

# Numerical Heat Transfer

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



**Instructor Tao, Wen-Quan; Chen, Li**

**CFD-NHT-EHT Center**

**Key Laboratory of Thermo-Fluid Science & Engineering**

**Xi'an Jiaotong University**

**Xi'an, 2017-Dec.-26**

# 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 陶文铨, 陈 黎

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  
2017年12月26日, 西安

# 第 13 章 求解流动换热问题的Fluent软件应用举例

**13.1 Heat transfer with source term**

**13.2 Unsteady cooling process of a steel ball**

**13.3 Lid-driven flow and heat transfer (Homework of Chapter 1)**

**13.4 Flow and heat transfer in a micro-channel (4-19)**

**13.5 Flow and heat transfer in chip cooling (4-19)**

**13.6 Flow and heat transfer in porous media**

**13.7 Flow and heat transfer in air film cooling**

## 第 13 章 求解流动换热问题的Fluent软件应用举例

**13.1 有内热源的导热问题**

导热问题

**13.2 非稳态圆球冷却问题**

**13.3 顶盖驱动流动换热问题**

混合对流问题

**13.4 微通道内流动换热问题**

微通道问题

**13.5 芯片冷却流动换热问题**

**13.6 多孔介质流动换热问题**

多孔介质流动

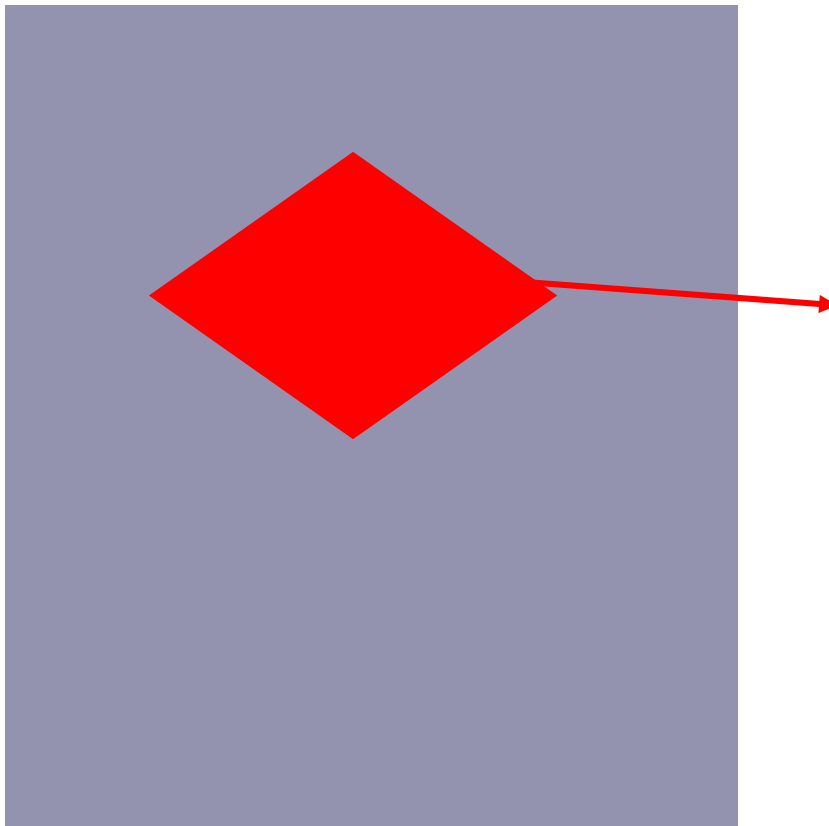
**13.7 气膜冷却流动换热问题**

湍流

# Review

## Example 2

### Patching (修补) Values in Selected Cells



**Domain**

**Sub-region need to Patch**

- 1. Define the sub-region**
- 2. Use Patch to specify related variables.**

# For transient problem you have to define

time stepping method, time step size, the max iteration per time step

Max iteration per time step

Inner iteration times

Time step size

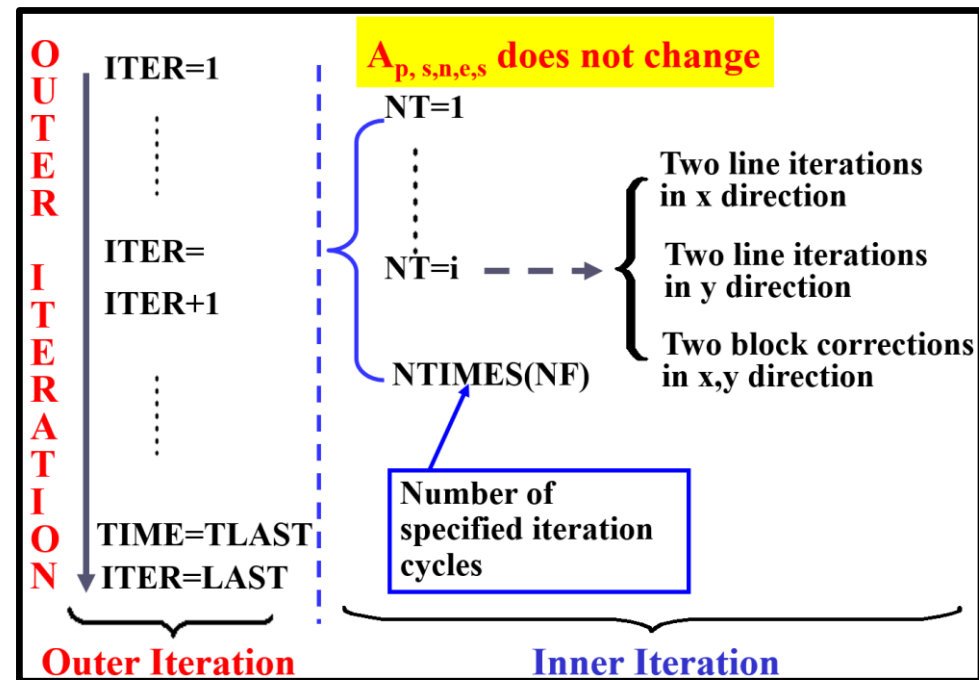
Outer iteration

Time stepping method

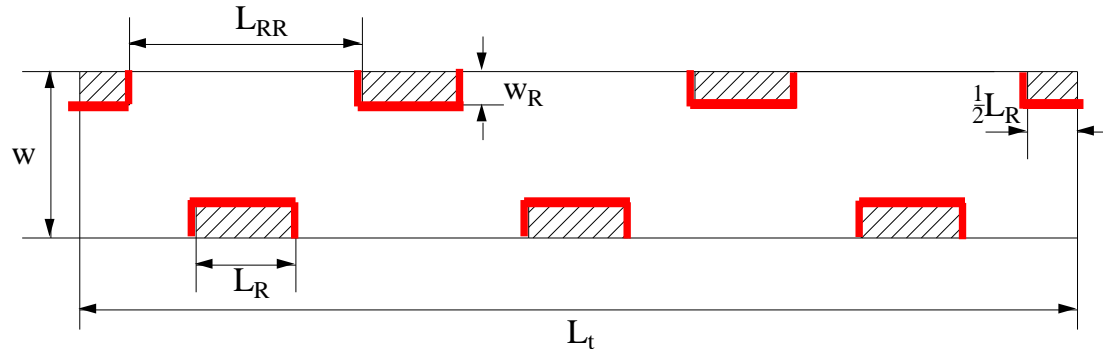
Fixed

Adaptive method

## Teaching code



## Example 4: Fluid-solid interface



**This wall type has fluid zone and solid zone on each side. This wall is called a “two-sided-wall”.**

**When such kind wall is read into Fluent, a “shadow” (影子) zone is automatically created.**

There are three options for the temperature boundary conditions of such “two-sided-wall”.

## Thermal Conditions

- Heat Flux
- Temperature
- Coupled

- Heat flux
- Temperature
- Coupled

If you choose “**Coupled**”, no additional information is required. The solver will calculate heat transfer directly from the solution of adjacent cells. **Such wall is not a boundary.**

# Uncoupled The original two side wall

Adjacent to fluid

Adjacent to solid

Specify BC for the fluid side

Thermal Conditions

- Heat Flux
- Temperature
- Coupled

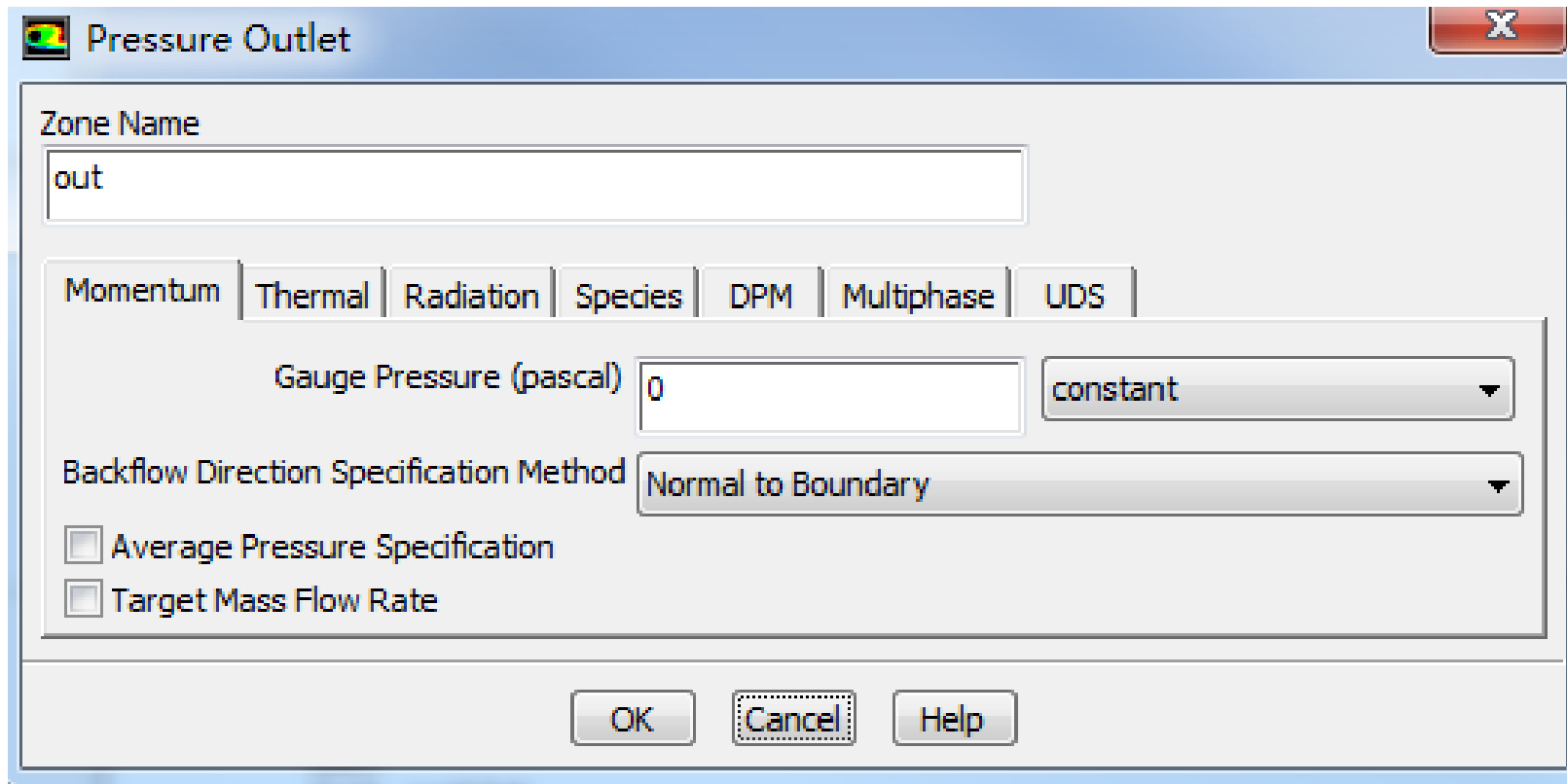
Specify BC for the solid side

Thermal Conditions

- Heat Flux
- Temperature
- Coupled

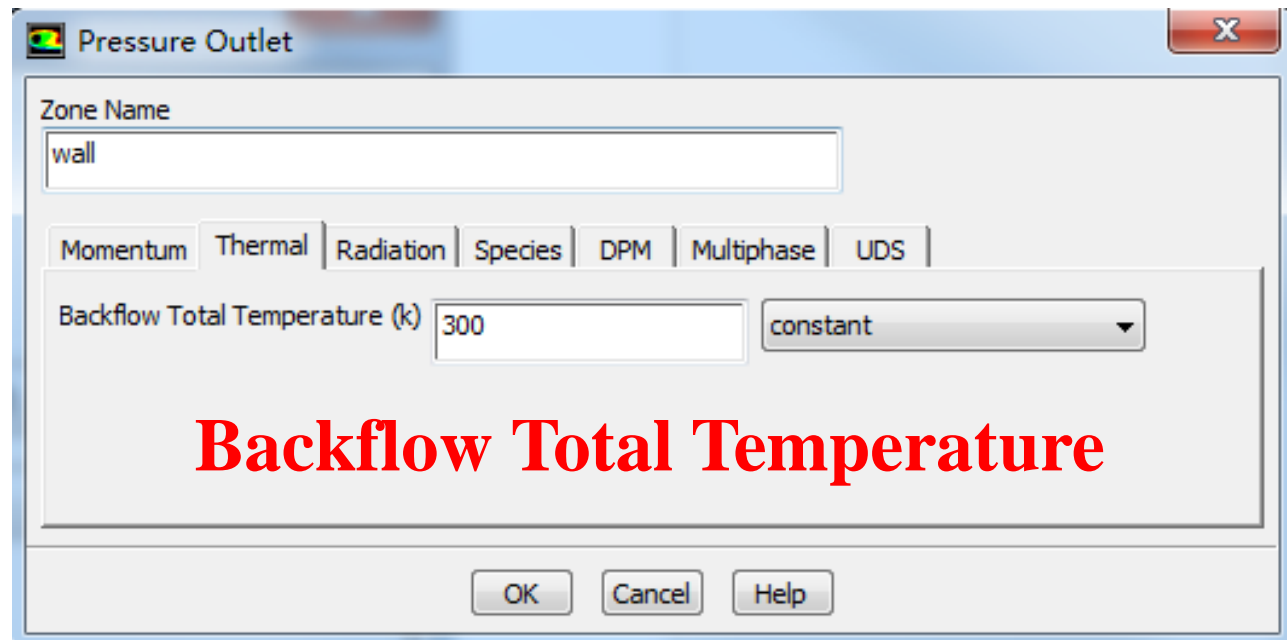
Its shadow created by Fluent

# Pressure outlet boundary condition



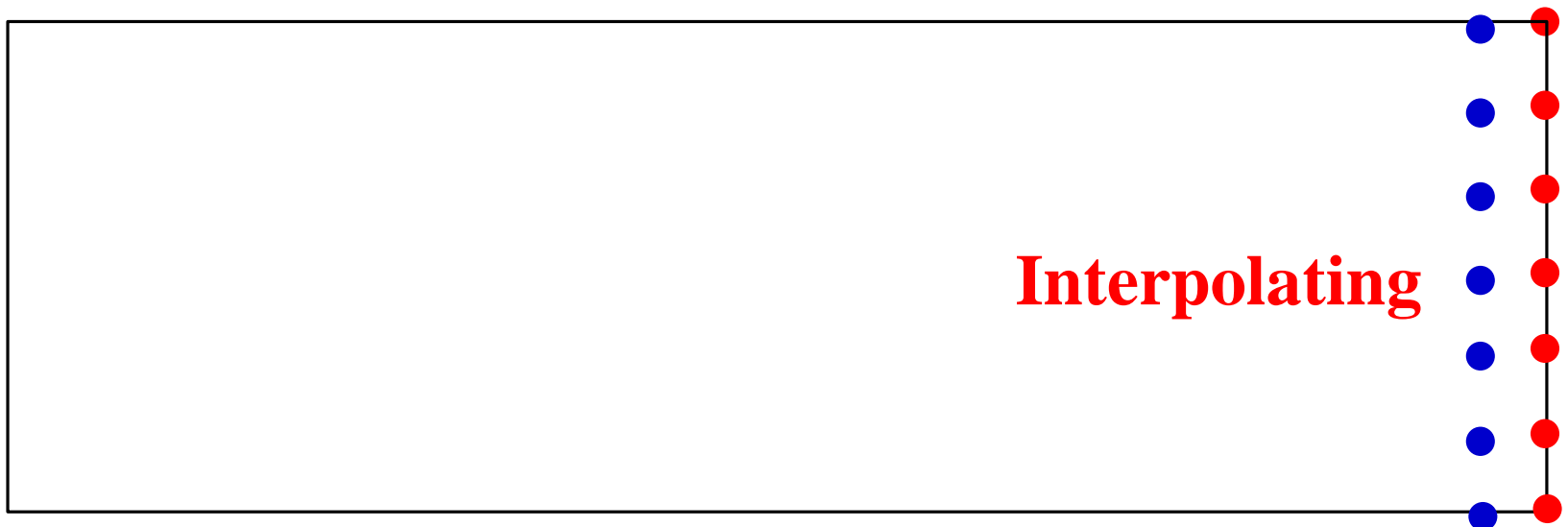
**Gauge Pressure (表压)**

For pressure outlet boundary condition, Fluent asks you to input a **Backflow (回流) Total Temperature**. However, it will play a role only if there is backflow. There is **no information provided by Fluent Help File** about what is the actual boundary condition for heat transfer.



