Numerical Heat Transfer

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

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, 2020-Dec.-22

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
2020年12月22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

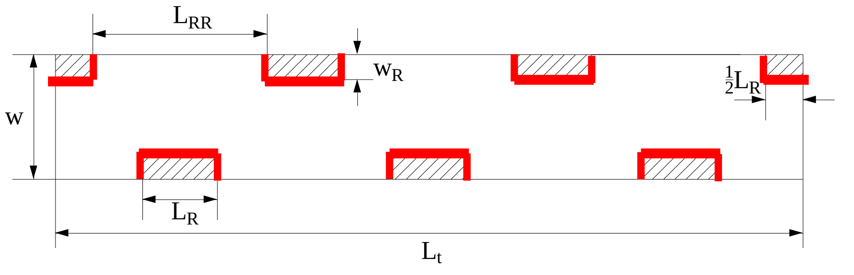
13.5 Flow and heat transfer in chip cooling

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

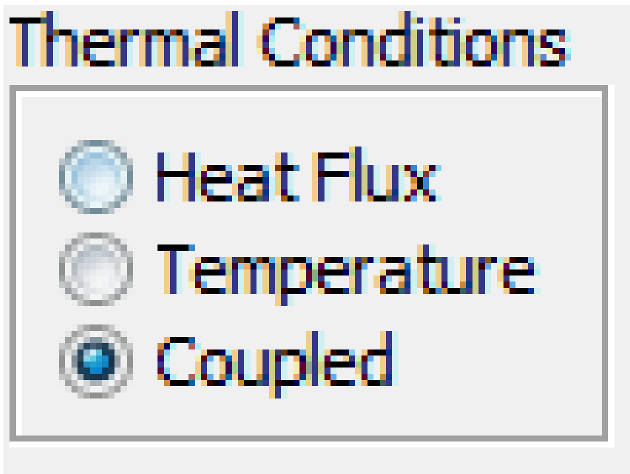
Example 4：Fluid-solid interface



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

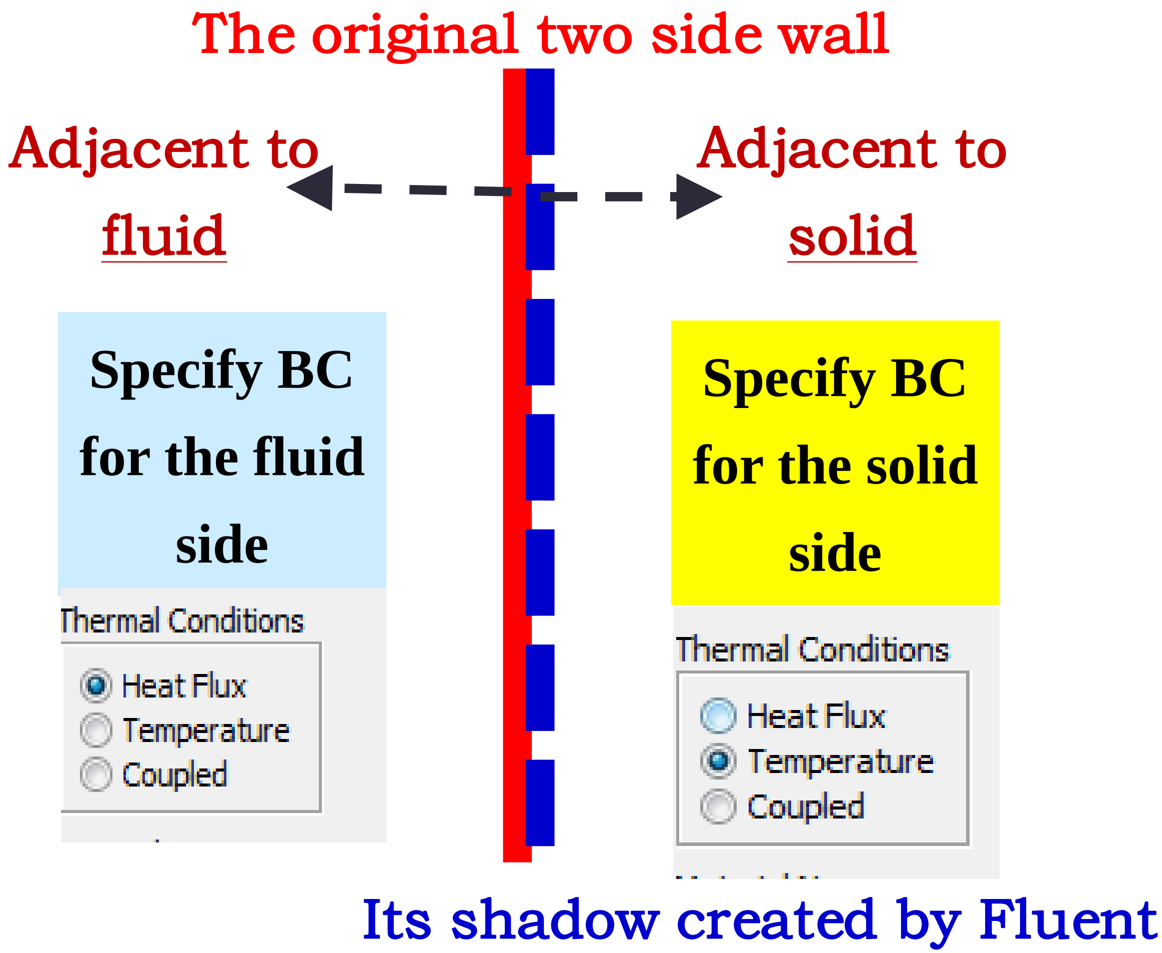
When such kind wall is read into Fluent, a "shadow" (影子） zone is automatically created.

