

# Numerical Heat Transfer

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



**Instructor Wen-Quan Tao; Qinlong Ren; Li Chen**

**CFD-NHT-EHT Center**  
**Key Laboratory of Thermo-Fluid Science & Engineering**  
**Xi'an Jiaotong University**  
**Xi'an, 2019-Dec.-23**

# 数值传热学

## 第 13 章 求解流动换热问题的Fluent软件基础应用举例



主讲 陶文铨

辅讲 任秦龙, 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

2018年12月23日, 西安

# 第 13 章 求解流动换热问题的Fluent软件基础应用举例

**13.1 Heat transfer with source term**

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

**13.3 Lid-driven flow and heat transfer**

**13.4 Flow and heat transfer in a micro-channel**

**13.5 Flow and heat transfer in chip cooling**

**13.6 Phase change material melting with fins**

# 第 13 章 求解流动换热问题的Fluent软件基础应用举例

**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**

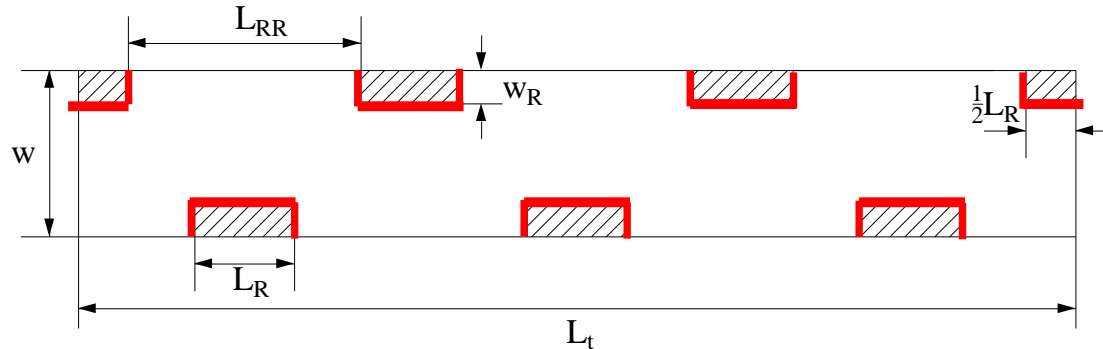
**13.5 芯片冷却流动换热问题**

微通道问题

**13.6 肋片强化相变材料融化**

相变传热

## 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”.

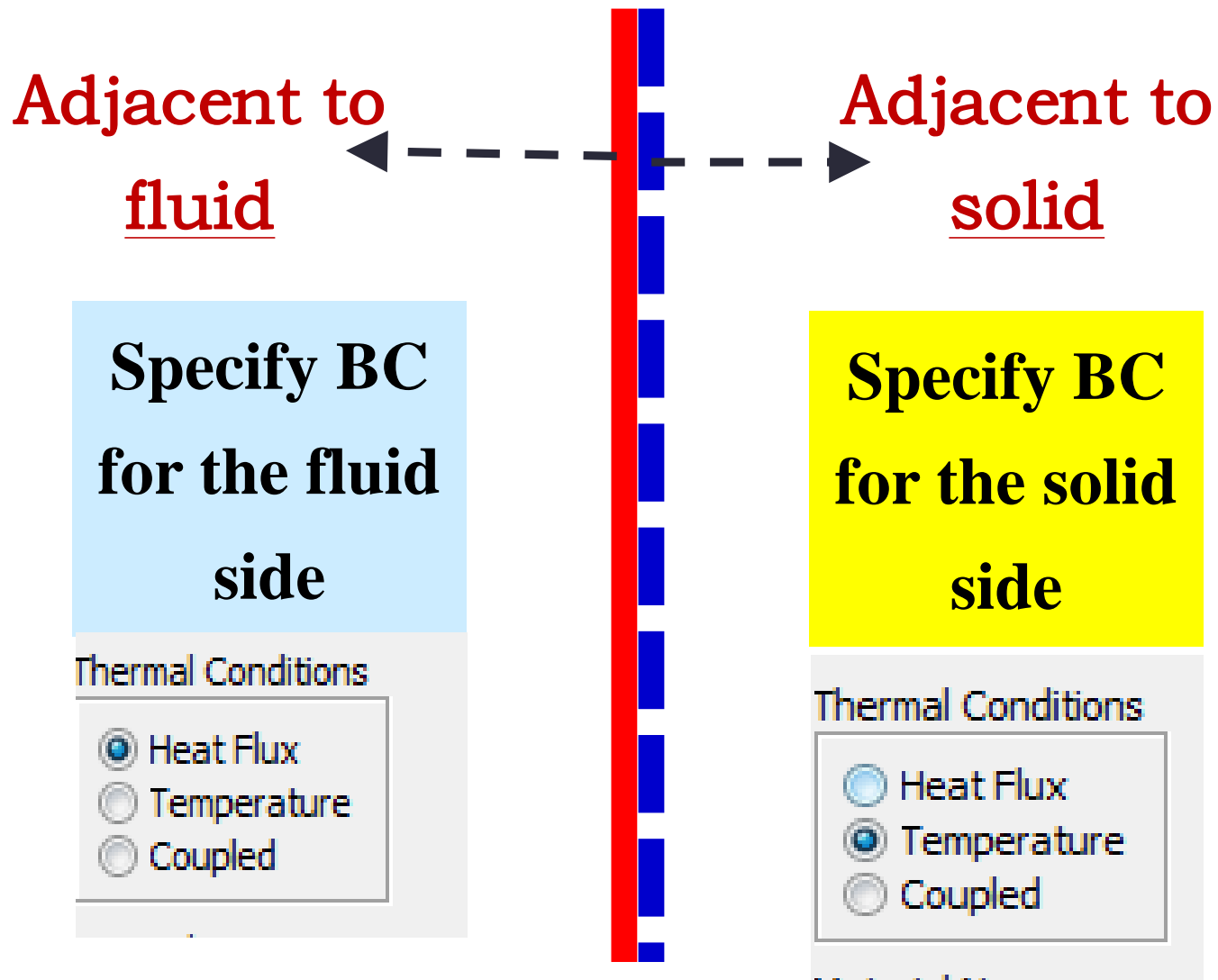
## Thermal Conditions

- Heat Flux
- Temperature
- Coupled

- Heat flux
- Temperature
- Coupled

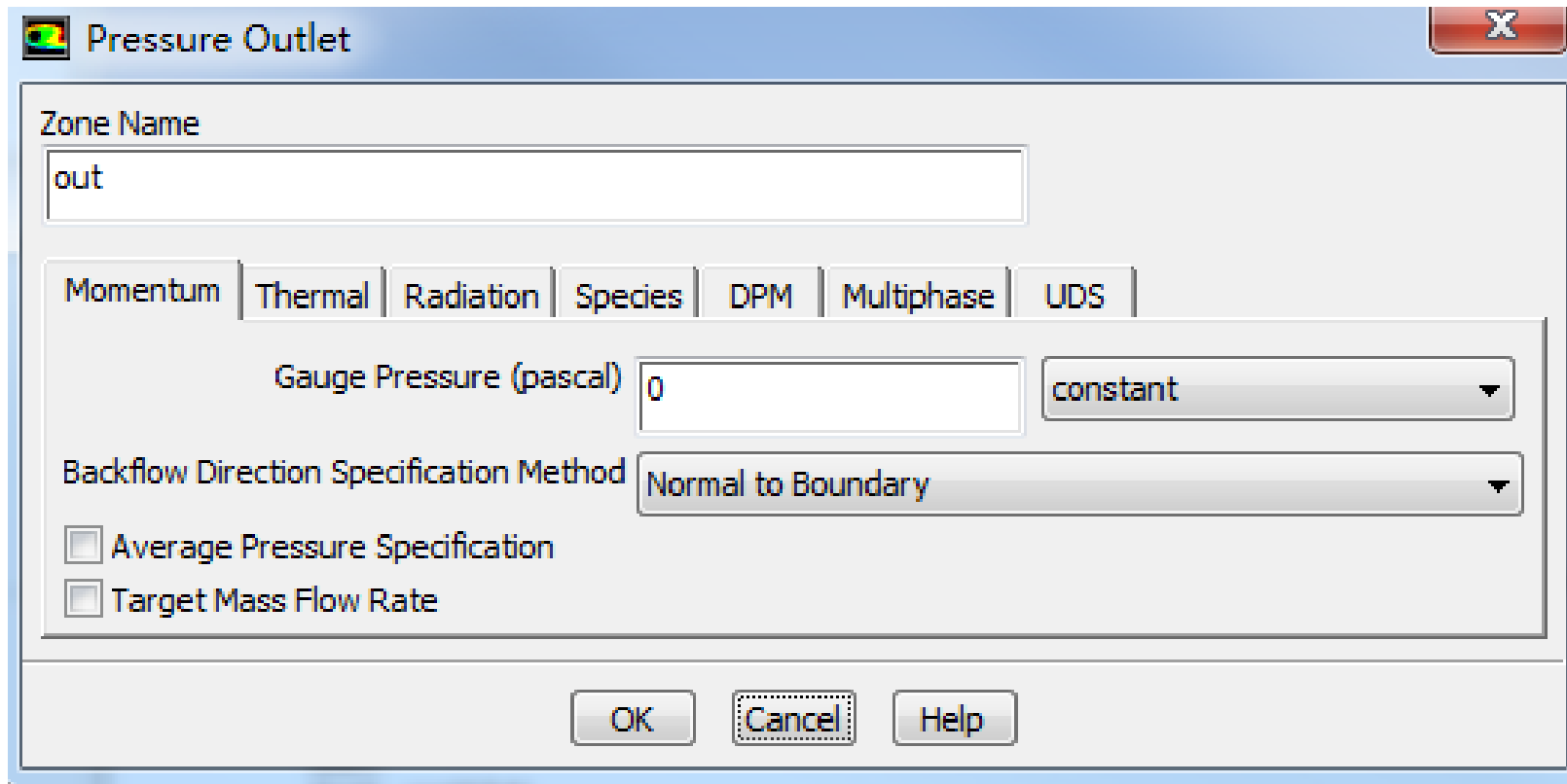
If you choose “**Coupled**”, no additional information is required. The solver will calculate heat transfer directly from the solution of adjacent cells. **Such wall is not a boundary.**