**The problem has been asked by many users.**

Someone indicate online that the actual value of temperature is calculated using **the value of last time step**, or by **interpolating methods** from values of neighboring nodes.



## Pressure in Fluent

**Atmospheric pressure (大气压)**

**Gauge pressure (表压):** the difference between the true pressure and the Atmospheric pressure.

**Absolute pressure (真实压力):** the true pressure  
**= Atmospheric pressure + Gauge pressure**

**Operating pressure (操作压力) :** the reference pressure (参考压力)

In our teaching code, a reference pressure point is defined.

## Pressure in Fluent

**Absolute pressure (真实压力):** the true pressure

**= Reference Pressure + Relative Pressure**

**Static pressure (静压):** the difference between true pressure and operating pressure.

**The same as relative pressure.**

**Dynamic pressure (动压):** calculated by  $0.5\rho U^2$

**Is related to the velocity.**

**Total pressure (总压):**

**= Static pressure + dynamic pressure**

## 13.5 Flow and heat transfer in chip cooling (4-19)

### 芯片冷却流动换热问题

**Focus:** compared with previous examples, this example is a relatively realistic problem. The domain of this Example contains fluid, board (电路板) and chip (芯片) .

**Note:** for steps similar to previous examples, you can refer to previous slides for details.

## 13.5 Flow and heat transfer in chip cooling

**Known:** Steady laminar flow and convective heat transfer around a board on top of which is a chip with source term. The domain and size is shown in **Fig. 1**. The boundary conditions are as follows:

- Inlet:  $u=0.5\text{m/s}$  (constant)

$$T=298\text{K}$$

- Pressure outlet: Gauge pressure (表压) : 0 Pa.

- Top and bottom boundary: 3<sup>rd</sup> boundary condition

Heat transfer coefficient:  $H=1.5\text{ W}/(\text{m}^2\text{K})$ ;

Free stream temperature:  $T_f=298\text{K}$ .

- Chip-- a constant source term,  $904055 \text{ W/m}^3$
- Front surface and back surface---symmetry

Pressure outlet

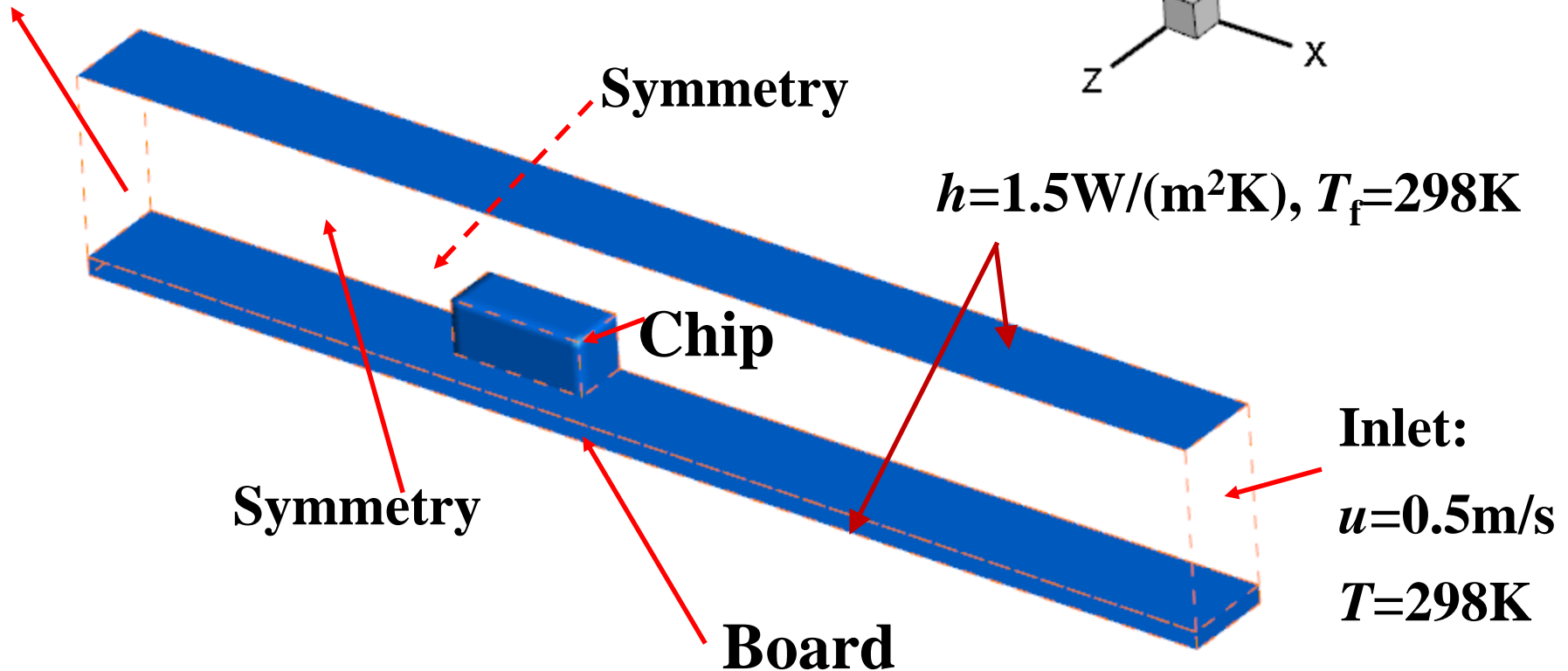
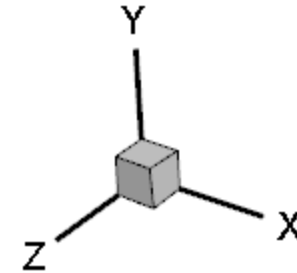


Fig.1 Computational domain

**Find:** Temperature distribution in the domain.

**Solution:**

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial x} = 0$$

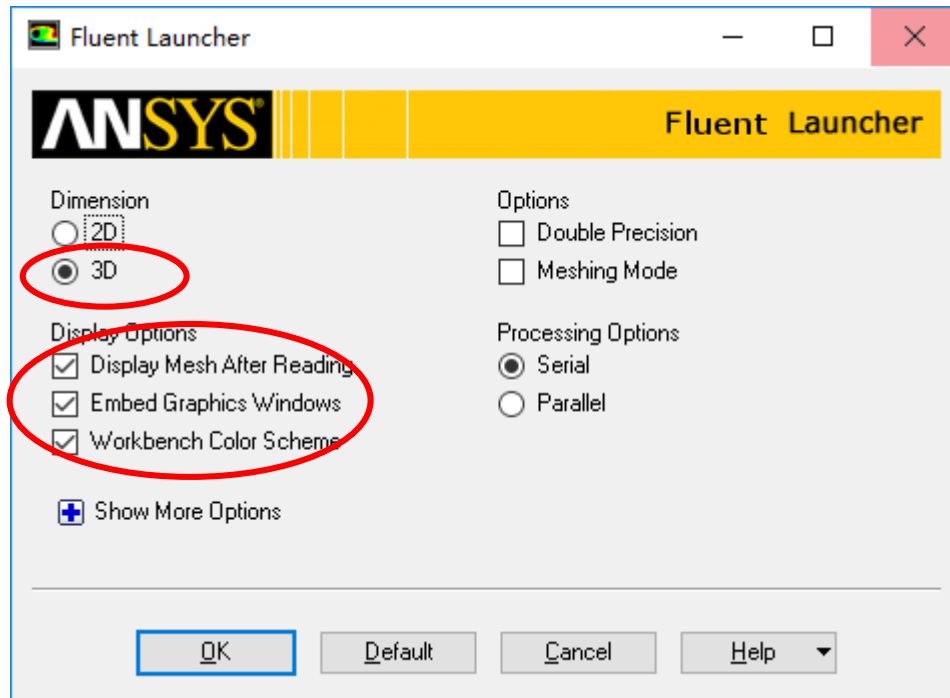
$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho_f} \frac{\partial p}{\partial x} + \frac{\mu_f}{\rho_f} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho_f} \frac{\partial p}{\partial y} + \frac{\mu_f}{\rho_f} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)$$

$$\frac{\partial(\rho_f C_{pf} u_f T_f)}{\partial x} + \frac{\partial(\rho_f C_{pf} v_f T_f)}{\partial y} = \lambda_f \left( \frac{\partial^2 T_f}{\partial x^2} + \frac{\partial^2 T_f}{\partial y^2} \right)$$

$$0 = \lambda_s \left( \frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} \right) + s$$

## 13.5.1 Start the Fluent software

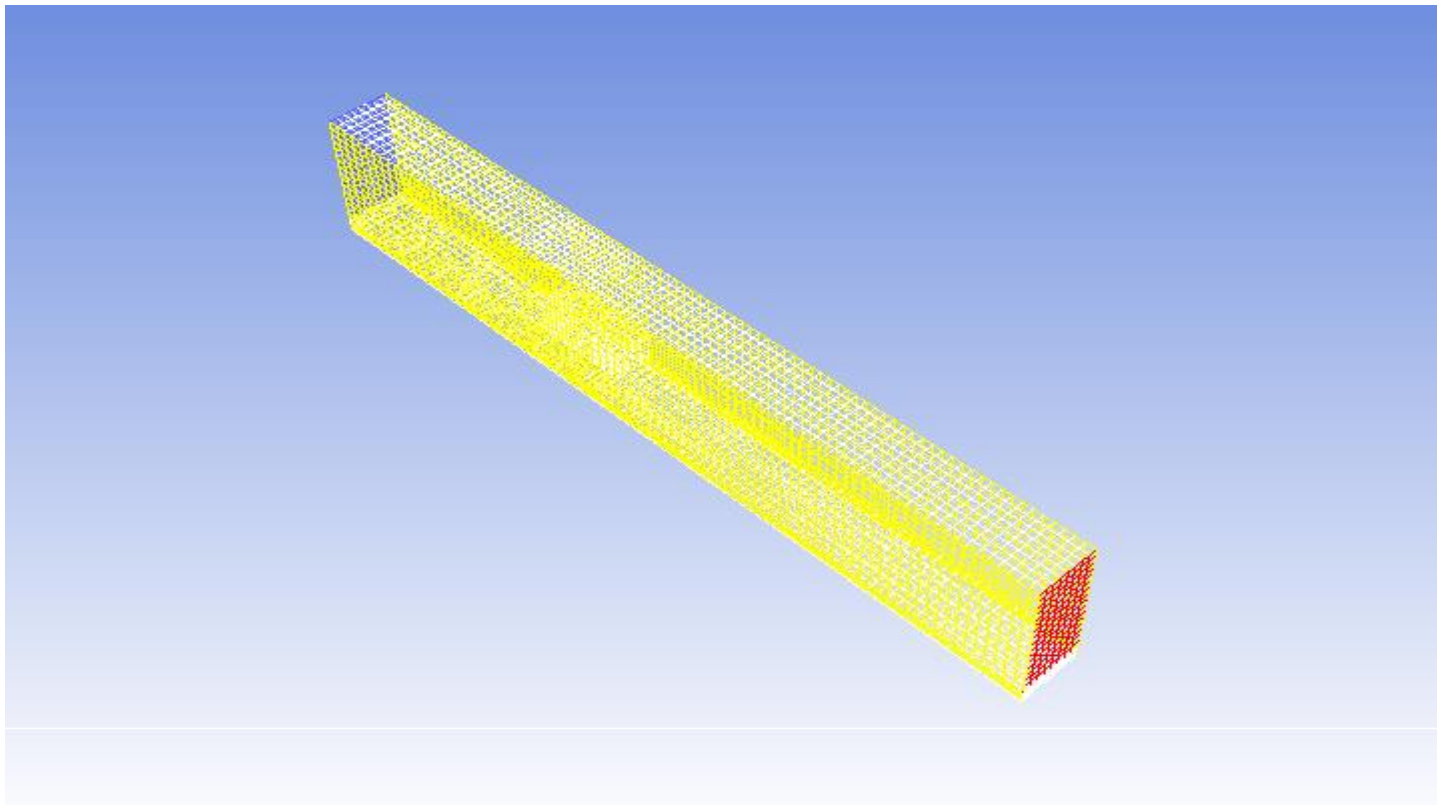


1. Choose **3-Dimension**
2. Choose **display options**
3. Choose **Serial processing option**



## 1st step: **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) “**xx.msh**”



# 1st step: Read and **check** the mesh

## Mesh→Check

- Check the **quality and topological information** of the mesh

### Mesh Check

#### Domain Extents:

x-coordinate: min (m) = 0.000000e+00, max (m) = 1.651000e-01

y-coordinate: min (m) = 0.000000e+00, max (m) = 2.794000e-02

z-coordinate: min (m) = -2.540000e-07, max (m) = 1.270000e-02

#### Volume statistics:

minimum volume (m3): 1.119834e-09

maximum volume (m3): 7.845747e-09

total volume (m3): 5.858386e-05

#### Face area statistics:

minimum face area (m2): 8.370037e-07

maximum face area (m2): 4.194085e-06

Checking mesh.....

Done.

## 2st step: Scale the domain size

General → Scale

## 3st step: Choose the physicochemical model

*Re* number is calculated to determine the fluid state (laminar or turbulent)

$$Re = \frac{\rho u l}{\mu}$$

The density of air is 1.29Kg/m<sup>3</sup>, the inlet velocity is 0.5m/s, characteristic length is about 2cm, and kinetic viscosity of air is 1.7894E-05. *Re* is 720 and thus flow is **laminar**.

### Models

Models

- Multiphase - Off
- Energy - Off**
- Viscous - Laminar
- Radiation - Off
- Heat Exchanger - Off
- Species - Off
- Discrete Phase - Off
- Solidification & Melting - Off
- Acoustics - Off
- Eulerian Wall

#### Energy

Energy

Energy Equation

OK Cancel Help

Edit...

Help

### Viscous Model

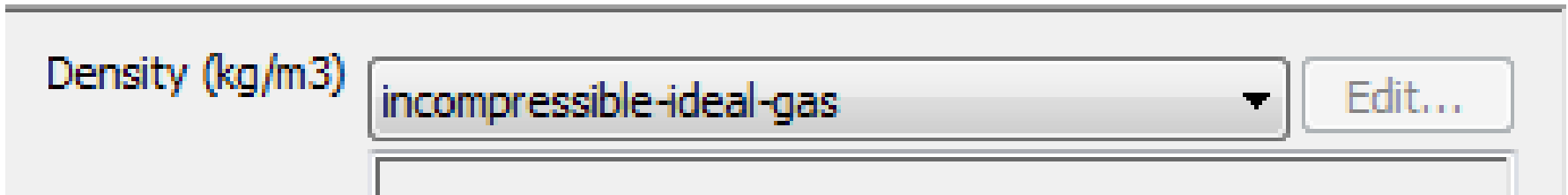
Model

- Inviscid
- Laminar
- Spalart-Allmaras (1 eqn)
- k-epsilon (2 eqn)
- k-omega (2 eqn)
- Transition k-k-omega (3 eqn)
- Transition SST (4 eqn)
- Reynolds Stress (7 eqn)
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation (DES)
- Large Eddy Simulation (LES)

OK Cancel Help

## Step 4: Define the material properties

If you calculate the density using the **ideal gas law**, the solver will compute the density according to **ideal gas state equation**.



Density (kg/m<sup>3</sup>)  Edit...