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

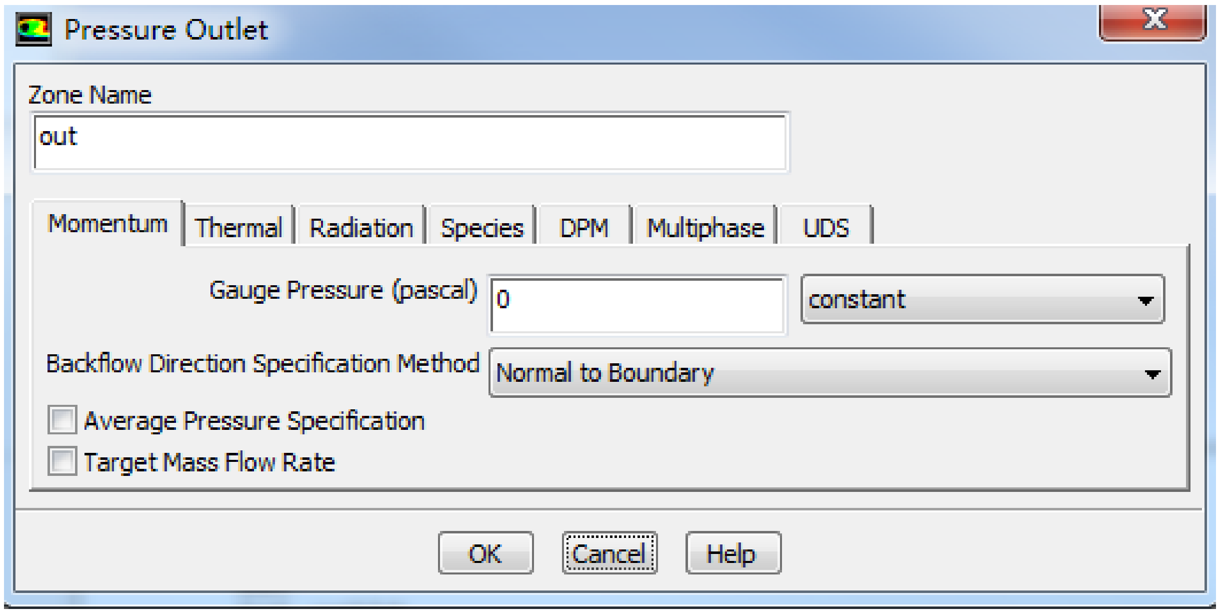


- Heat flux
- Temperature
- Coupled

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.

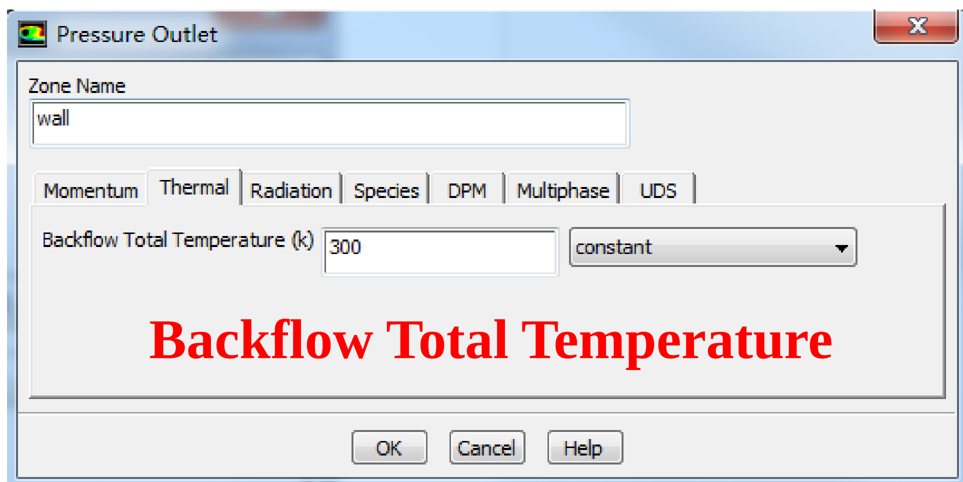

The original two side wall
Adjacent to fluid
Adjacent to solid
Specify BC for the fluid side
Thermal Conditions
Heat Flux
Temperature
Coupled
Specify BC for the solid side
Thermal Conditions
Heat Flux
Temperature
Coupled
Its shadow created by Fluent

