -2.77556e-17 Xmax (m) 2.25	Ymin (m) -1.40379 -14 Ymax (m) 0.0003	Mesh Was Created In
Ymin (m) -1.40379e-11 Ymax (m) 0.3	Zmin (m) -6.93889e-11 Zmax (m) 0.0004	Scaling Factors
Zmin (m) -6.93889e-18 Zmax (m) 0.4	View Length Unit In m	Y 0.001
View Length Unit In		Z 0.001
	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, ρ , c_p and λ 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 copper and liquid water in our case.





Create/Edit Materials



×

Name		Material Type	Order Materials by
water-liquid		Material Type	Name
Chemical Form	nula		Chemical Formula
h2o <l></l>		Fluent Fluid Materials water-liquid (h2o <l>)</l>	Fluent Database
1		Mixture	User-Defined Database
		none	
Properties			
	Density (kg/m3)	constant	
		998.2	
Cp (Spe	cific Heat) (j/kg-k)	constant	
		4182	
Thermal Cor	nductivity (w/m-k)	constant Edit	
		0.6	
	Viscosity (kg/m-s)	constant	
		0.001003	
1			
		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 U	Material Type solid	Order Materials by
Fluid air Solid u aluminum	Chemical Formula	Fluent Solid Materials	 Chemical Formula Fluent Database User-Defined Database
	Properties Density (kg/m3) Cp (Specific Heat) (j/kg-k) Thermal Conductivity (w/m-k)	constant 19070 constant ↓ Edit 116 constant ↓ Edit 27.4	*
Create/Edit Delete		Change/Create Delete Close He	elp



Our general Code:

12. GAMSOR

(1) Determine Γ_{ϕ} for different variables:







Step 5: Define zone condition

Solution Setup→Cell Zone Condition

Cell Zone Cor	ditions	
Zone		
fluid		
solid		
Phase	Туре	
mixture	✓ fluid	18
Edit	Copy Profiles]
Parameters	Operating Conditions]
Display Mesh		
Porous Formulation		
 Superficial Velo Physical Velocit 		
	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 porous media should be given such as porosity, permeability (渗透率), etc. We will discuss it in Example A2.

Fluid	
Zone Name fluid	
Material Name water-liquid Frame Motion 3D Fan Zone Source Ter Hesh Histor	
	3 Fan Zone Embedded LES Reaction Source T
Conical	2
Viscous Resistance (Inverse Absolute Permeab Direction-1 (1/m2) 2.111e+08	constant •
Direction-2 (1/m2) 2.111e+08 Direction-3 (1/m2) 2.111e+08	constant
Inertial Resistance	
Alternative Formulation Direction-1 (1/m) 0	constant 💌
Direction-2 (1/m)	constant 💌
Direction-3 (1/m)	constant •
J. Power Law Model	
	OK Cancel Help





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

Add 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.

E Fluid			
Zone Name			
fluid			
Material Name water-liquid	▼ Edit		
🔲 Frame Motion 📃 3D Fan Z	o 📃 Source Terms		
Mesh Motion	Fixed Values		
V Porous Zone			
Reference Fram	tion P Jus Zone 3D Far	Zone Embedded LES Reaction	Source Te
Conical			
	Updata	From Plane Tool	<u> </u>
	Opuate	From Flane roor	
Direction-1 Vector			
X	1	constant 👻	=
Y			
	0	constant 👻	
Z	0	constant 👻	
	-		
Direction-2 Vector			
x	0	constant 👻	
	v	constant •	
Y	1	constant 🔹	
z	0	constant 🔹	

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.





For example, inter_surface_sf: 29 is listed here. It is the interface between fluid and solid zones.

It is treated as coupled, conjugate condition (流固耦合)







Other BCs are as follows: For fluid inlet: velocity inlet

Zone	
down_wall_s	
in_f	
int_fluid	
int_solid	
int_solid-shadow	
inter_surface_sf	
inter_surface_sf-shadow	
out_f out_s	=
up_wall_f	
up_wall_s	
wall_left_f	
wall_left_s wall_right_f	
wall_right_s	
	Ψ.
Phase Type ID	
mixture velocity-inlet 21	_
Edit C _c exhaust-fan	
Parameters Op intake-fan	
Display Mesh	
Highlight Zone outflow outlet-vent	
pressure-far-field	
Help pressure-outlet	
pressure oddet	
velocity-inlet	