### Define a new material as Chip:

density 1000 kg/m<sup>3</sup>, Cp 500 J/(kg K) and thermal conductivity 1 W/(mK)

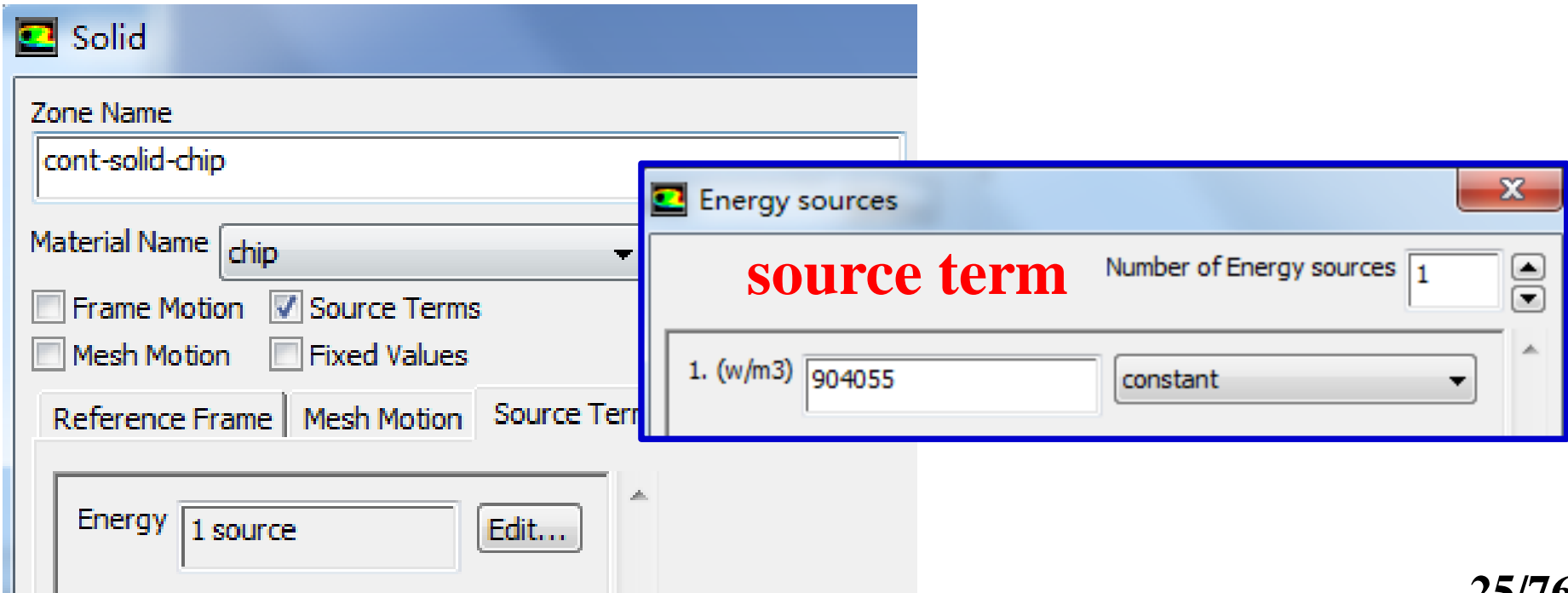
### Define a new material as Board:

density 2000 kg/m<sup>3</sup>, Cp 600 J/(kg K) and thermal conductivity 0.1 W/(mK)

## Step 5: Define zone condition

Assign different regions with the corresponding materials.

For the chip, there is a source term with value of  $904055 \text{ W/m}^3$



The image shows a screenshot of the ANSYS Fluent software interface. The 'Solid' dialog box is open, showing the 'Zone Name' field set to 'cont-solid-chip' and the 'Material Name' dropdown set to 'chip'. The 'Source Terms' checkbox is checked. The 'Energy sources' dialog box is also open, showing a 'source term' in red text. The 'Number of Energy sources' is set to 1. The first source term is defined with a value of 904055 (w/m3) and a type of 'constant'.

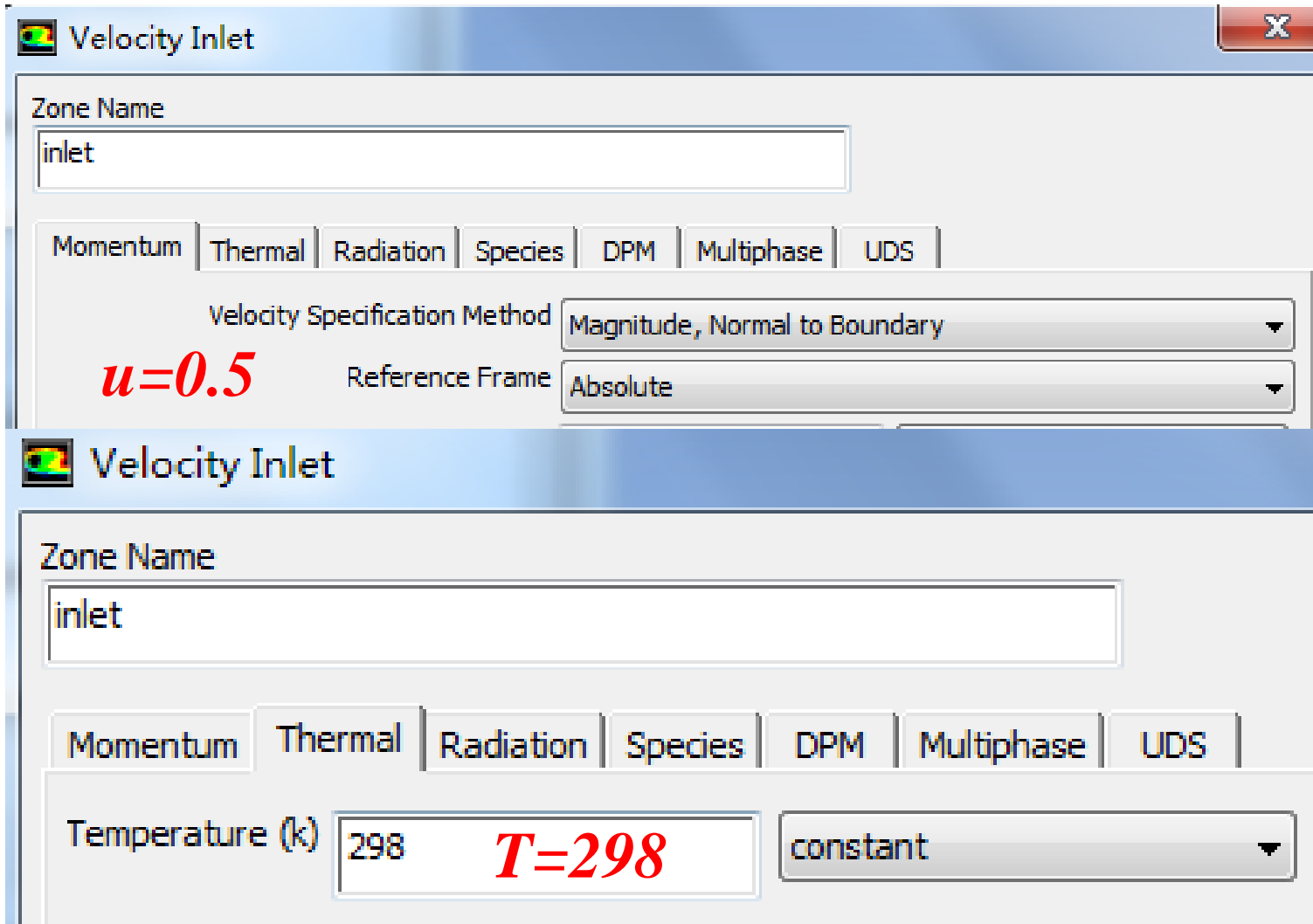
**source term**

Zone Name: cont-solid-chip  
Material Name: chip  
 Frame Motion  Source Terms  
 Mesh Motion  Fixed Values  
Reference Frame | Mesh Motion | Source Term  
Energy: 1 source [Edit...]

Energy sources: 1. (w/m3) 904055 constant

## Step 6: Define the boundary condition

Inlet:  $u$  and  $T$  are specified.



The image shows two screenshots of the ANSYS Fluent Velocity Inlet dialog box. The top screenshot shows the Momentum tab selected, with the Velocity Specification Method set to 'Magnitude, Normal to Boundary' and the Reference Frame set to 'Absolute'. A red annotation  $u=0.5$  is placed next to the Velocity Specification Method dropdown. The bottom screenshot shows the Thermal tab selected, with the Temperature (k) field set to 298 and the Temperature Specification Method set to 'constant'. A red annotation  $T=298$  is placed next to the Temperature (k) field.

**Velocity Inlet**

Zone Name: inlet

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

$u=0.5$

---

**Velocity Inlet**

Zone Name: inlet

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

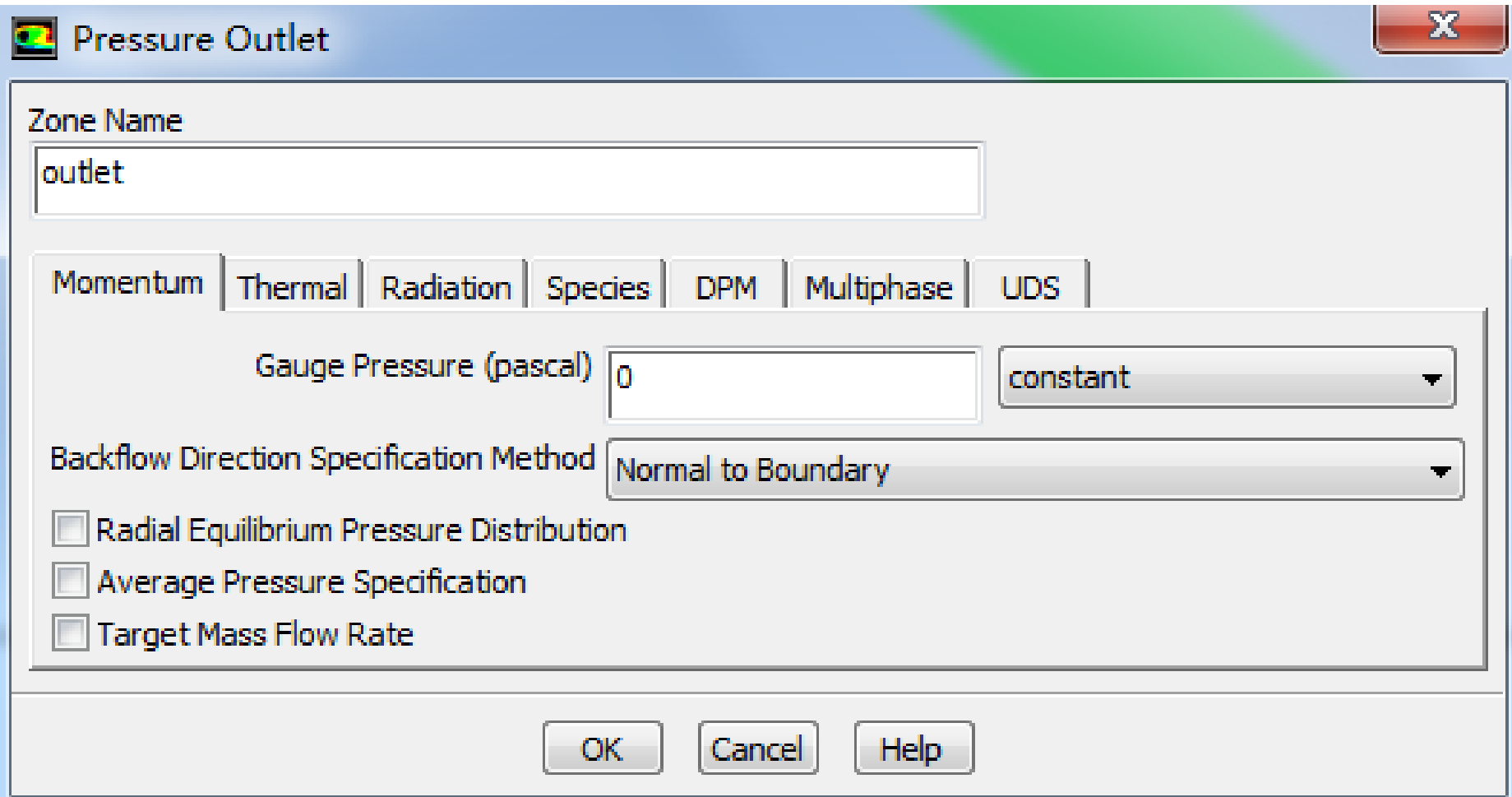
Temperature (k): 298

Temperature Specification Method: constant

$T=298$

## Step 6: Define the boundary condition

**Outlet: pressure outlet, Gauge pressure as 0.**



Pressure Outlet

Zone Name  
outlet

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

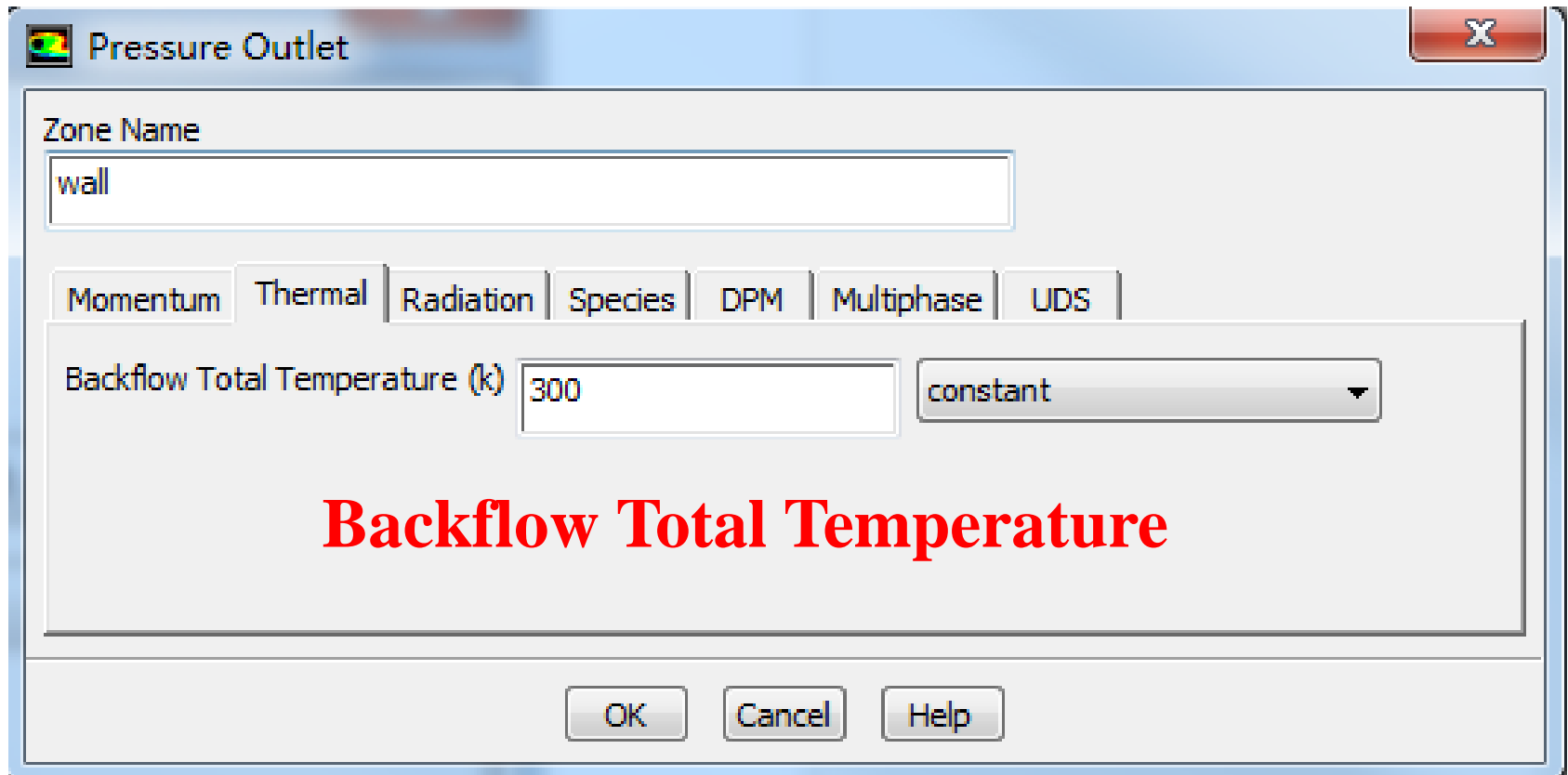
Gauge Pressure (pascal) 0 constant

Backflow Direction Specification Method Normal to Boundary

Radial Equilibrium Pressure Distribution  
 Average Pressure Specification  
 Target Mass Flow Rate

OK Cancel Help

For pressure outlet boundary condition, Fluent asks you to input a **Backflow (回流) Total Temperature**.