Pressure outlet boundary condition



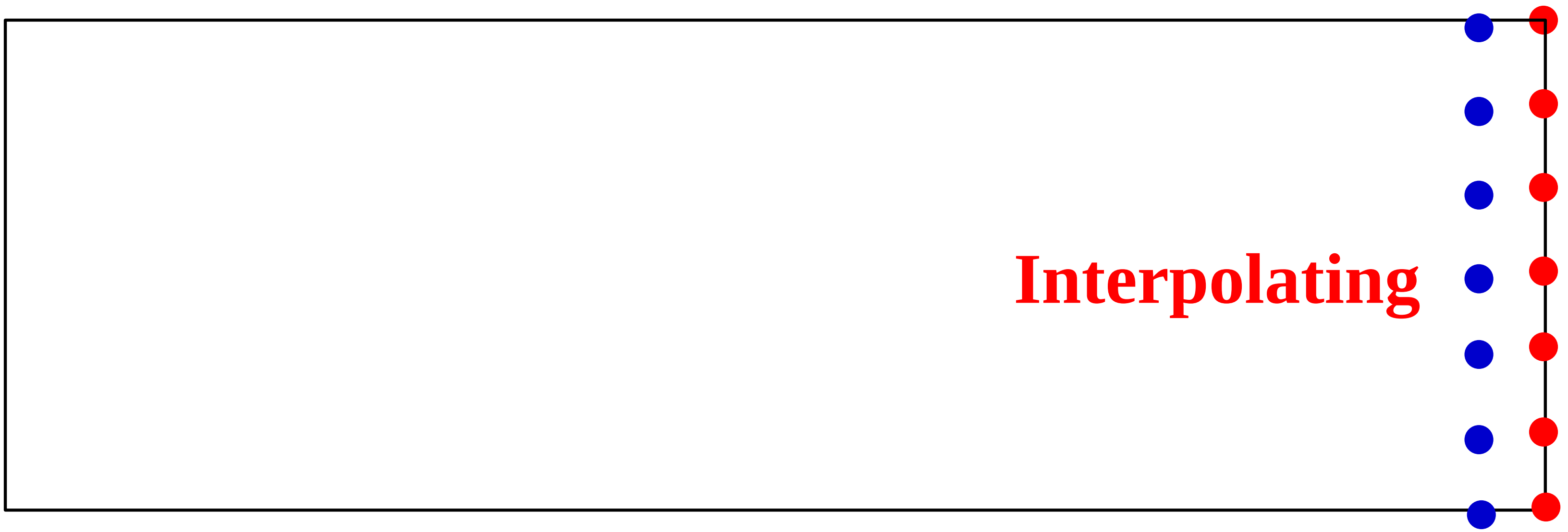
Gauge Pressure (表压)

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 0.5ρU^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*---0.5m/s (constant)**

  ***T*---298K**

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

- **Top and bottom boundary: 3rd boundary condition**

  Heat transfer coefficient: H=1.5 W/(m^2K);
  Free stream temperature: T_f=298K.

- Chip-- a constant source term, 904055 W/m^3
- Front surface and back surface---symmetry