💶 Velocity Inl	let		×
Zone Name in_f			
Momentum T	hermal Radiation Species	DPM Multiphase	UDS
Velocity Specification Method Magnitude, Normal to Boundary			
	Reference Frame	Absolute	•
	Velocity Magnitude (m/s)	0.41	constant 👻
Supersonic/Init	tial Gauge Pressure (pascal)	0	constant 🗸
OK Cancel Help			

Velocity Inlet	×
Zone Name in_f	
Momentum Thermal Radiation Species DPM Multiphase UDS	1
Temperature (k) 300 constant	
OK Cancel Help	





Х

 \sim

 \sim

×

Other BCs are as follows:

For fluid outlet: pressure outlet

bermal Radiation Species DPM Multiphase Gauge Pressure (pascal) tion Specification Method Normal to Boundary	UDS Constant
Gauge Pressure (pascal)	_
	Thermal Radiation Species DPM Multiphase

OK

Help

Cancel





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





68

Other BCs are as follows:

For bottom surface: constant heat flux

Boundary Conditions		💶 Wall
Zone down_wall_s III_f in_s int_fluid int_solid int_solid-shadow inter_surface_sf inter_surface_sf-shadow out_f out_s	A E	Zone Name down_wall_s Adjacent Cell Zone solid Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film This page is not applicable under current settings.
uur_wall_f up_wall_s wall_left_f wall_left_s wall_right_f wall_right_s	*	Wall Zone Name Image: Cone Name down_wall_s Image: Cone Adjacent Cell Zone Image: Cone solid Image: Cone Momentum Thermal Radiation Species DPM Multiphase UDS
Phase Type ID mixture wall 32 Edit Copy Profiles Parameters Operating Conditions Display Mesh Periodic Conditions Highlight Zone		Thermal Conditions Heat Flux Temperature Convection Radiation Wall Thickness (m) Mixed Heat Generation Rate (w/m3) via Mapped Interface Shell Conduction Material Name Edit
		OK Cancel Help 33





Other BCs are as follows:

For left and right 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.		







Other BCs are as follows: For top surface, solid in and out surfaces: adiabatic and non-slipping wall

Boundary Conditions		
· · · · · ·	Zone Name	🖸 Wall
Zone	up_wall_f	Zone Name
down_wall_s	Adjacent Cell Zone	up_wall_f
in_s	fluid	Adjacent Cell Zone
int_solid		fluid
int_solid-shadow	Momentum Thermal Radiation	
inter_surface_sf inter_surface_sf-shadow	Wall Motion Motion	Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film
out_f	Stationary Wall √ Rela	
out_s up_wall_f	O Moving Wall	Heat Flux Heat Flux (w/m2)
up_wall_s	Shear Condition	Convection Wall Thickness (m)
wall_left_f wall_left_s	No Slip	Convection Wall Thickness (m) 0
wall_right_f	Specified Shear Specularity Coefficient	Mixed Heat Generation Rate (w/m3) 0 constant
wall_right_s	Marangoni Stress	Via System Coupling
	Wall Roughness	1 Layer
		Material Name
Phase Type	Roughness Height (m)	aluminum 👻 Edit
mixture 🔻 wall	Roughness Constant 0.5	Adiabatic wall
Edit		
Parameters Operating Cor		
Display Mesh Periodic Cond		OK Cancel Help
Highlight Zone		





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	Scheme
General Models	SIMPLE
Materials	Spatial Discretization
Phases	Gradient
Cell Zone Conditions Boundary Conditions	Least Squares Cell Based
Mesh Interfaces	Pressure
Dynamic Mesh	Second Order 🗸
Reference Values	Momentum
Solution Solution Methods	Second Order Upwind 🗸
Solution Controls	Energy
Monitors	Second Order Upwind 👻
Solution Initialization Calculation Activities	
Run Calculation	Transient Formulation
Results	
Graphics and Animations	Non-Iterative Time Advancement
Plots Reports	Erozon Flux Formulation
	Pseudo Transient High Order Term Relaxation Ontions
	Default
	Help
	50/08


- 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.