# The original two side wall



Its shadow created by Fluent

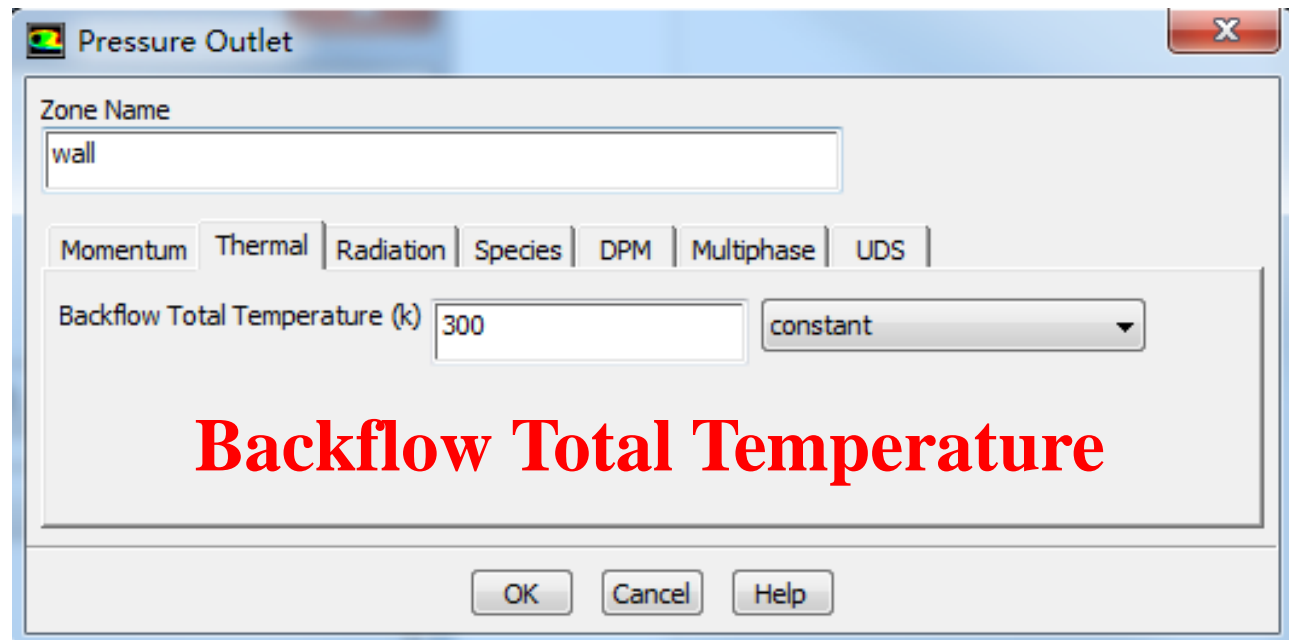
# Pressure outlet boundary condition



**Gauge Pressure (表压)**

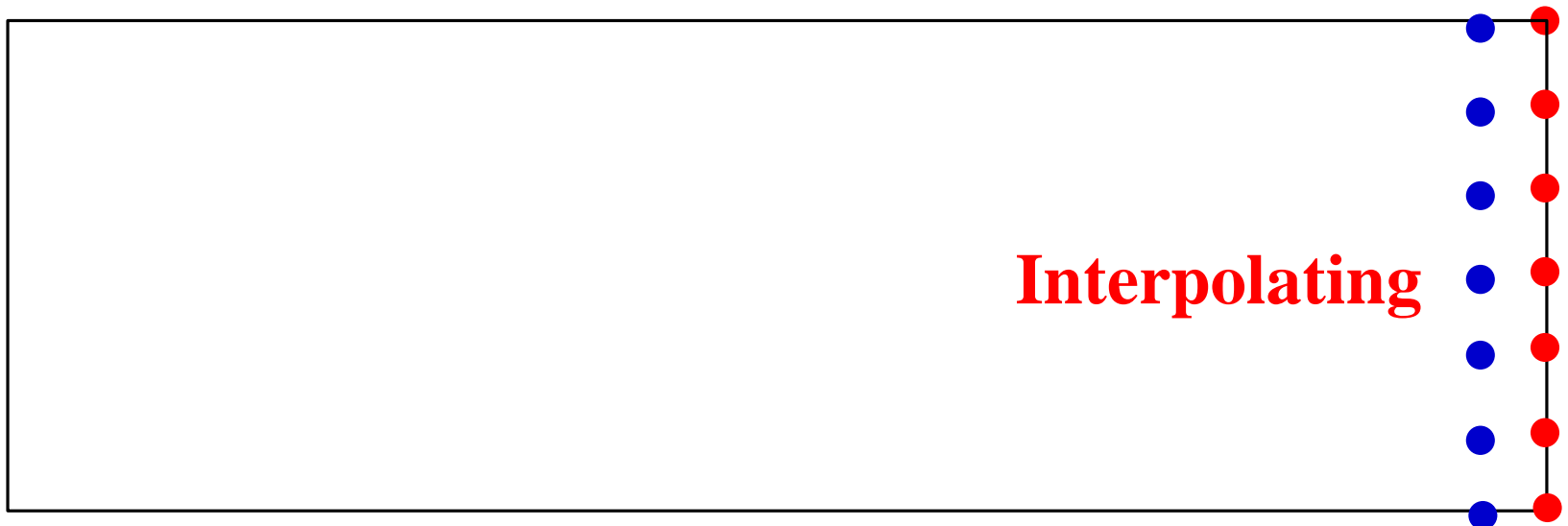


For pressure outlet boundary condition, Fluent asks you to input a **Backflow (回流) Total Temperature**. However, it will play a role only if there is backflow. There is **no information provided by Fluent Help File** about what is the actual boundary condition for heat transfer.



**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

### 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) .

## 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

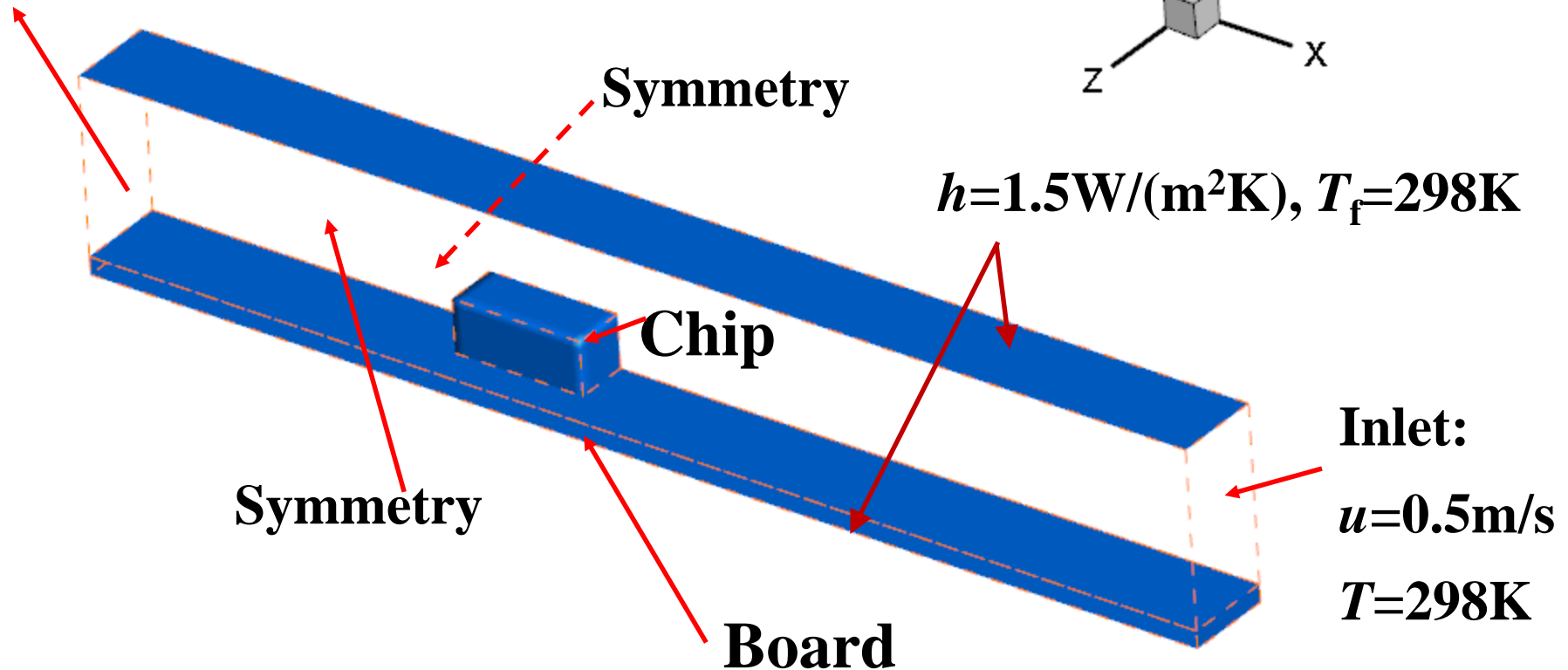
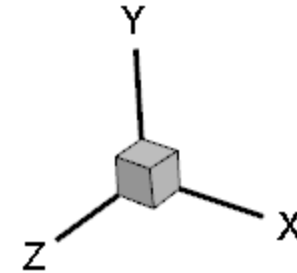