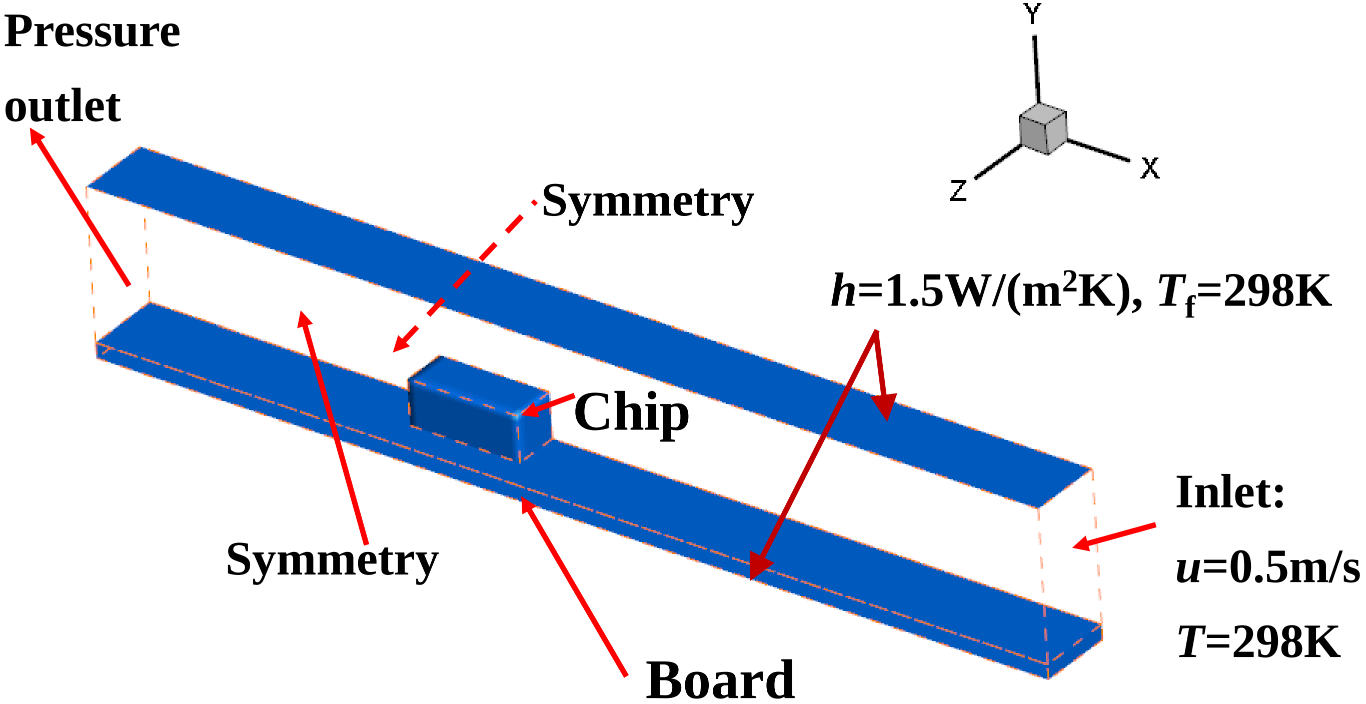


Fig.1 Computational domain

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

Solution:

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

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

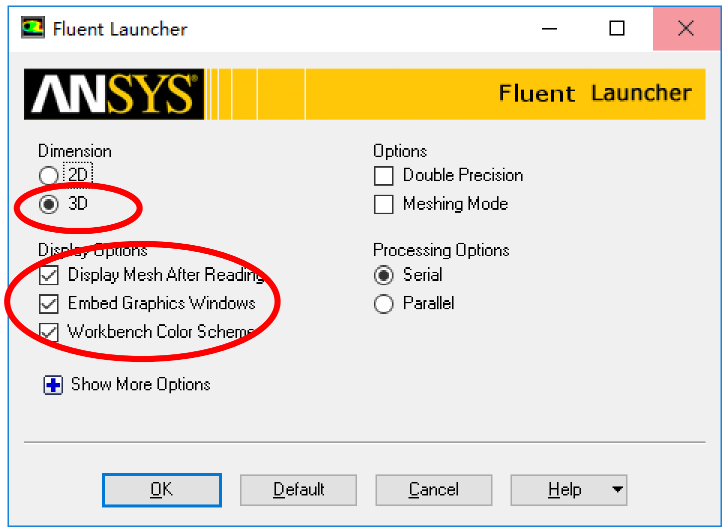
$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-\frac{1}{\rho_f}\frac{\partial p}{\partial y}+\frac{\mu_f}{\rho_f}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