# Step 6: Define the boundary condition

## Top and bottom wall: convective boundary condition

The screenshot shows the 'Wall' dialog box in ANSYS Fluent. The 'Zone Name' is 'wall-board-bottom' and the 'Adjacent Cell Zone' is 'cont-solid-board'. The 'Thermal' tab is selected. Under 'Thermal Conditions', 'Convection' is selected. The 'Heat Transfer Coefficient (w/m2-k)' is set to 1.5 with a 'constant' dropdown. The 'Free Stream Temperature (k)' is set to 298 with a 'constant' dropdown. The 'Wall Thickness (in)' is set to 0. The 'Heat Generation Rate (w/m3)' is set to 0 with a 'constant' dropdown. The 'Material Name' is 'aluminum'. There are 'OK', 'Cancel', and 'Help' buttons at the bottom.

Zone Name  
wall-board-bottom

Adjacent Cell Zone  
cont-solid-board

Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film

Thermal Conditions

- Heat Flux
- Temperature
- Convection
- Radiation
- Mixed
- via System Coupling

Heat Transfer Coefficient (w/m2-k) 1.5 constant

Free Stream Temperature (k) 298 constant

Wall Thickness (in) 0 P

Heat Generation Rate (w/m3) 0 constant

Material Name aluminum Edit...

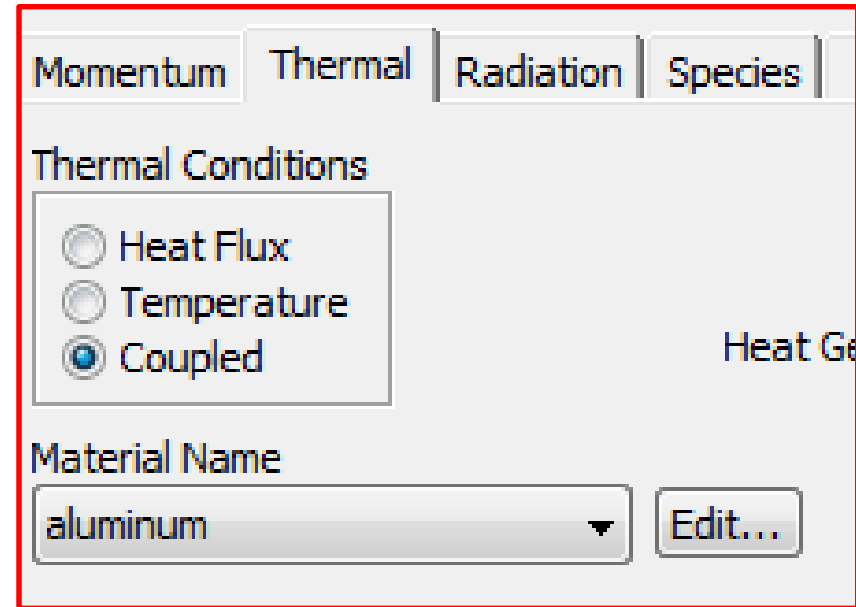
Shell Conduction Define...

OK Cancel Help

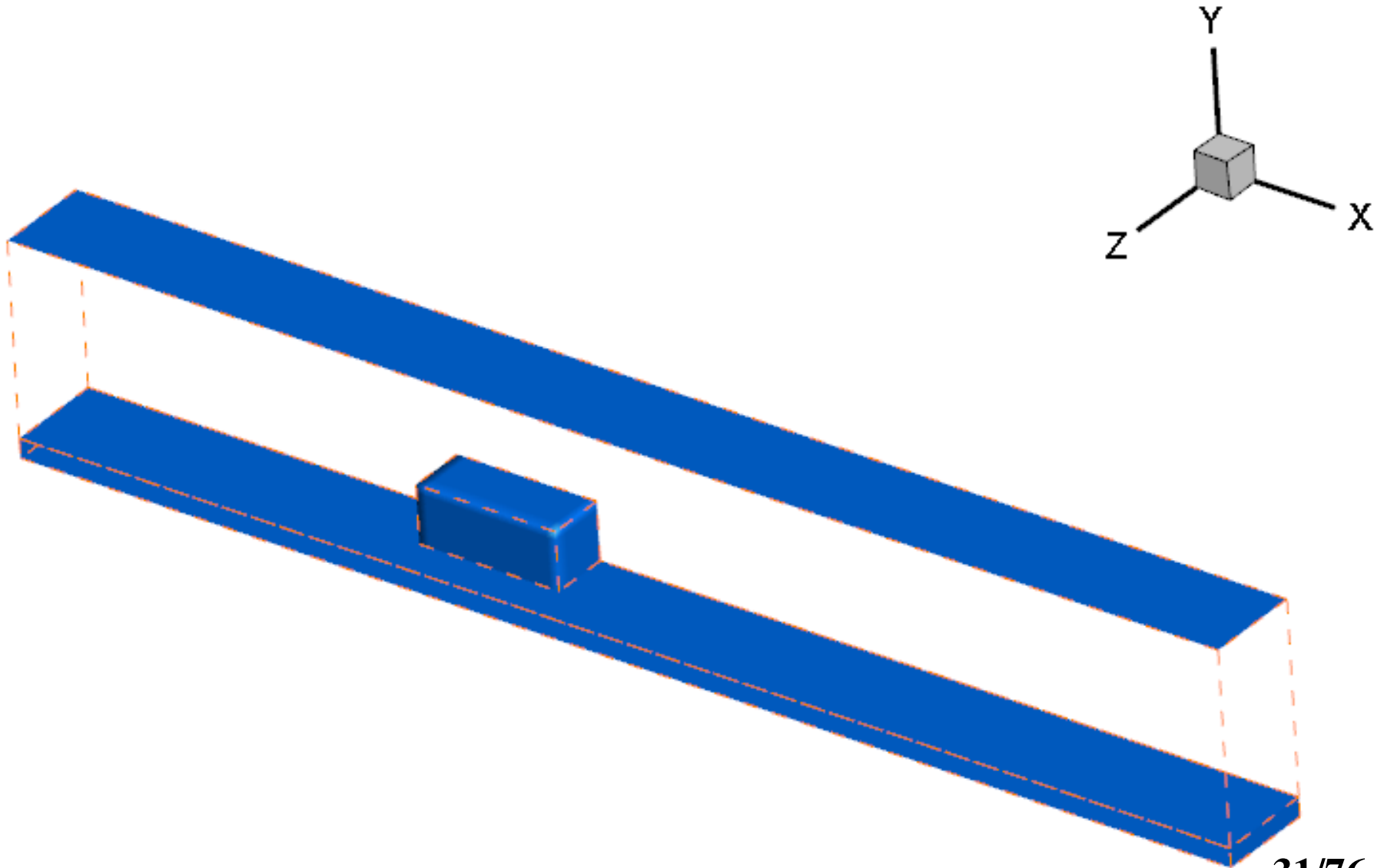
## Step 6: Define the boundary condition

For the front and back boundaries, keep the default set up of **Symmetry**.

For all the other “two-side-walls” boundaries in the domain, keep the default set up for thermal conditions, namely “**Coupled**”. *For details of “Coupled” and “uncoupled” conditions, refer to Example 4 in Chapter 13.*

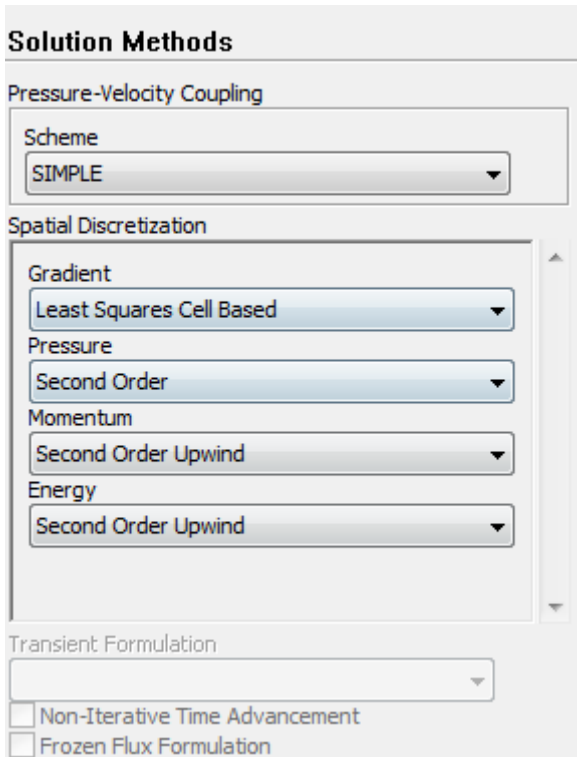


There are many **two-sided-wall** in this Example.



## 7st step: Define the solution

For algorithm and schemes, keep it as default. For more details of this step, **one can refer to Example 1** of Chapter 13.



**Algorithm:** simple

**Gradient:** Least Square Cell Based

**Pressure:** second order

**Momentum:** second order upwind

**Energy:** second order Upwind

## 7st step: **Define the solution**

For under-relaxation factor, keep it default. For more details, refer to **Example 1**.

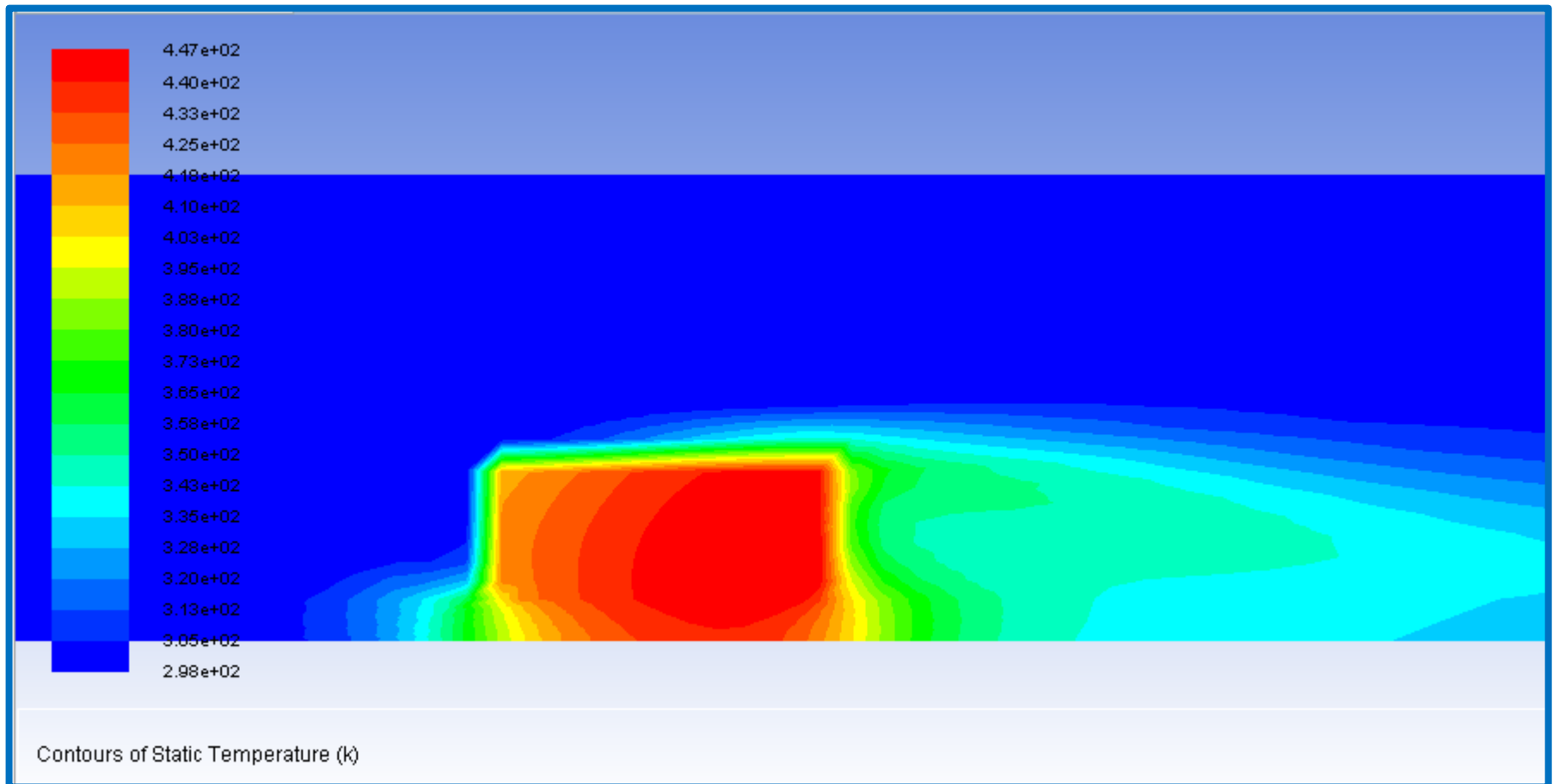
## 8st step: Initialization

Use the standard initialization, for more details of Hybrid initialization, refer to Example 1.

## Step 9: Run the simulation

## Step 10: Post-processing results

# Static Temperature(K) of back boundary



## 13.6 Flow and heat transfer in porous media

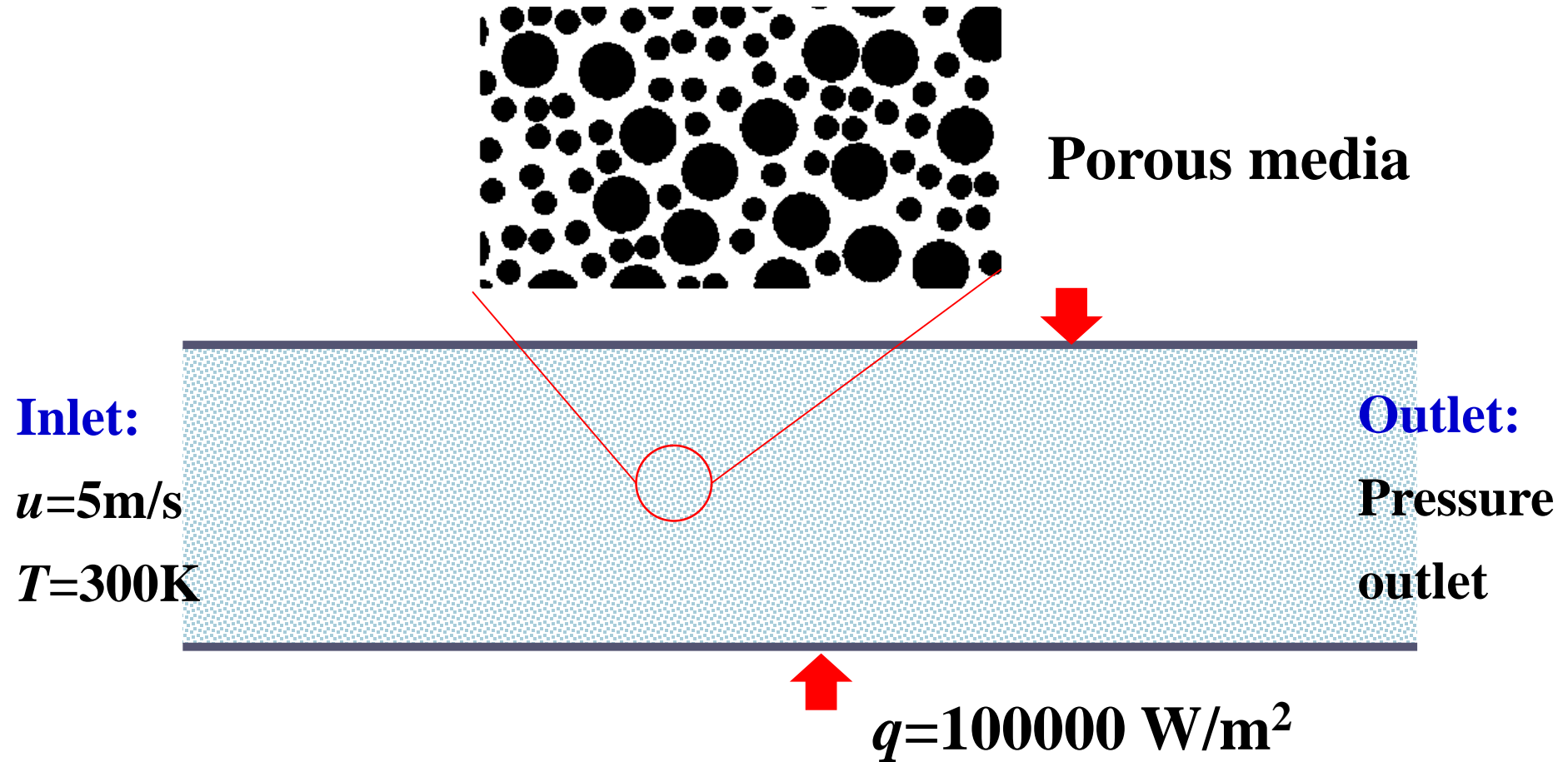
### 多孔介质流动换热问题

**Focus:** in Example 6, the **background of porous media** is introduced, and fluid flow and heat transfer in porous media is discussed in detail.