Fig.1 Computational domain

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

**Solution:**

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

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

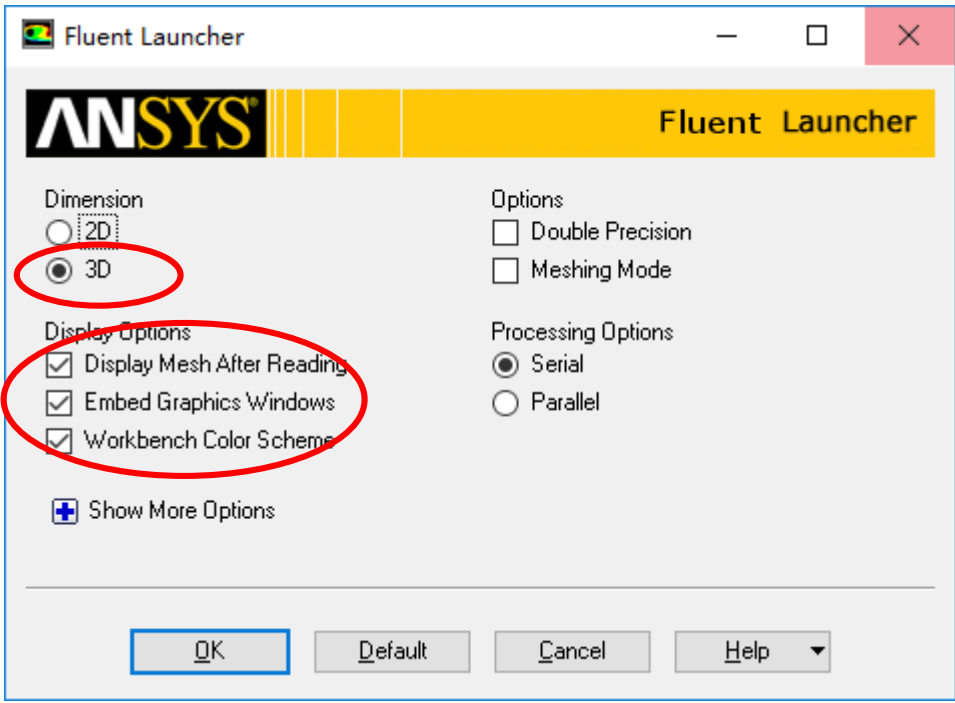
$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{1}{\rho_f} \frac{\partial p}{\partial y} + \frac{\mu_f}{\rho_f} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)$$

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

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



# 13.5.1 Start the Fluent software

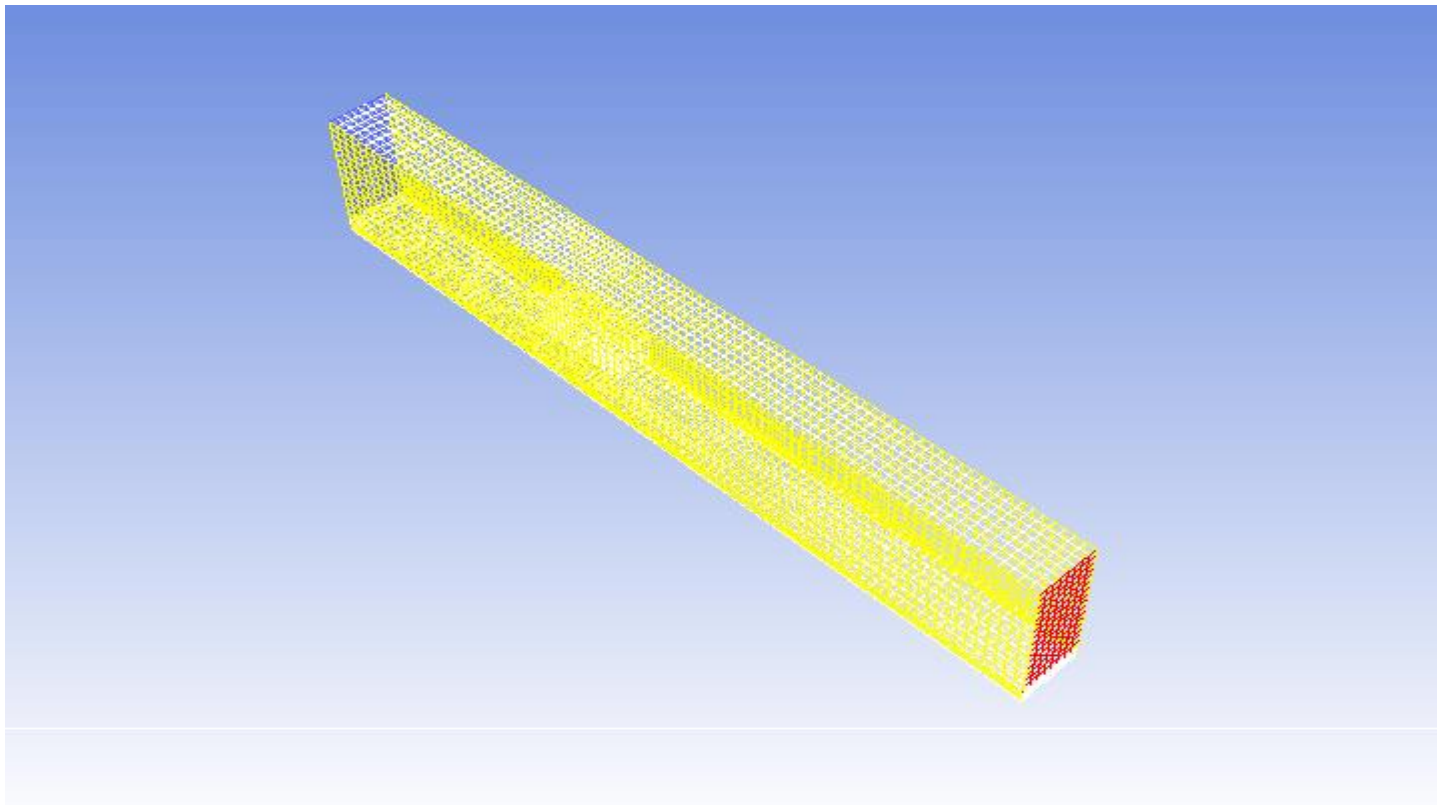


- 1. Choose **3-Dimension**
- 2. Choose **display options**
- 3. Choose **Serial processing option**