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

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

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

13.5.1 Start the Fluent software



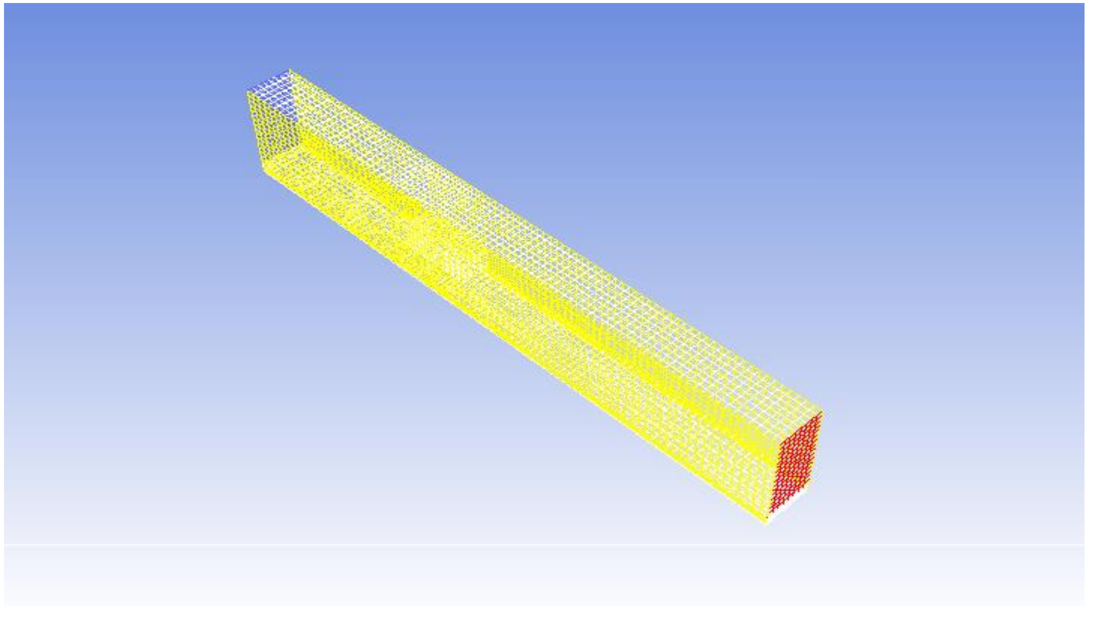
1. Choose 3-Dimension

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

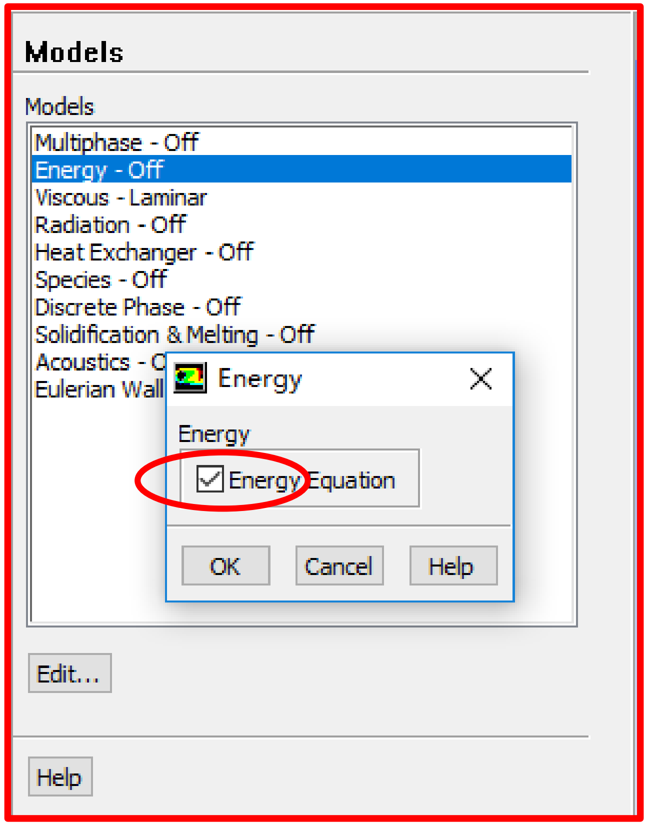
General→Scale

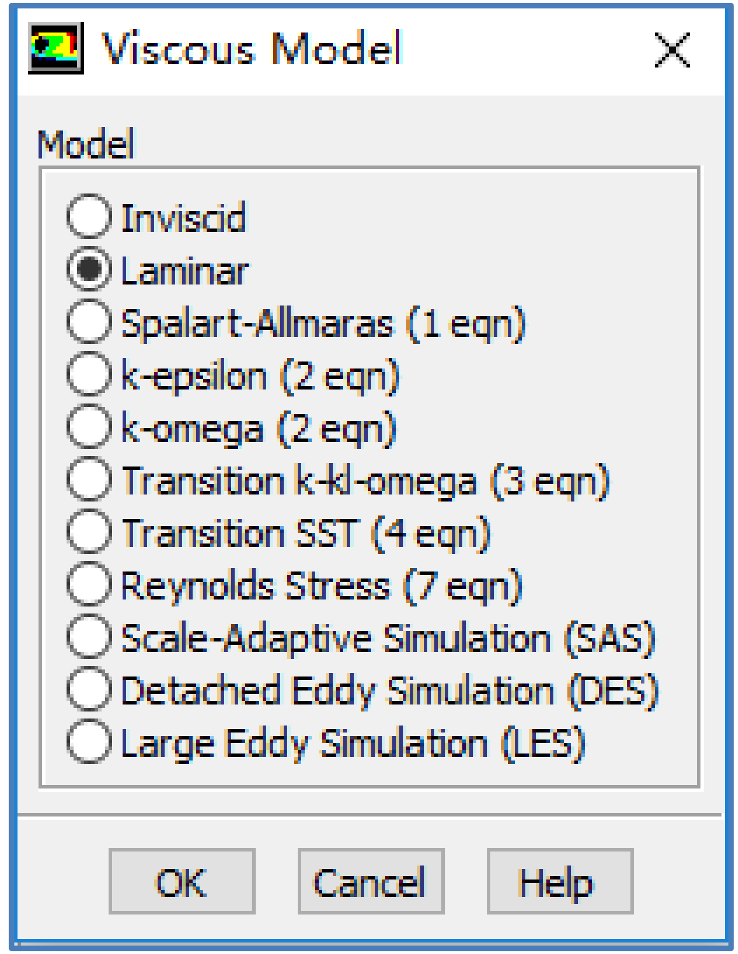
3st step: Choose the physicochemical model