## 13.6 Flow and heat transfer in porous media

**Known:** Steady state fluid flow and heat transfer of air in a channel filled with porous medium made of Aluminum (铝). The porosity (孔隙率) of the porous medium is 0.8. The permeability (渗透率) of the porous medium is  $7.11\text{E-}8 \text{ m}^2$ . The computational domain is shown in Fig. 1. The boundary condition is as follows.

- Inlet:  $u=5\text{m/s}$  ;  $T=300\text{K}$
- Pressure outlet: Gauge pressure (表压) : 0 Pa.
- Top and bottom boundary: 2<sup>rd</sup> boundary condition  
Heat flux:  $q=100000 \text{ W/m}^2$



**Fig.1 Computational domain**

**Find:** Temperature and velocity distribution in the domain

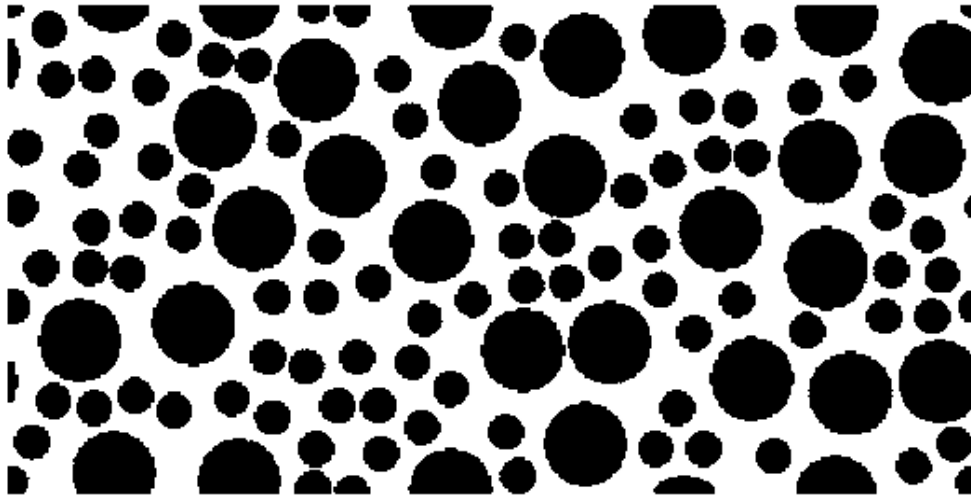
**Solution:**

Continuity, momentum and energy equation for  
porous media????

The governing equations for porous media are quite different from the original equation. Thus, we will study together background information of porous media and then derive the governing equations.

## Porous media

A porous medium is a material that contains plenty of pores (孔) between solid skeleton (骨架) .



**Black: solid**

**White: pores**

Two necessary elements in a porous medium:

**Skeleton** : maintaining the shape of a porous medium

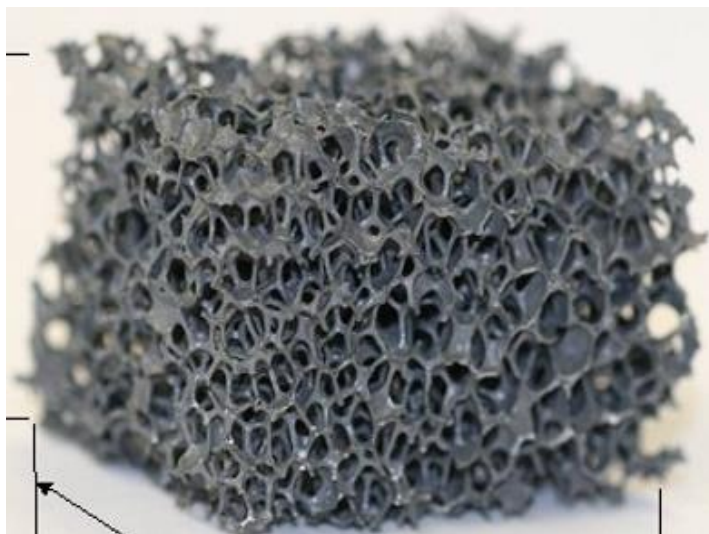
**Pores**: providing pathway for fluid transport.



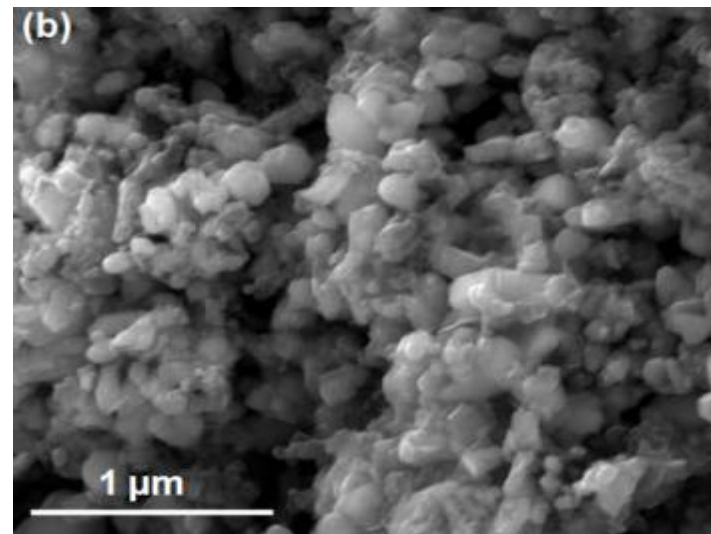
**Stone**



**Carbon fiber**



**Metal foam**



**Catalyst**

# Porosity

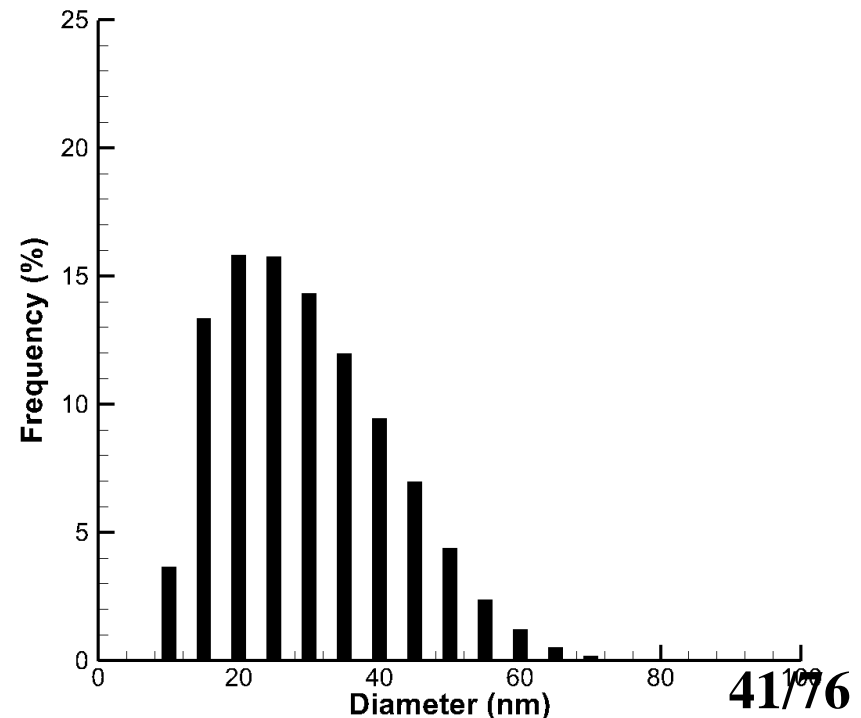
The volume ratio between pore volume and total volume

$$\varepsilon = \frac{V_{\text{pore}}}{V_{\text{total}}}$$

In the range of 0~1.

## Pore size

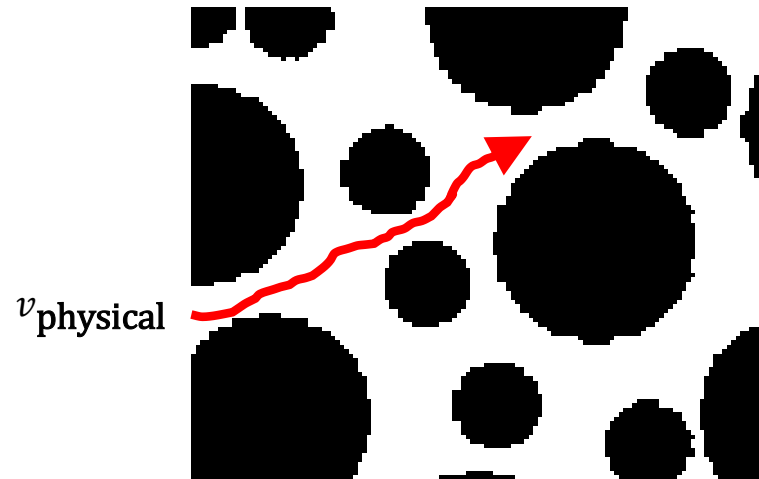
The size of pores. Use **pore size distribution** to characterize (表征) size of pores of a porous medium.



## Two velocity definition in a porous medium:

$$v_{\text{superficial}} = \varepsilon v_{\text{physical}}$$

**Porosity**



$V_{\text{physical}}$ : the actual flow velocity in the pores.

$V_{\text{superficial}}$  (表观速度): the averaged velocity in the entire domain.

$$V_{\text{superficial}} < V_{\text{physical}}$$

**Fluent uses superficial velocity as the default velocity.**

## Original continuity and momentum equation

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

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + (\mathbf{u} \cdot \nabla)(\rho \mathbf{u}) = -\nabla p + \eta \nabla^2 \mathbf{u}$$

### Continuity equation for porous media:

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

As the total mass of fluid is  $\rho V_f = \rho \varepsilon V_{\text{total}} = \rho \varepsilon \Delta x \Delta y \Delta z$

Fluent uses superficial velocity as the default velocity.

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

## Momentum equation for porous media:

$$\frac{\partial(\varepsilon \rho \mathbf{u}_{\text{physical}})}{\partial t} + (\mathbf{u}_{\text{physical}} \cdot \nabla)(\varepsilon \rho \mathbf{u}_{\text{physical}}) = -\varepsilon \nabla(p) + \eta \varepsilon \nabla^2 \mathbf{u}_{\text{physical}} + \mathbf{F}$$



**Total force due to porous media**

$$\frac{\partial(\rho \mathbf{u}_{\text{superficial}})}{\partial t} + \left( \frac{\mathbf{u}_{\text{superficial}}}{\varepsilon} \cdot \nabla \right) (\rho \mathbf{u}_{\text{superficial}}) = -\varepsilon \nabla(p) + \varepsilon \eta \nabla^2 \left( \frac{\mathbf{u}_{\text{superficial}}}{\varepsilon} \right) + \mathbf{F}$$

For incompressible steady state problem:

$$\nabla \cdot \mathbf{u}_{\text{superficial}} = 0$$

$$\left( \frac{\mathbf{u}_{\text{superficial}}}{\varepsilon} \cdot \nabla \right) (\mathbf{u}_{\text{superficial}}) = -\frac{1}{\rho} \varepsilon \nabla(p) + \eta \nabla^2 (\mathbf{u}_{\text{superficial}}) + \mathbf{F}$$

**The fluid-solid interaction is strong in porous media. Porous media are modeled by adding a momentum source term:**

$$\mathbf{F} = -\frac{\varepsilon \nu}{k} \mathbf{u} - \frac{\varepsilon F_\varepsilon}{\sqrt{k}} |\mathbf{u}| \mathbf{u}$$

The first term is the **viscous loss term** (黏性项) or the **Darcy term**.

The second term is **inertial loss term** (惯性项) or the **Forchheimer term**.

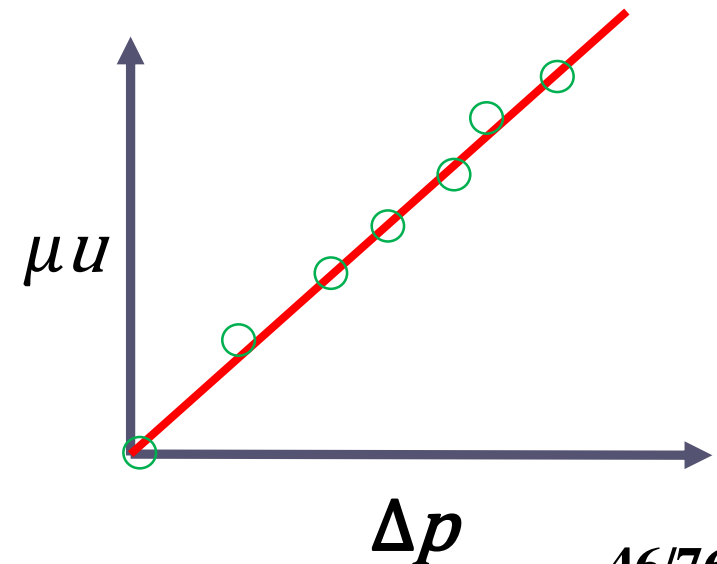
$k$  is the **permeability** (渗透率) of a porous media, one of the most important parameter of a porous media

## Permeability (渗透率)

In 1856, Darcy (法国工程师) noted that for laminar flow through porous media, the flow rate  $\langle u \rangle$  is linearly proportional to the applied pressure gradient  $\Delta p$ , thus he introduced **permeability to describe the conductivity of the porous media**. The Darcy' law is as follows

$$\langle u \rangle = - \frac{k}{\mu} \frac{\Delta p}{l}$$

**$k$  is permeability** with unit of  $\text{m}^2$



In Fluent, this force source term is expressed as

$$\mathbf{F} = -\frac{\mu}{k}\mathbf{u} - C_2 \frac{1}{2} \rho |\mathbf{u}| \mathbf{u}$$

**$k$ : permeability;  $C_2$ : inertial resistance factor**

Viscous Resistance

Direction-1 (1/m<sup>2</sup>)

constant

Direction-2 (1/m<sup>2</sup>)

constant

**Here, viscous resistance is  $1/k$ ! Permeability is a transport property of a porous medium, and there are database of  $k$  for different porous materials.**

The second term can be canceled if the fluid flow is slow

$$\mathbf{F} = -\frac{\mu}{k}\mathbf{u} - C_2 \frac{1}{2} \phi |\mathbf{u}| \mathbf{u}$$

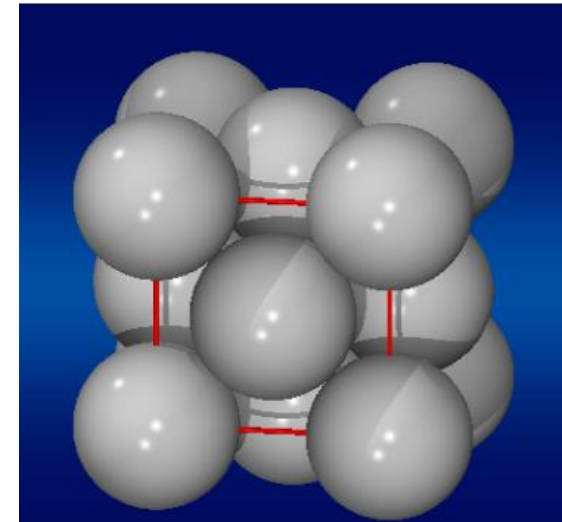
$\mathbf{u}$  is small, thus  $\mathbf{u}*\mathbf{u}$  is smaller.