## 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

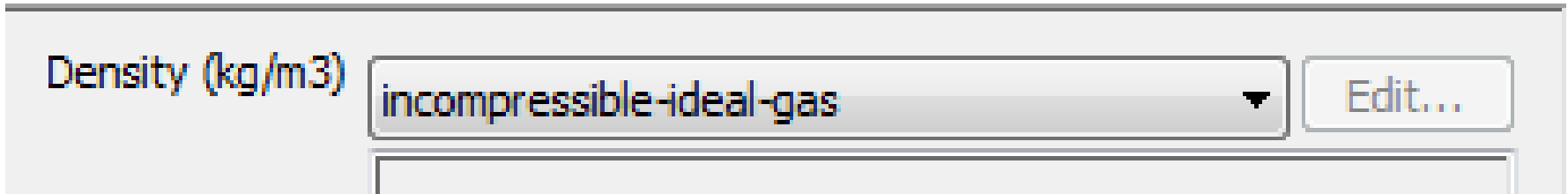
Model

- 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)

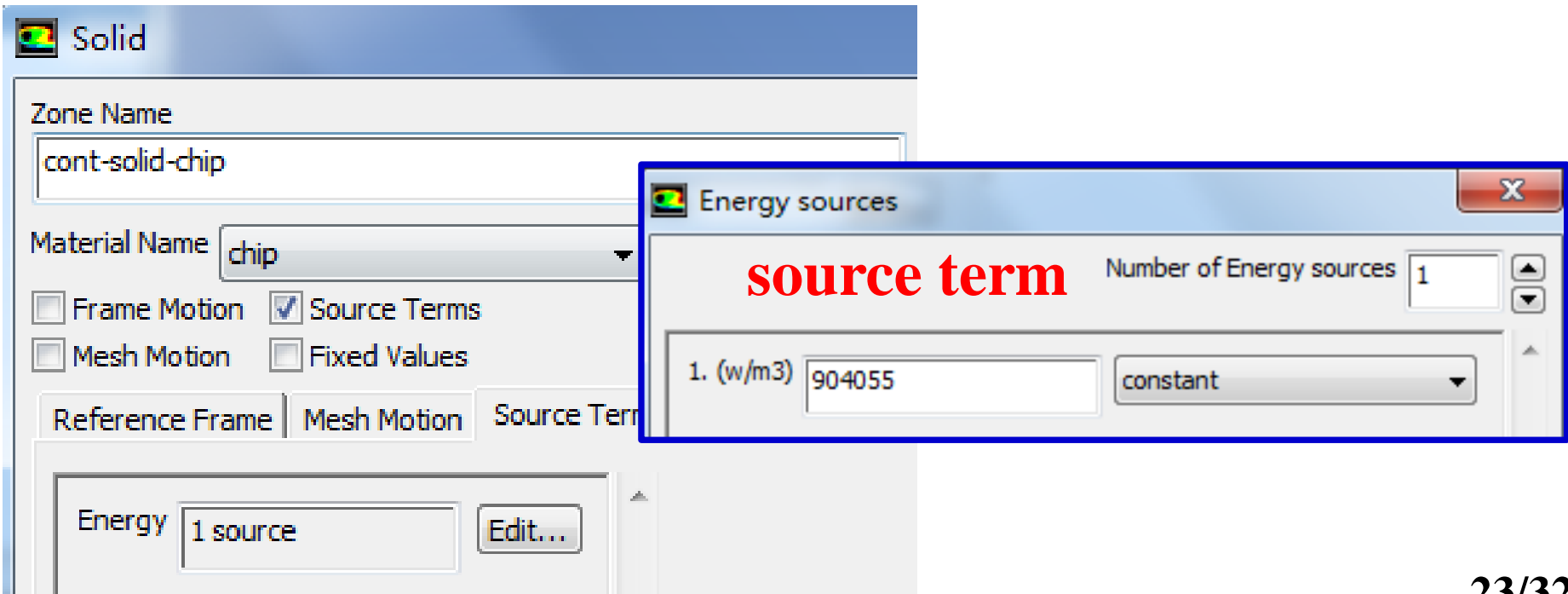
### Define a new material as Board:

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.

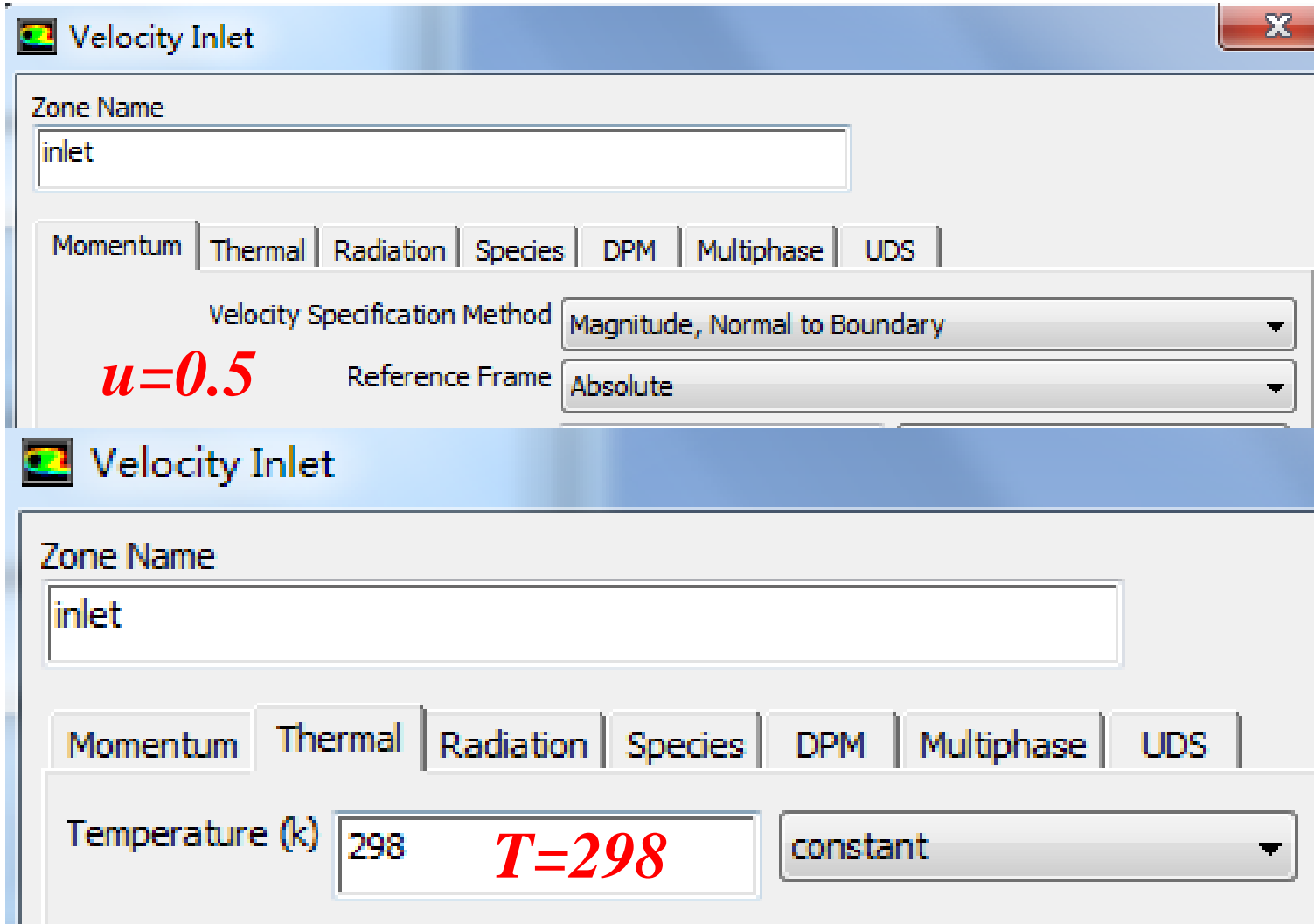
For the chip, there is a source term with value of  $904055 \text{ W/m}^3$