Green-Gauss Theory:

The averaged gradient over a control domain is:

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



38/68

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:

$$<\nabla\phi_{\text{centroid}}>V_{C}=\sum\phi_{f}\cdot\mathbf{S}$$

$$\phi_{f}$$

$$\phi_{f}$$

$$\phi_{f}$$





39/68

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

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

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

Calculate ϕ_f using cell centroid values ϕ_{C1} (网格中心点).

$$\phi_f = \frac{\phi_{c0} + \phi_{c1}}{2} \qquad \phi_{c0}$$

3 历安交通大学

2. Green-Gauss Node-Based (格林-高斯基于节点法) Calculate ϕ_f by the average of the node values. (面顶 点的代数平均值) $\phi_f = \frac{1}{N_c} \sum \phi_n$ $\phi_n = \sum_{i}^{N_{cells}(n)} \phi_{c_i} w_{c_i,n}$ ϕ_f

Nf: number of nodes on the face, ϕ_n : node value. ϕ_n , is calculated by weighted average of the cell values surrounding the nodes ϕ_{Ci} .

<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 C₀ and C_i, 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 ϕ !





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 Φ_{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 ξ !

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

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	m
Solution Setup	Scheme	PCentroid
General Models	SIMPLE	
Materials	Spatial Discretization	
Phases	Gradient	
Cell Zone Conditions Boundary Conditions	Least Squares Cell Based	p_f
Mesh Interfaces	Pressure	
Dynamic Mesh	Second Order	
Reference Values	Momentum	
Solution	Second Order Upwind	
Solution Methods Solution Controls	Energy	
Monitors	Second Order Upwind 👻	
Solution Initialization		
Calculation Activities Run Calculation		
Results	Transient Formulation	Pressure
Graphics and Animations		Second Order 👻
Plots	Non-Iterative Time Advancement	Second Order
Reports	Pseudo Transient	Second Order
	High Order Term Relaxation Options	Standard
	Default	PRESTO!
		Linear Redu Tassa Weishhad
		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.

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





46/68

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

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 α !



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 Image: Options <t< th=""><th>Equations Residual continuity x-velocity y-velocity</th><th>Monitor C</th><th>heck Convergence V V</th><th>Absolute Criteria 0.001 0.001 0.001</th><th></th></t<>	Equations Residual continuity x-velocity y-velocity	Monitor C	heck Convergence V V	Absolute Criteria 0.001 0.001 0.001	
1000	energy			1e-06	~
	Residual Values			Convergence C	riterion
Iterations to Store	Normalize		Iterations	absolute	•
	V Scale				
	Compute Loca	al Scale			
OK Plot	Renormaliz	e Ca	ancel Hel	p	





Step 8: Initialization

Hybrid Initialization Standard Initialization More Settings Patch
Patch
Reset DPM Sources Reset Statistics

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



CFD-NHT-EHT

Colution Initialization

Or you can simply chose Standard initialization method.

	SUIUUUN IIIIUANZAUUN
Click Compute from, the drop-down	Initialization Methods
	Hybrid Initialization
list will show, and you can select an	Standard Initialization
nst will show, and you can select an	Compute from
•	 The second second
region.	Reference Frame
	Relative to Cell Zone
	Absolute
Compute from	Initial Values
	Gauge Pressure (pascal)
	0
	V Valasita (n. (r.)
all-zones	X Velocity (m/s)
	0
wallright	Y Velocity (m/s)
	0
walleft	=
walldown	Temperature (k) 300
	300
wallup	
walldown:009	
walldown:011	
walluar012	-
wallup:012	, [Teitietine] [Deceth]
wallup:013	Initialize Reset Patch
Transpire as	Reset DPM Sources Reset Statistics





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.**







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)	0.5	1	1.5	2	2.5
Re	100	200	300	400	500
					50/68





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

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



Re	100	200	300	400	500
Nu	6.21	7.89	9.158	10.2	11.12
$\Delta P(\mathbf{Pa})$	979.63	2262.75	3793.16	5532.08	7451.92
f	0.691	0.399	0.297	0.244	0.21
<i>T_W</i> (K)	321.68	316.21	313.73	312.22	311.17

3 历安交通大学











Geometry parameters optimization for a microchannel heat sink with secondary flow channel



Xiaojun Shi*, Shan Li, Yingjie Mu, Bangtao Yin

School of Mechanical Engineering, Xi'an Jiaotong University, Xi'an 710049, China





(1) あ安え近大学