Otherwise, this term should be considered.

There have been lots of experiments in the literature to determine the relationship between pressure drop and velocity of different kinds of porous media, and thus to determine permeability.

**Ergun equation** is one of the most adopted empirical equations (经验公式) for packed bed porous media.

$$\frac{\Delta P}{l} = \frac{150\mu (1-\varepsilon)^2}{\underline{D_p^2} \varepsilon^3} u + \frac{1.75\rho (1-\varepsilon)}{D_p \varepsilon^3} u^2$$



**Diameter of solid particle**

$$\mathbf{F} = -\frac{\mu}{k} \mathbf{u} - C_2 \frac{1}{2} \rho |\mathbf{u}| \mathbf{u}$$

Comparing the two equations, you can obtain  $C_2$ .

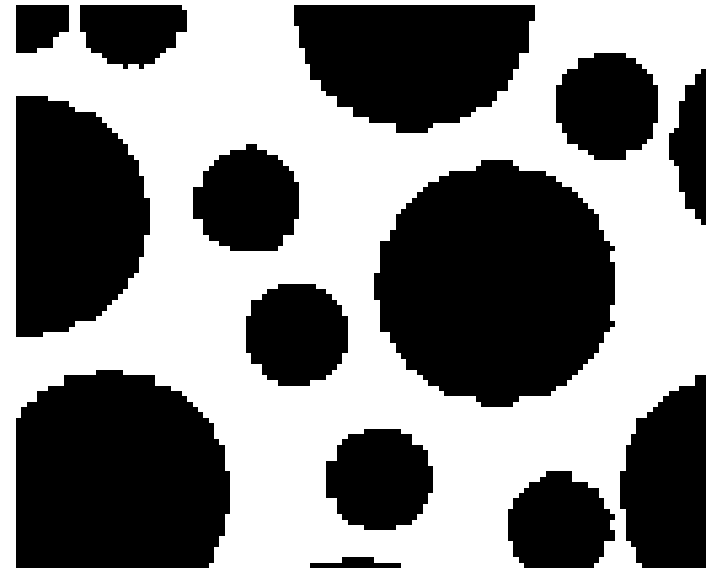
$$k = \frac{D_p^2 \varepsilon^3}{150 (1-\varepsilon)^2}$$

$$C_2 = \frac{3.5 (1-\varepsilon)}{D_p \varepsilon^3}$$

**Original energy equation:**

$$\frac{\partial(\rho C_p T)}{\partial t} + (\mathbf{u} \cdot \nabla)(\rho C_p T) = \lambda \nabla^2 T + S$$

**For porous media:**



**Heat transfer in fluid phase as well as in solid phase.**

**There are two models for heat transfer:**

**Equilibrium thermal model**

**Non-Equilibrium thermal model**

## Equilibrium thermal model

Assume solid phase and fluid phase are in thermal equilibrium. At the fluid-solid phase, temperature and heat flux are continuous.

**Original**

$$\frac{\partial(\rho C_p T)}{\partial t} + (\mathbf{u} \cdot \nabla)(\rho C_p T) = \lambda \nabla^2 T + S$$

**For the first term:**

$$\begin{aligned} \rho C_p T V &= (1 - \varepsilon) V (\rho C_p)_{\text{solid}} T_{\text{solid}} + \varepsilon V (\rho C_p)_{\text{fluid}} T_{\text{fluid}} \\ &= \left[ (1 - \varepsilon) (\rho C_p)_{\text{solid}} + \varepsilon (\rho C_p)_{\text{fluid}} \right] V T \end{aligned}$$

$$\rho C_p T = \left[ (1 - \varepsilon) (\rho C_p)_{\text{solid}} + \varepsilon (\rho C_p)_{\text{fluid}} \right] T$$

**For the second term:**

$$(\mathbf{u} \cdot \nabla)(\varepsilon \rho C_p T)$$

**As convective term is only for fluid phase!**

**For the diffusion term:**

$$\begin{aligned} \lambda \nabla^2 T V &= V(1 - \varepsilon) \lambda_s \nabla^2 T_s + V \varepsilon \lambda_f \nabla^2 T_f \\ &= \left[ V(1 - \varepsilon) \lambda_s + V \varepsilon \lambda_f \right] \nabla^2 T \end{aligned}$$

$$\lambda \nabla^2 T = \left[ (1 - \varepsilon) \lambda_s + \varepsilon \lambda_f \right] \nabla^2 T$$

**For the source term**

$$SV = (1 - \varepsilon) V S_s + \varepsilon V S_f$$

$$\frac{\partial \left[ (1-\varepsilon)(\rho C_p)_{\text{solid}} + \varepsilon(\rho C_p)_{\text{fluid}} \right] T}{\partial t} + (\mathbf{u} \cdot \nabla)(\varepsilon \rho C_p T)$$

$$= \left[ (1-\varepsilon)\lambda_s + \varepsilon\lambda_f \right] \nabla^2 T + \left[ (1-\varepsilon)S_s + \varepsilon S_f \right]$$

$$(\rho C_p)_{\text{eff}} = \left[ (1-\varepsilon)(\rho C_p)_{\text{solid}} + \varepsilon(\rho C_p)_{\text{fluid}} \right]$$

$$\lambda_{\text{eff}} = (1-\varepsilon)\lambda_s + \varepsilon\lambda_f$$

$$S_{\text{eff}} = (1-\varepsilon)S_s + \varepsilon S_f$$

## The final energy equation for porous media

$$\frac{\partial \left( (\rho C_p)_{\text{eff}} T \right)}{\partial t} + (\mathbf{u}_{\text{superficial}} \cdot \nabla)(\rho C_p T) = \lambda_{\text{eff}} \nabla^2 T + S_{\text{eff}}$$

## No equilibrium thermal model

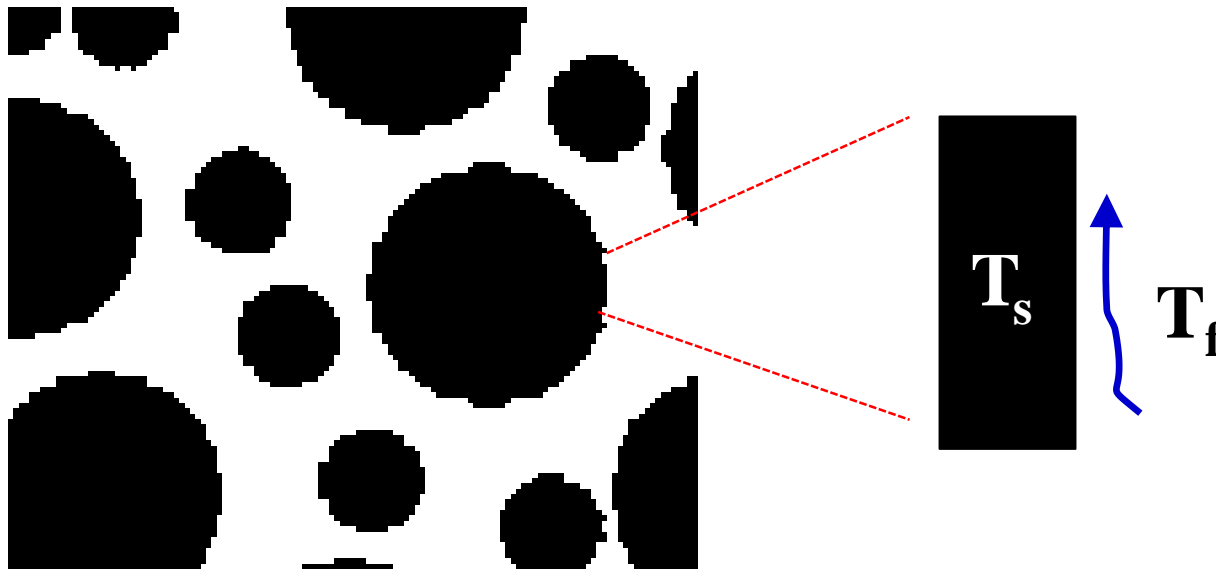
Solid phase and fluid phase are not in thermal equilibrium. The energy equation are solved for fluid and solid region respectively. At the fluid-solid phase, they are coupled **by convective boundary condition**.

### Fluid region

$$\frac{\partial([\varepsilon(\rho C_p)_{\text{fluid}}]T_f)}{\partial t} + (\mathbf{u} \cdot \nabla)(\varepsilon \rho C_p T_f) \\ = [\varepsilon \lambda_f] \nabla^2 T_f + [\varepsilon S_f] + hA(T_f - T_s)$$

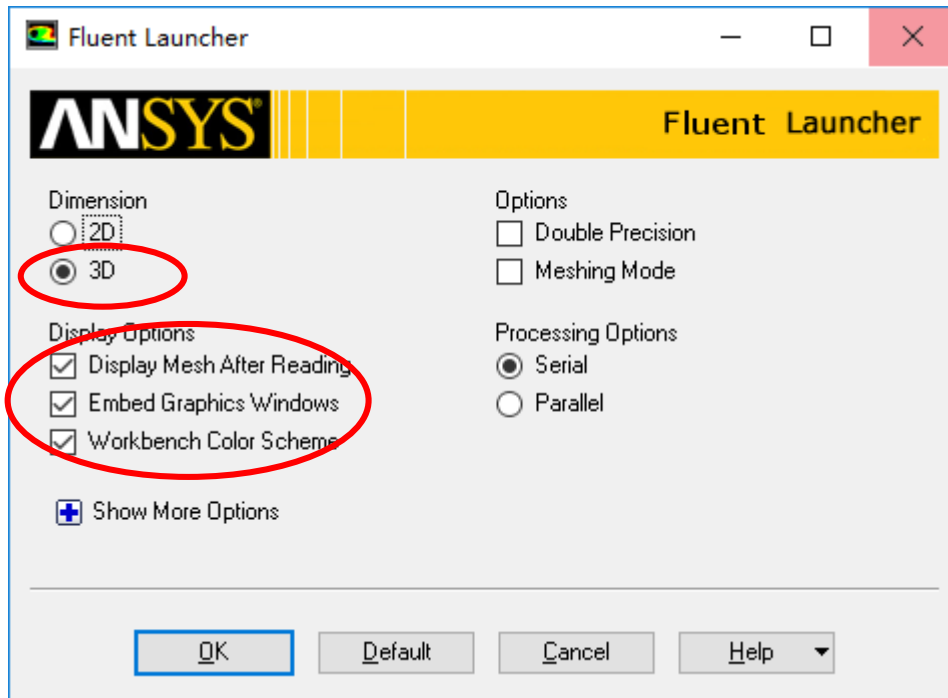
# Solid region

$$\frac{\partial \left[ (1-\varepsilon)(\rho C_p)_{\text{solid}} T \right]}{\partial t} = \left[ (1-\varepsilon)\lambda_s \right] \nabla^2 T + \left[ (1-\varepsilon)S_s \right] + \underline{hA(T_f - T_s)}$$



Two equations are solved separately, and 3<sup>rd</sup> boundary condition is adopted to couple the two equations.

# Start the Fluent software

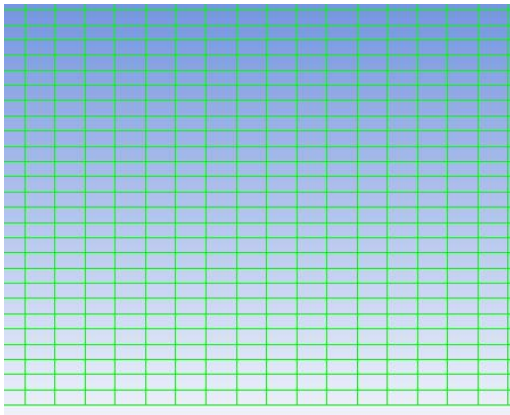


1. Choose **2-Dimension**
2. Choose **display options**
3. Choose **Serial processing option**



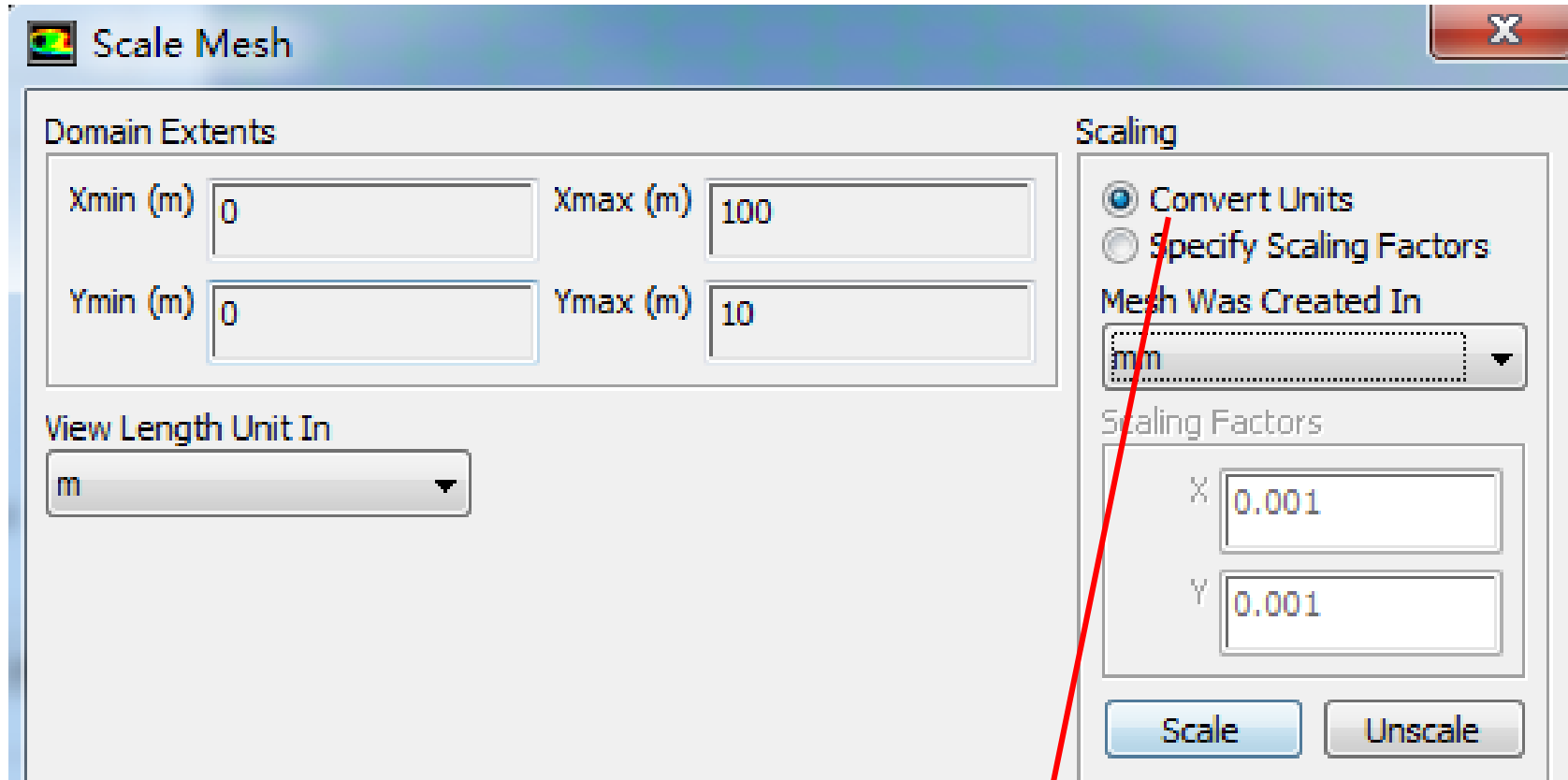
## Step 1: **Read** and check the mesh

Read the mesh and check the quality and topological information of the mesh.



```
Done.  
  
Preparing mesh for display...  
Done.  
  
Domain Extents:  
  x-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e+02  
  y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e+01  
Volume statistics:  
  minimum volume (m3): 2.024164e-02  
  maximum volume (m3): 2.024260e-02  
  total volume (m3): 1.000000e+03  
Face area statistics:  
  minimum face area (m2): 1.010094e-01  
  maximum face area (m2): 2.004013e-01  
Checking mesh.....  
Done.
```

## Step 2: Scale the domain size



The mesh is generated in Fluent using unit of **mm**. Fluent import it as unit of **m**. Thus, “**Convert units**” is used.