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

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

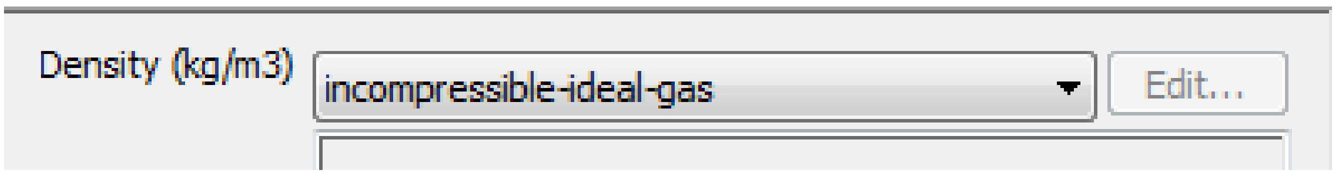
The density of air is 1.29 Kg/m³, the inlet velocity is 0.5 m/s, characteristic length is about 2 cm, and kinetic viscosity of air is 1.7894E-05. *Re* is 720 and thus flow is laminar.


Models
Models
Multiphase - Off
Energy - Off
Viscous - Laminar
Radiation - Off
Heat Exchanger - Off
Species - Off
Discrete Phase - Off
Solidification & Melting - Off
Acoustics - O
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-kl-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.



Define a new material as Chip:

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

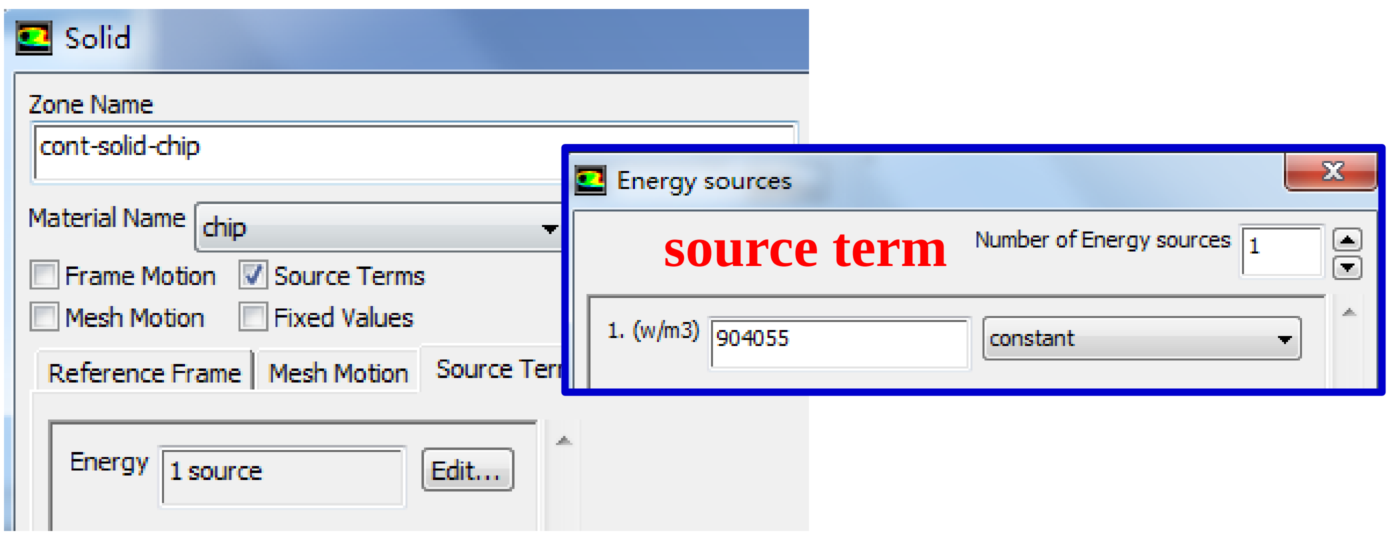
Define a new material as Board:

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

Step 5: Define zone condition

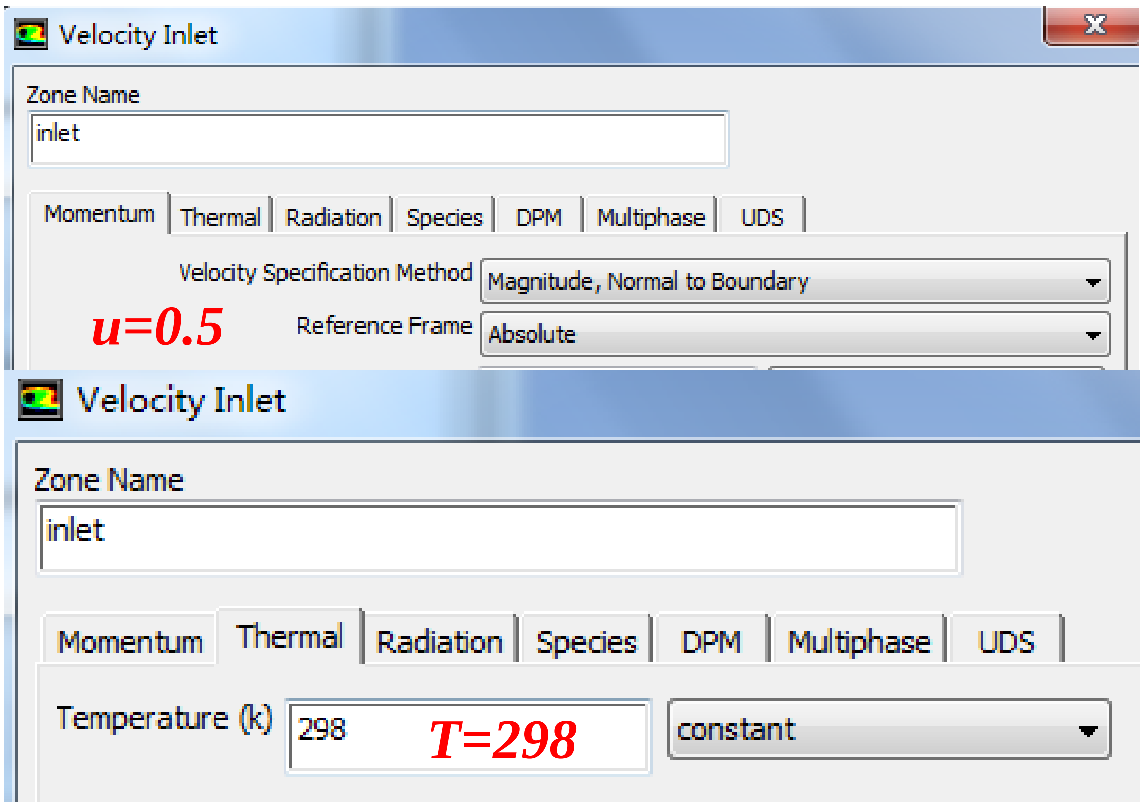
Assign different regions with the corresponding materials.