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'. The 'Energy' field in the 'Solid' dialog shows '1 source'.

## 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) set to 298 and the Temperature type set to "constant". A red annotation  $T=298$  is placed next to the Temperature (k) input field. Both dialog boxes have the Zone Name set to "inlet".

**Velocity Inlet**

Zone Name: inlet

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

Velocity Specification Method: Magnitude, Normal to Boundary

$u=0.5$  Reference Frame: Absolute

**Velocity Inlet**

Zone Name: inlet

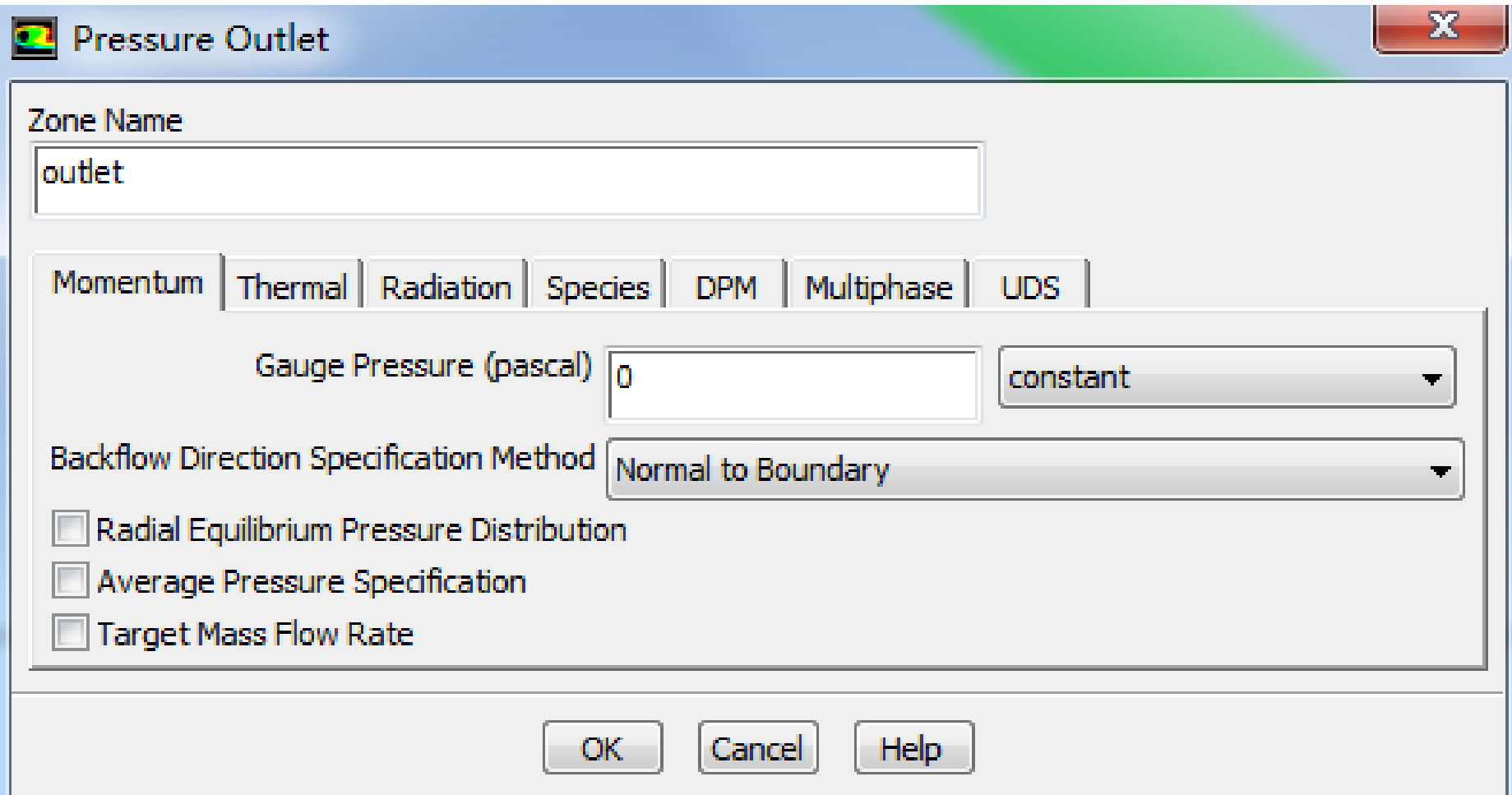
Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Temperature (k): 298  $T=298$  constant



## 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

# 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...

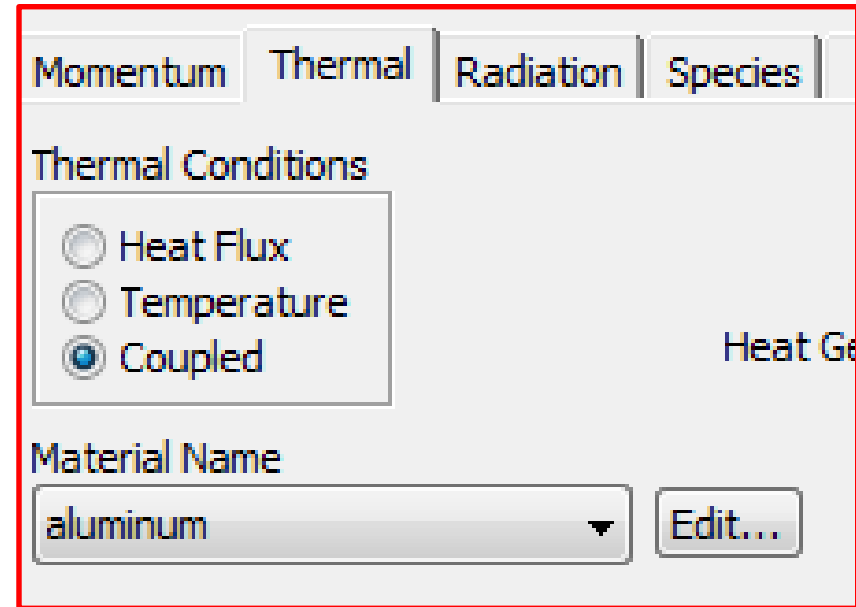
Shell Conduction Define...