# Step 3: Choose the physicochemical model

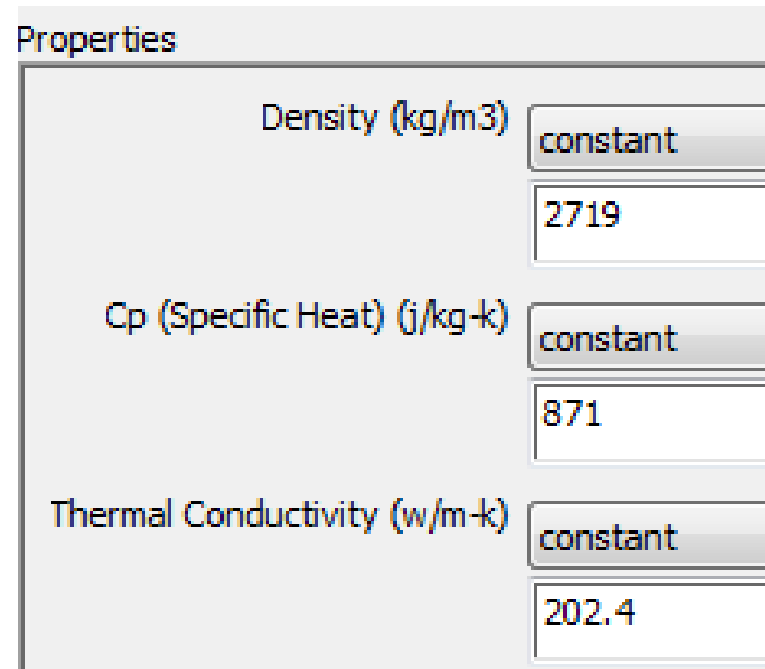
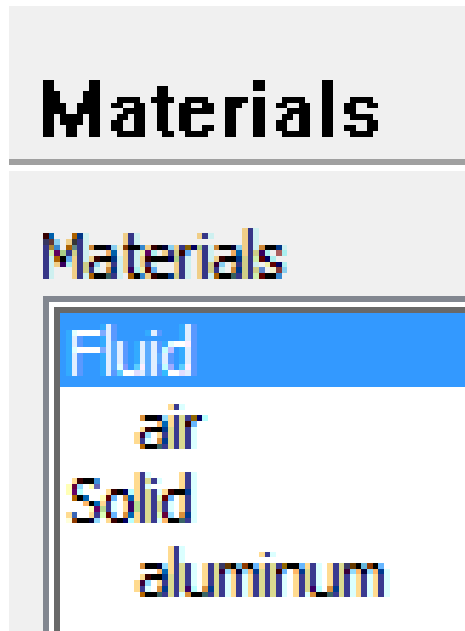
## Active fluid flow and energy model in Fluent.

The screenshot displays the Fluent software interface. On the left is a tree view with categories: Meshing, Solution Setup, Solution, and Results. Under Solution Setup, 'Models' is selected. The 'Models' panel on the right lists various models: Multiphase - Off, Energy - On (highlighted with a red box), viscous - Laminar, Radiation - Off, Heat Exchanger - Off, Species - Off, Discrete Phase - Off, Solidification & Melting - Off, and Acoustics - Off. An 'Energy' dialog box is open in the foreground, showing a checked checkbox for 'Energy Equation' and buttons for 'OK', 'Cancel', and 'Help'.

## Step 4: Define the material

Define the materials and their properties required for modeling!

In Fluent, the default fluid is **air** and the default solid is **Al**. They are the materials we will use in Example 6.

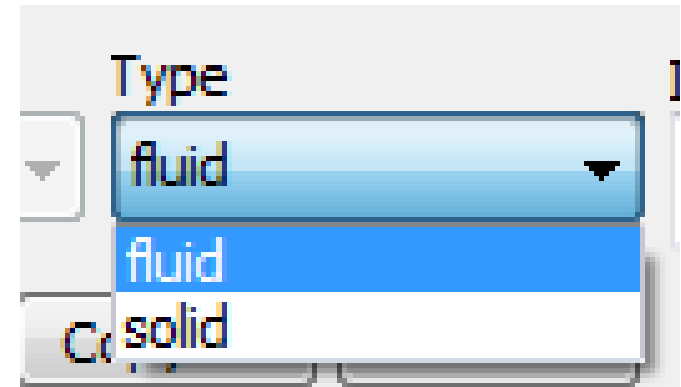


## Step 5: Define zone condition

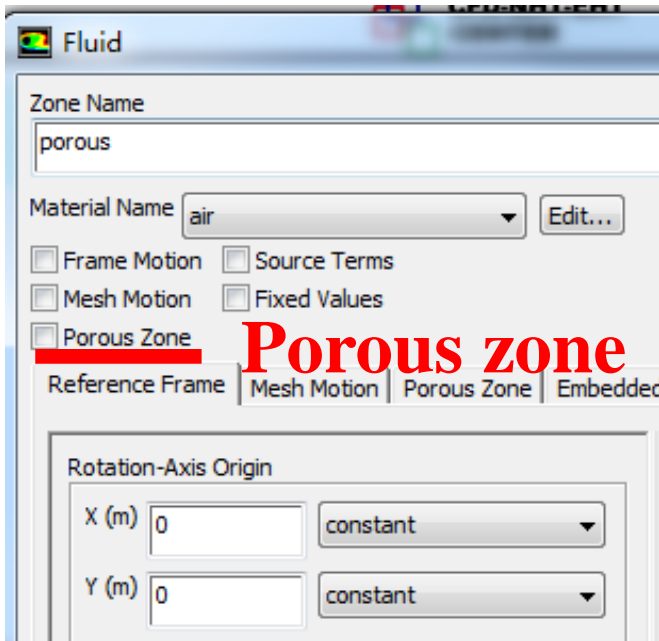
There are two options in Fluent for zone condition:

**Fluid**

**Solid**



**Porous media is treated as Fluid in Fluent.**



Here you can click “**Porous zone**” to define the porous media.

Then you can define related porosity and transport properties.

Relative Velocity Resistance Formulation

Viscous Resistance

## Viscous resistance

Direction-1 (1/m<sup>2</sup>)

0

constant

Direction-2 (1/m<sup>2</sup>)

0

constant

$\frac{1}{k}$



$$\mathbf{F} = -\frac{\mu}{k} \mathbf{u} - C_2 \frac{1}{2} \rho |\mathbf{u}| \mathbf{u}$$



$C_2$

Inertial Resistance

Alternative Formulation

## Inertial resistance

Direction-1 (1/m)

0

constant

Direction-2 (1/m)

0

constant

**KC equation is adopted, another equation obtained from experiments**

$$\frac{\Delta P}{l} = \frac{180\mu}{D_p^2} \frac{(1-\varepsilon)^2}{\varepsilon^3} u$$

$$\mathbf{F} = -\frac{\mu}{k} \mathbf{u} - C_2 \frac{1}{2} \rho |\mathbf{u}| \mathbf{u}$$

$$k = \frac{D_p^2}{180} \frac{\varepsilon^3}{(1-\varepsilon)^2}$$

$$C_2 = 0$$

$$D_p = 1\text{mm}$$

$$\varepsilon = 0.8$$

$$k = 7.11\text{E-}8, 1/k = 1.4\text{E}7$$

$$C_2 = 0$$

# Porosity

Fluid Porosity

Porosity

constant

$k=7.11E-8, 1/k=1.4E7$

Relative Velocity Resistance Formulation

Viscous Resistance

Direction-1 (1/m<sup>2</sup>)

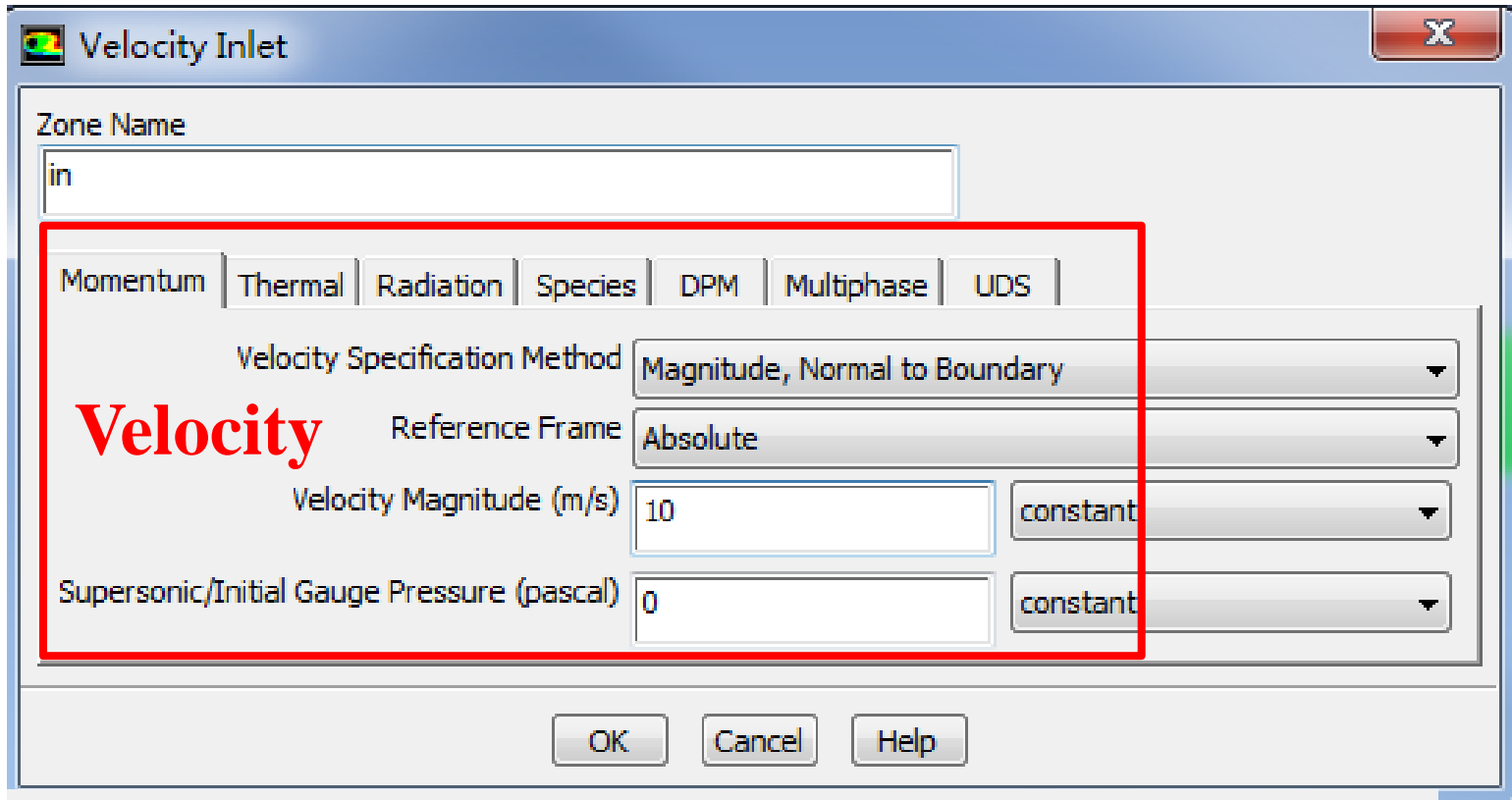
constant

Direction-2 (1/m<sup>2</sup>)

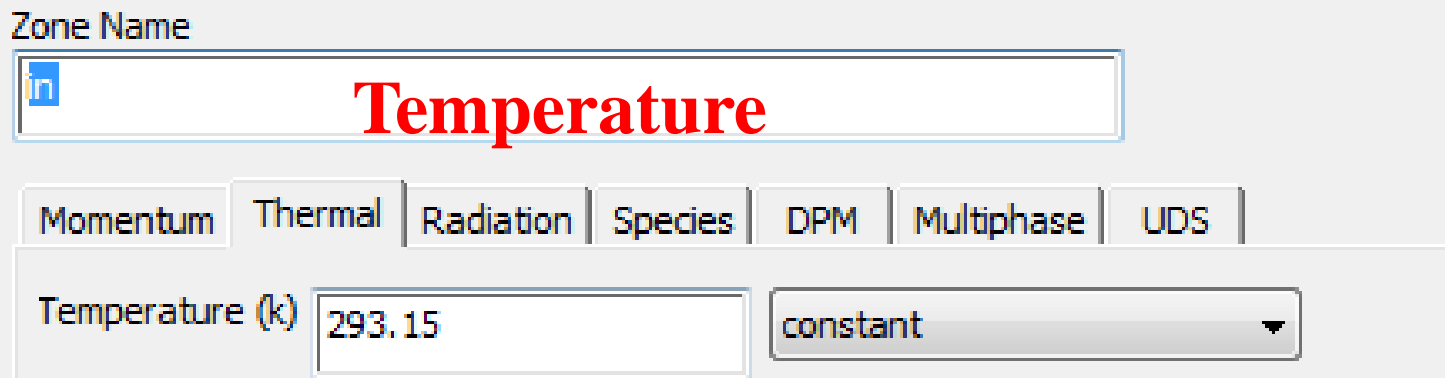
constant

# Step 6: Define the boundary condition

**Inlet**



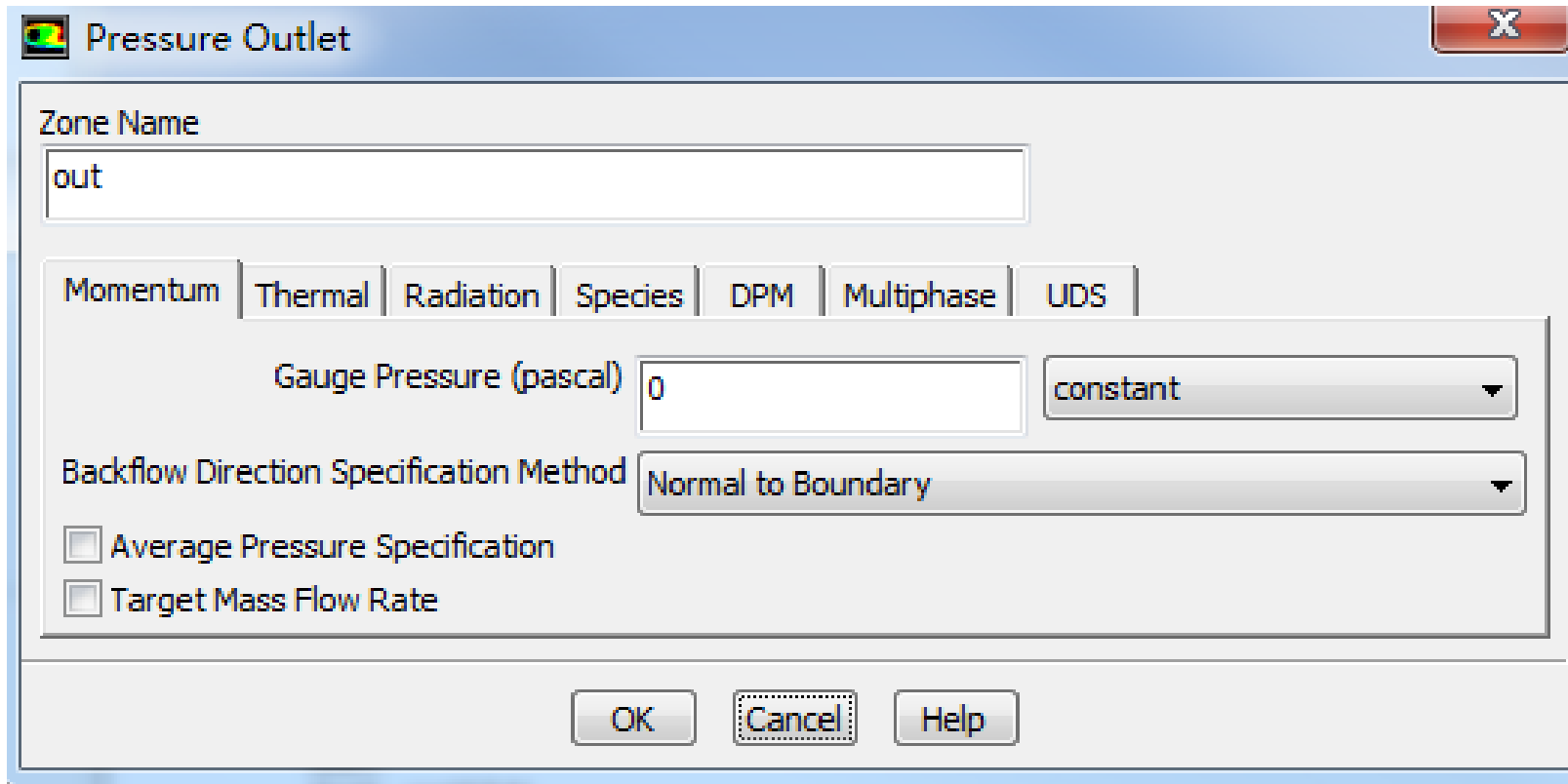
The image shows a software dialog box titled "Velocity Inlet". It has a "Zone Name" field containing "in". Below this are several tabs: "Momentum", "Thermal", "Radiation", "Species", "DPM", "Multiphase", and "UDS". The "Momentum" tab is selected. Under this tab, there are three main settings: "Velocity Specification Method" set to "Magnitude, Normal to Boundary", "Reference Frame" set to "Absolute", and "Velocity Magnitude (m/s)" set to "10" with a "constant" dropdown. Below these is "Supersonic/Initial Gauge Pressure (pascal)" set to "0" with a "constant" dropdown. At the bottom are "OK", "Cancel", and "Help" buttons. A red rectangular box highlights the "Velocity Specification Method", "Reference Frame", "Velocity Magnitude", and "Supersonic/Initial Gauge Pressure" sections.