OK Cancel Help

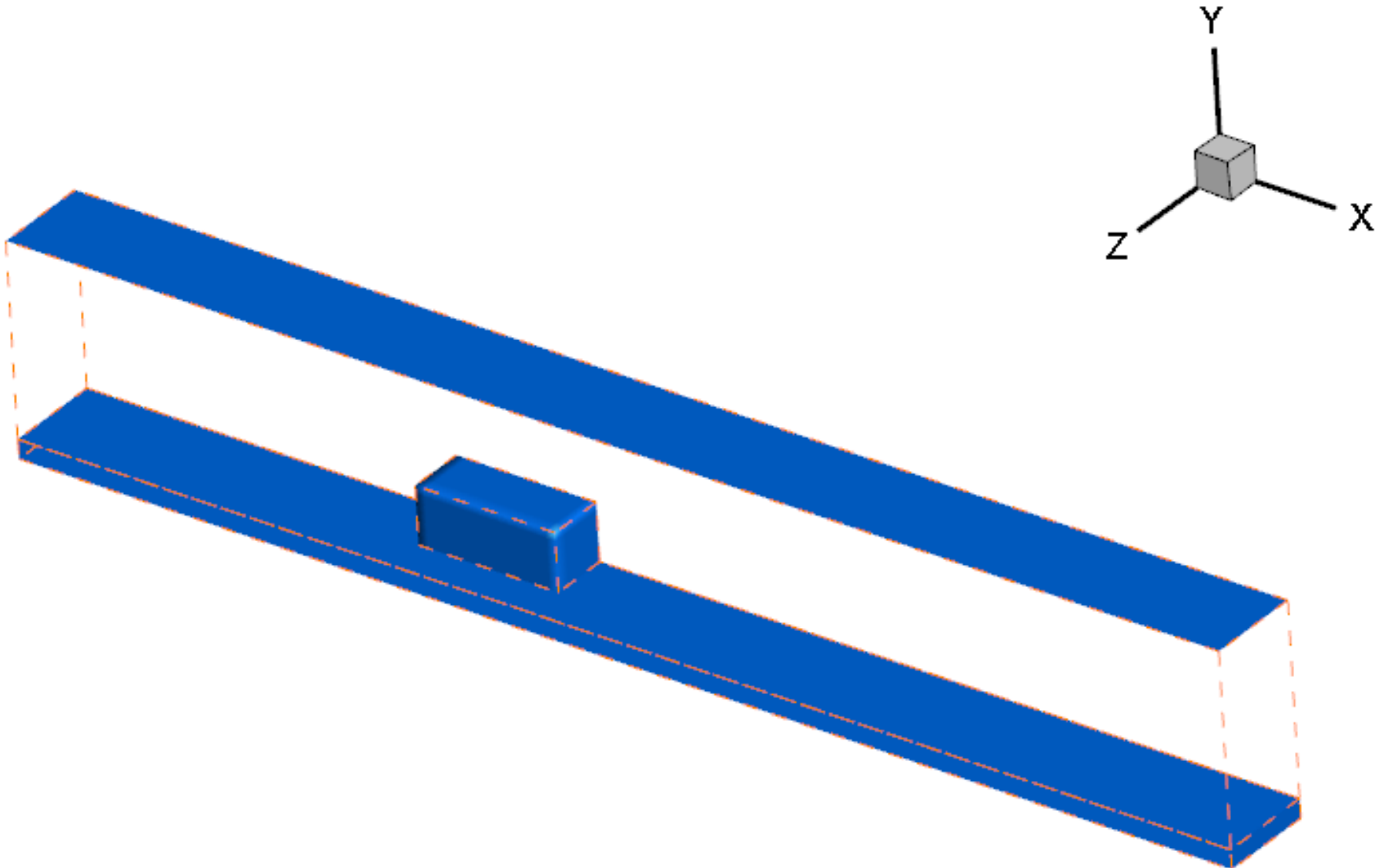
## Step 6: Define the boundary condition

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.*

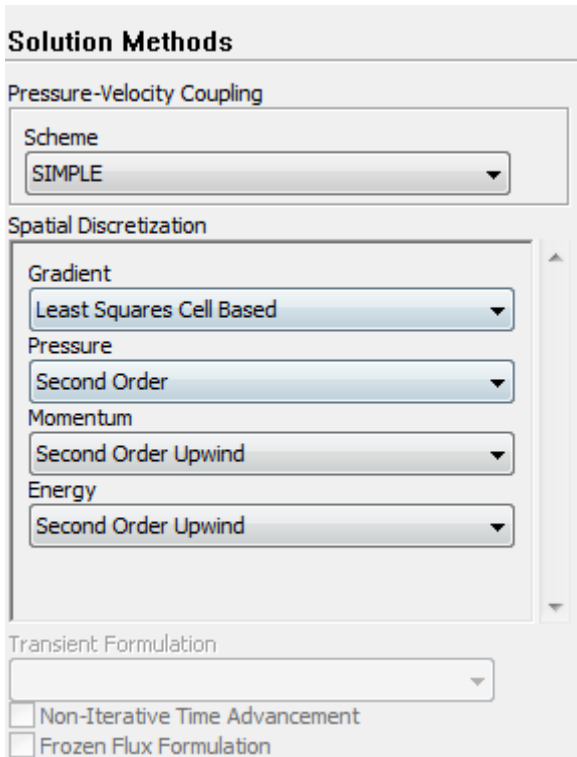


There are many **two-sided-wall** in this Example.



## 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**.

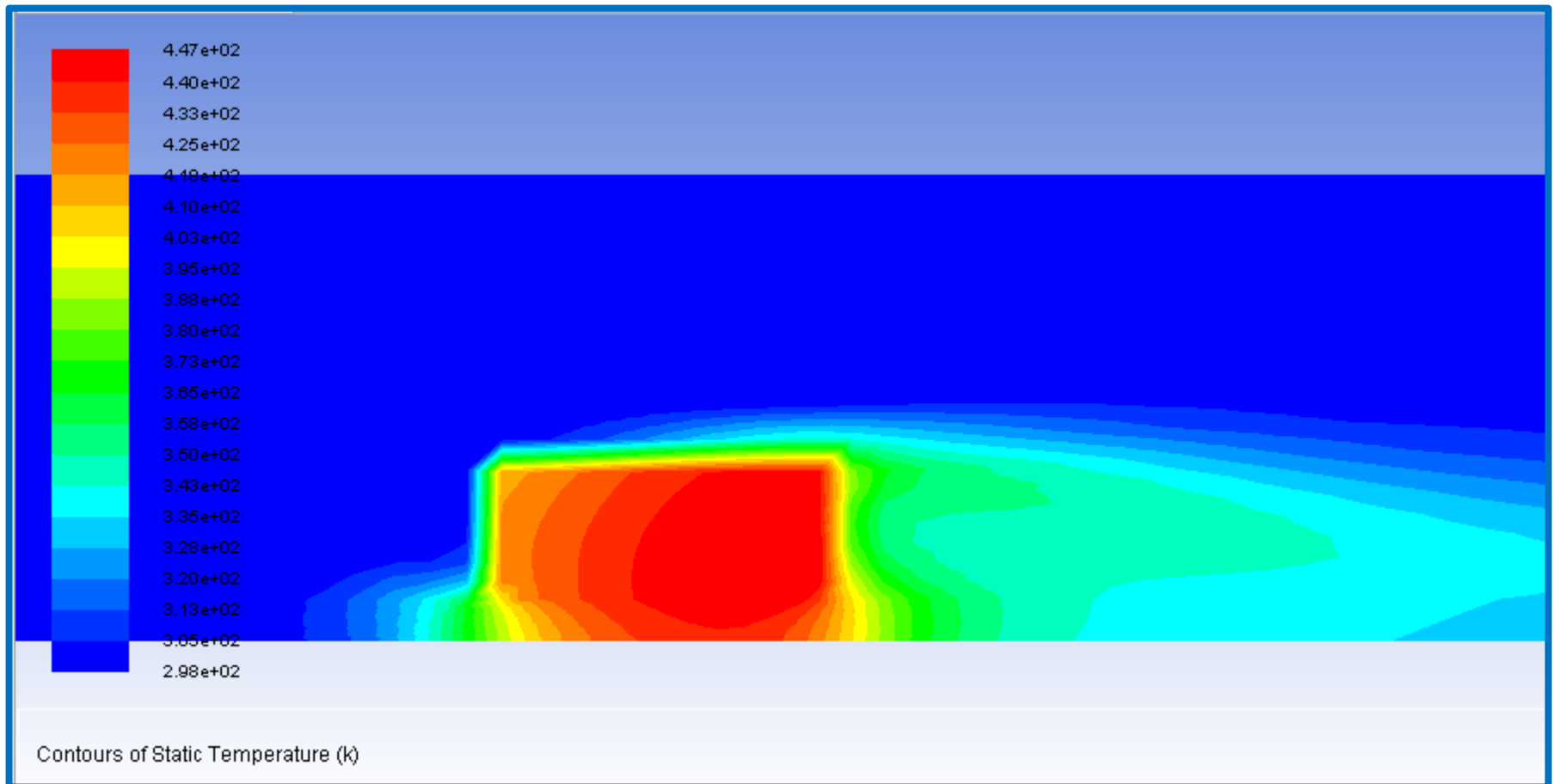
## 8st step: Initialization

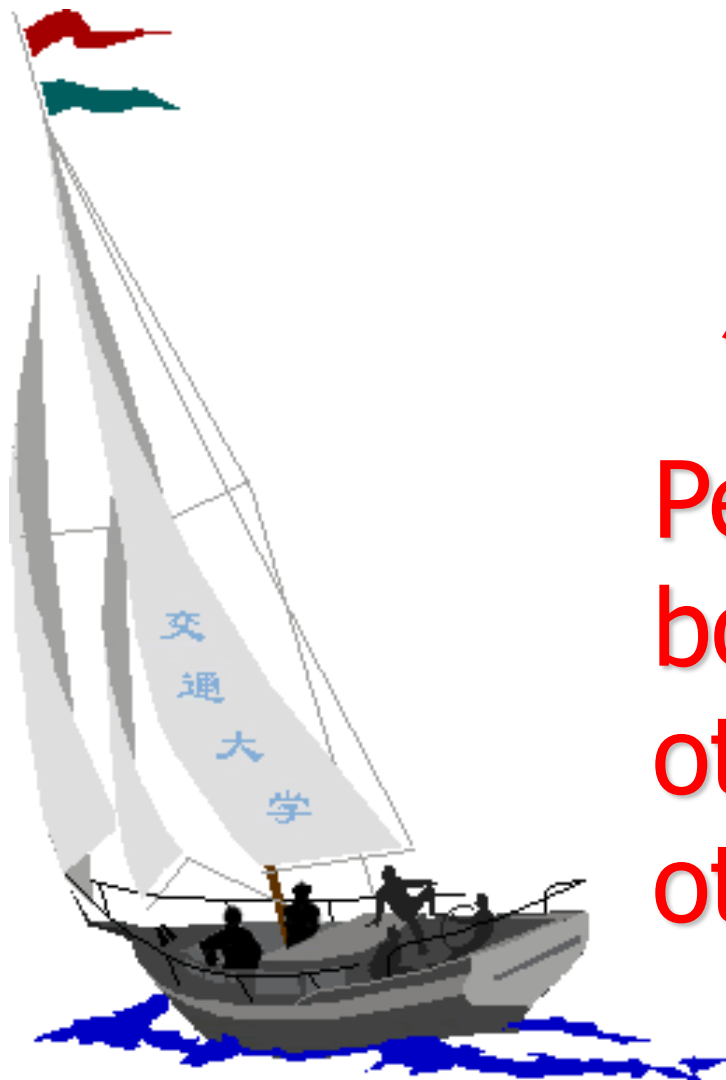
Use the standard initialization, for more details of Hybrid initialization, refer to Example 1.

## Step 9: Run the simulation

## Step 10: Post-processing results

# Static Temperature(K) of back boundary





# 同舟共济 渡彼岸!

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