The image shows a software dialog box titled "Temperature". It has a "Zone Name" field containing "in". Below this are several tabs: "Momentum", "Thermal", "Radiation", "Species", "DPM", "Multiphase", and "UDS". The "Thermal" tab is selected. Under this tab, there is one main setting: "Temperature (k)" set to "293.15" with a "constant" dropdown.

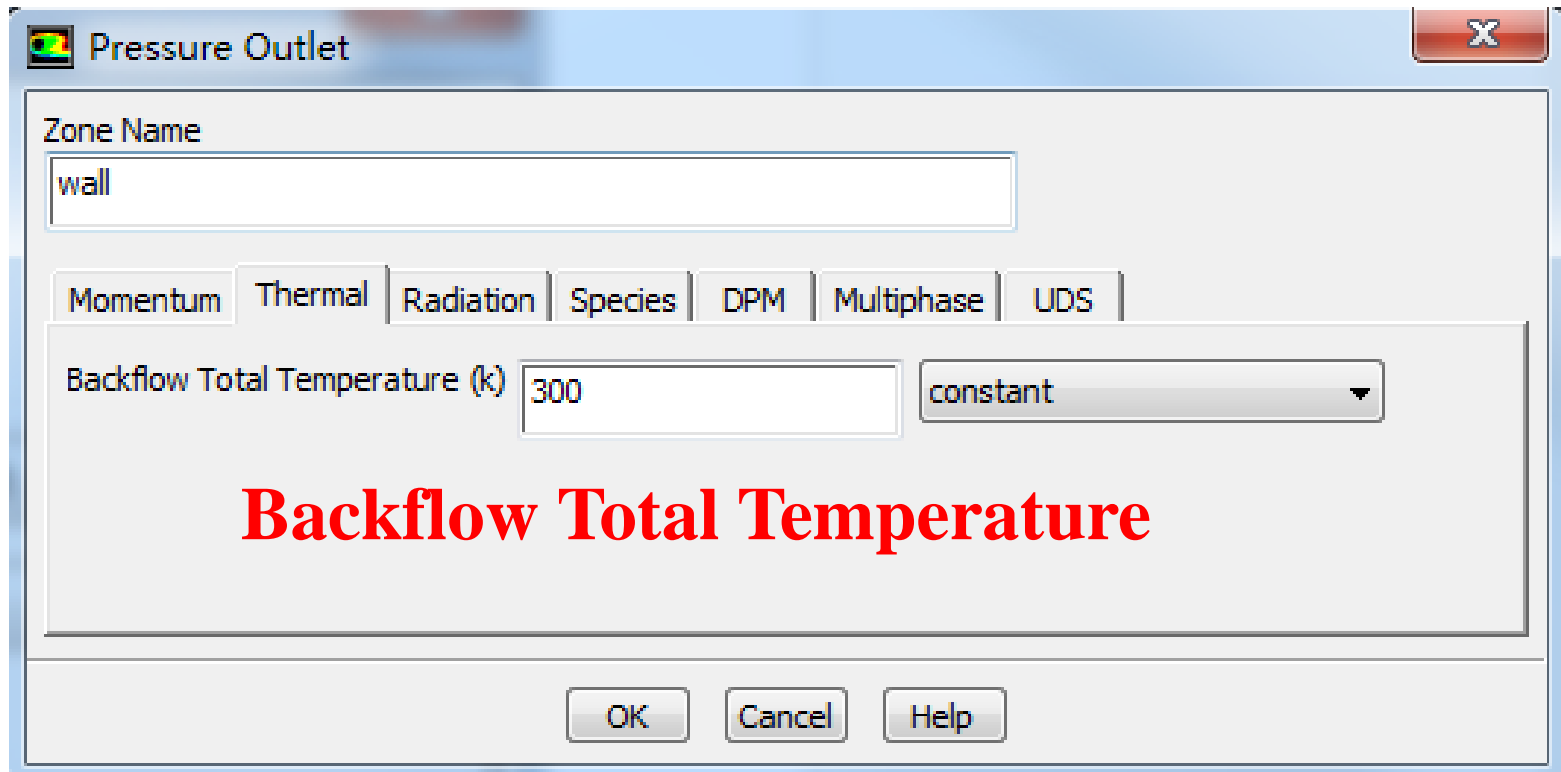
**Temperature**

# Outlet: pressure outlet



**Gauge Pressure (表压)**

For pressure outlet boundary condition, Fluent asks you to input the **Backflow Total Temperature**. However, it will play a role only if there is backflow. There is no information provided by Fluent Help File about what is the actual boundary condition for heat transfer.



# Wall: heat flux

Wall

Zone Name  
wall

Adjacent Cell Zone  
porous

Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film

Thermal Conditions

Heat Flux **heat flux** Heat Flux (w/m2) 100000 constant

Temperature

Convection

Radiation

Mixed

via System Coupling

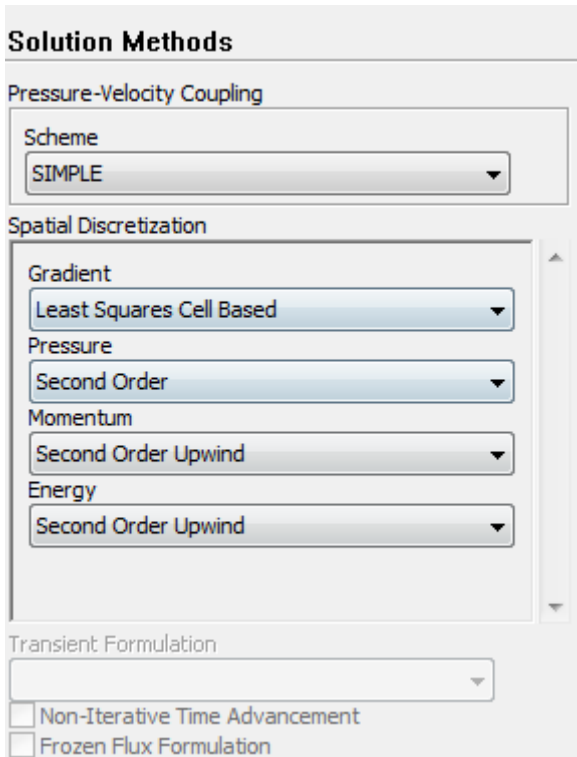
Wall Thickness (m) 0

Heat Generation Rate (w/m3) 0 constant

Material Name  
aluminum Edit...

## 7st step: Define the solution

For algorithm and schemes, keep it as default. For more details of this step, one can refer to Example 1 of Chapter 13.



**Algorithm:** simple

**Gradient:** Least Square Cell Based

**Pressure:** second order

**Momentum:** second order upwind

**Energy:** second order Upwind

## 7st step: **Define the solution**

For under-relaxation factor, keep it default. For more details, refer to **Example 1**.

## 8st step: Initialization

Use the standard initialization, for more details of Hybrid initialization, refer to Example 1.

## Step 9: Run the simulation

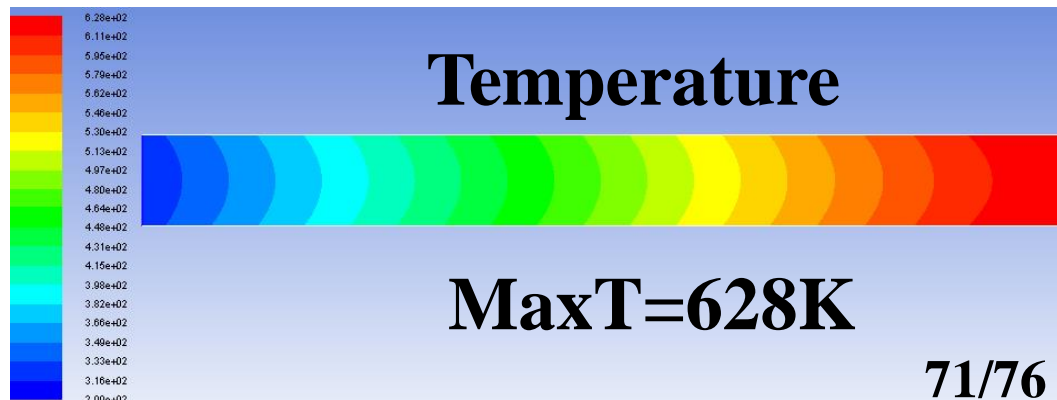
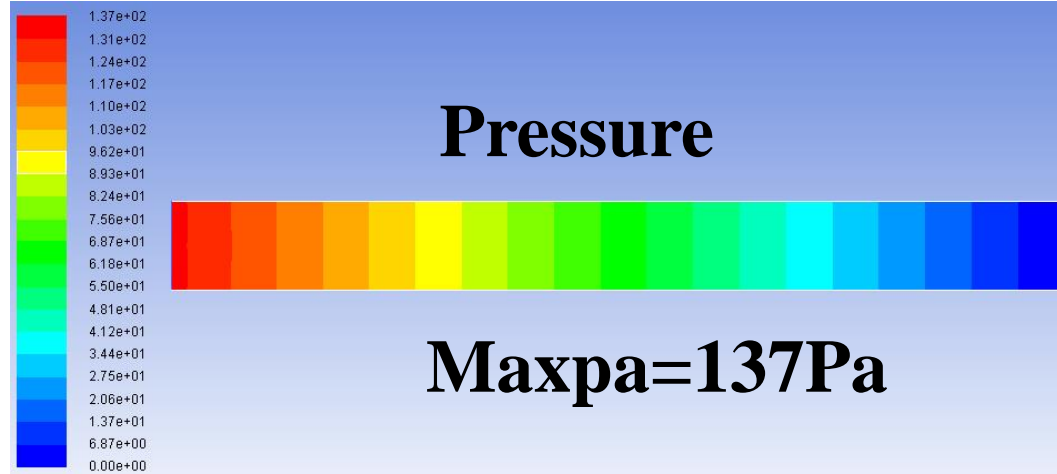
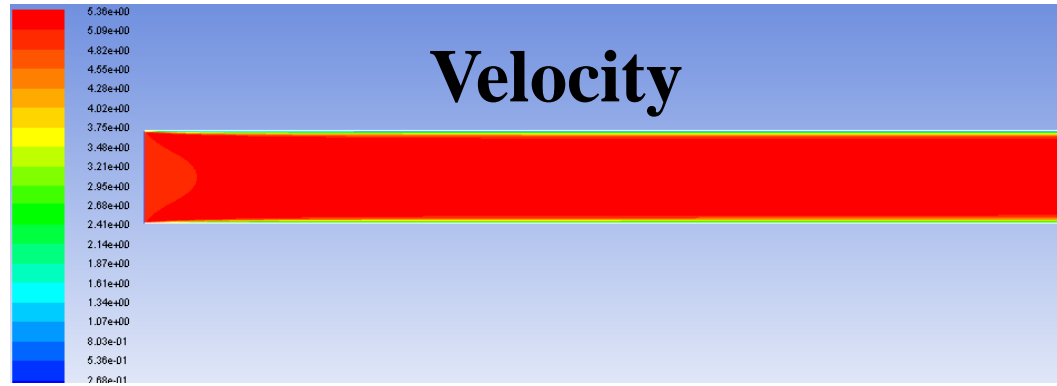
## Step 10: Post-processing results

$$1/k=1.4E7$$

$$C_2=0$$

$$\text{Porosity}=0.8$$

$$u=5$$



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

最后一次作业

2017年-2018学年

“数值传热学”课程教学评价调查

2017-2018

Evaluation of “Numerical Heat Transfer”

14 problems, less than 5 minutes to complete.

Your suggestions are highly required.

**All Chinese students must complete.**



## 2017 年-2018 学年“数值传热学”课程教学评价调查

(请在您同意部分括号内打钩或写下相应的数字)

1. 目前“数值传热学”的教学分成三部分：(1) 基础知识；(2) 二维流动传热问题开源代码；(3) 商业软件 Fluent。您认为这样的组成是否合适？  
很合适 (     ), 基本合适 (     ), 不合适 (     )。
2. 目前三部分学时分配大致是：(1) 基础部分 40；(2) 开源代码 8；(3) Fluent 12。您认为是否合适？  
很合适 (     ), 基本合适 (     ), 不合适 (     )；  
如果不合适，您认为哪一部分应加强 (     ), 哪一部分可减少 (     )。
3. 目前基础部分学习分为 9 章 (Ch1 – Ch 7, Ch10, Ch11), 您认为各章学时分配是否合理？  
合理 (     ), 基本合理 (     ), 不合理 (     )；  
如果不合理，哪一章应当加强？ (     ), 哪一章可以减少？ (     )。
4. 为了使学生有更长时间学习二维开源代码，目前开源代码及其应用在湍流模型及网格生成技术前讲授，这样的安排是否合适？  
很合适 (     ), 基本合适 (     ), 不合适 (     )。  
如果不合适，您的建议是\_\_\_\_\_。
5. 目前 Fluent 的教学分成两部分，(1) 基本使用方法及网格生成和 (2) 应用举例。您认为这样的安排是否合适？  
很合适 (     ), 基本合适 (     ), 不合适 (     )；  
如果不合适，您的建议是\_\_\_\_\_。



6. Fluent 两部分教学时间比例各 6 个学时, 您认为是否合适?  
合适 (    ), 不合适 (    );  
如果不合适哪一部分应增加时间? (    );
  
7. 鉴于部分学生已经有一定的使用 Fluent 的基础, 下次教学拟将 Fluent 的教学分为基础班及提高班 (不多于 60 人) 两部分, 您认为合适否?  
合适 (    ), 不合适 (    )。
  
8. 全程英语授课对教师是一个挑战, 为加深对基本概念及重点内的理解, 对重点部分采用了英语讲授后用汉语复述的方法, 您认为是否合适?  
合适 (    ), 不合适 (    )。
  
9. 您认为助教批改作业是否合理准确?  
是 (    ), 否 (    )  
您对于改进助教工作的建议是\_\_\_\_\_。
  
10. 您认为课程基础知识部分与 Fluent 介绍部分的衔接是否顺畅?  
是 (    ), 基本是 (    ), 不是 (    )  
如果不是您的建议是\_\_\_\_\_
  
11. 您认为 Fluent 讲解的两部分衔接是否流畅?  
是 (    ), 否 (    )
  
12. ICEM 与 Fluent 的教学采用流程介绍与演示相结合的方法, 在演示环节您可否能跟得上老师的思路?  
ICEM 教学: 可 (    ), 否 (    ); Fluent 教学: 可 (    ) 否 (    )  
如果不是您的建议是\_\_\_\_\_
  
13. Fluent 软件教学部分您是赞成使用现场演示操作(    )还是根据算例截图讲解(    )?
  
14. 热烈欢迎您写下对改进本课程教学效果的其他宝贵意见:

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....

# Thanks!

欢迎交流

陈黎，教授，博导

热流科学与工程教育部重点实验室，西安交通大学

邮箱：[lichennht08@mail.xjtu.edu.cn](mailto:lichennht08@mail.xjtu.edu.cn)

个人主页：<http://lichennht08.gr.xjtu.edu.cn/>

**硕博士：每年招收3-4硕博士**

**博士后岗位：年薪18万，五险一金，子女入托上学，两年后考核副研岗**



*同舟共济渡彼岸!*

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