For the chip, there is a source term with value of 904055 W/m^3



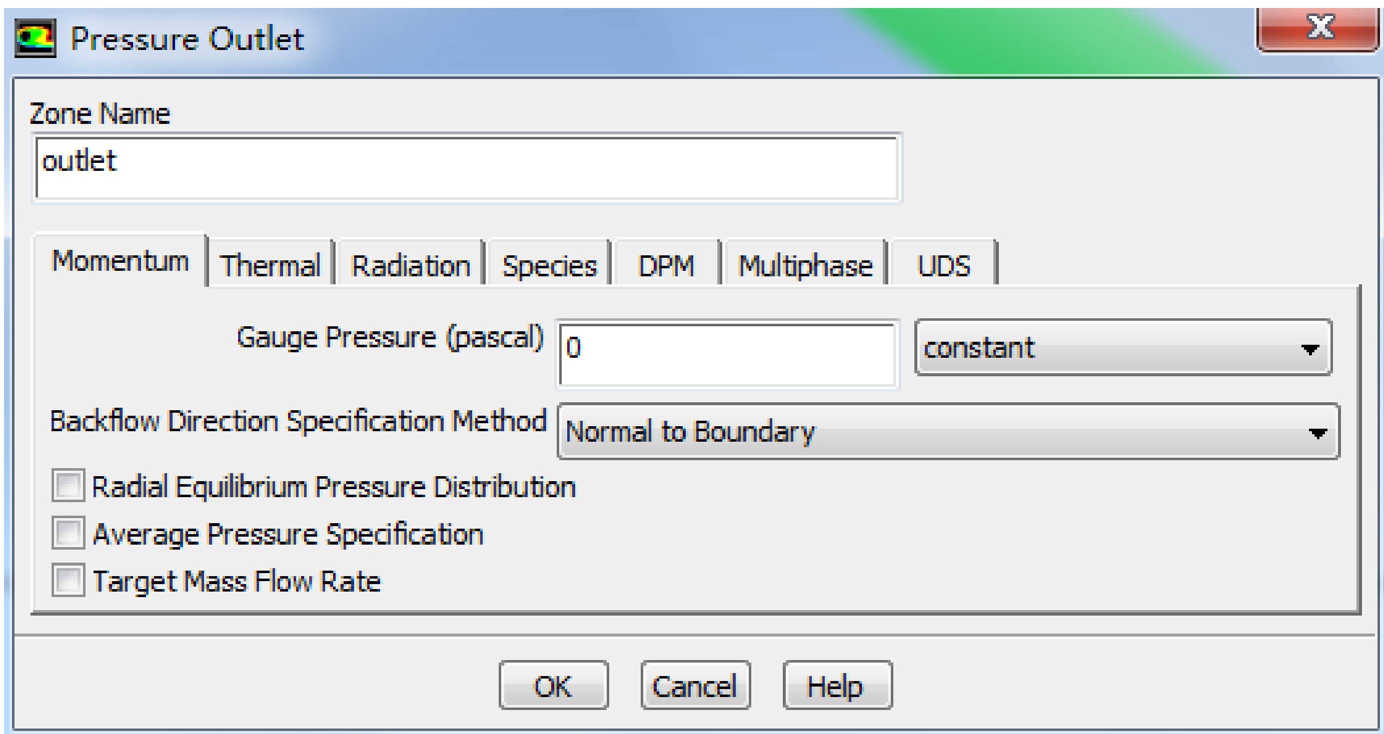
Step 6: Define the boundary condition

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



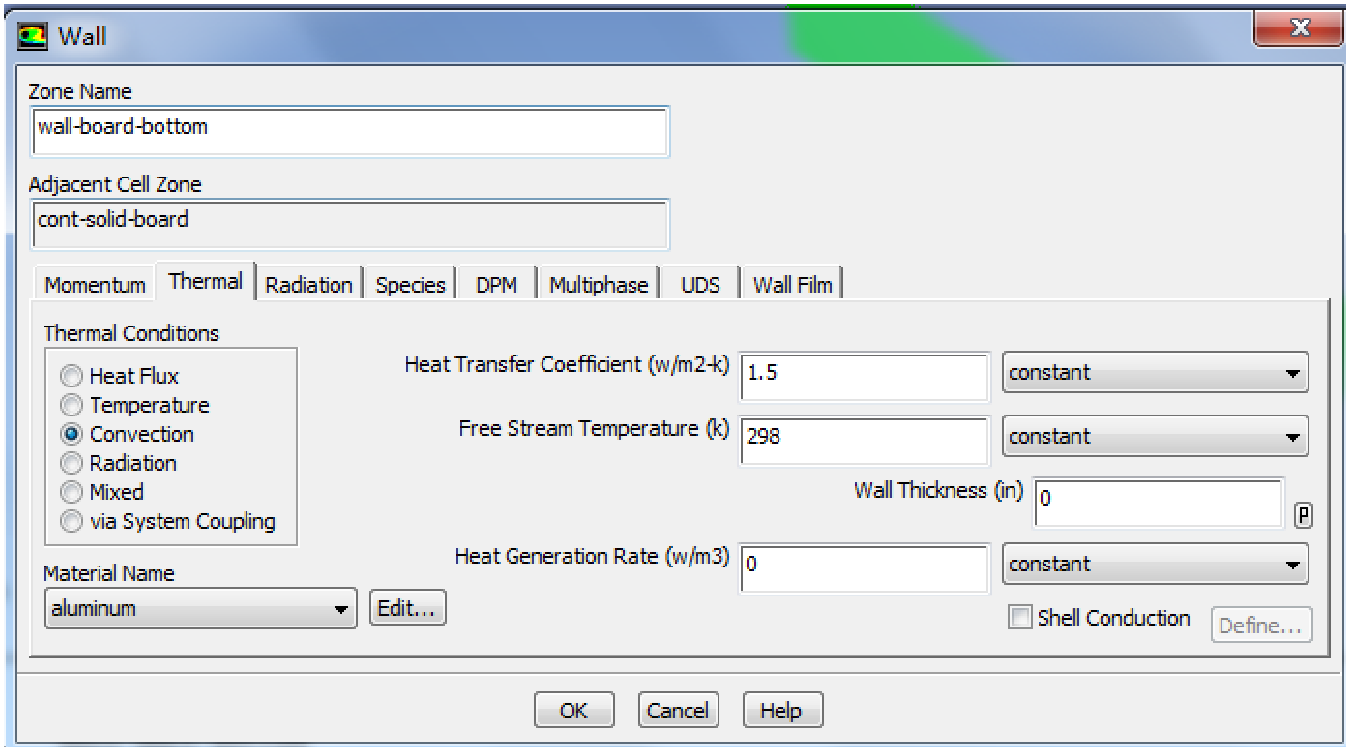
Step 6: Define the boundary condition

Outlet: pressure outlet, Gauge pressure as 0.



Step 6: Define the boundary condition

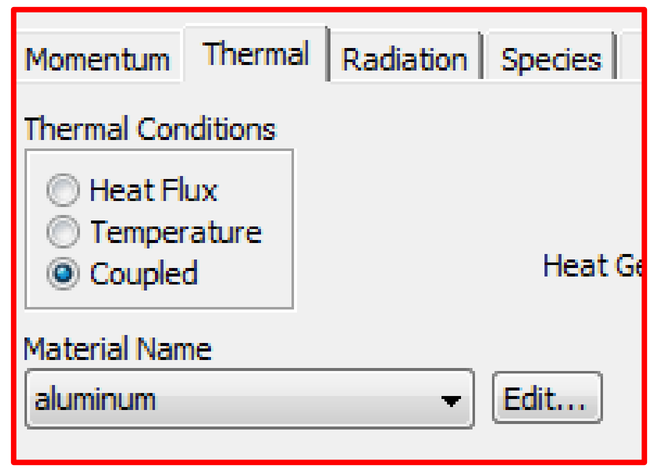
Top and bottom wall: convective boundary condition



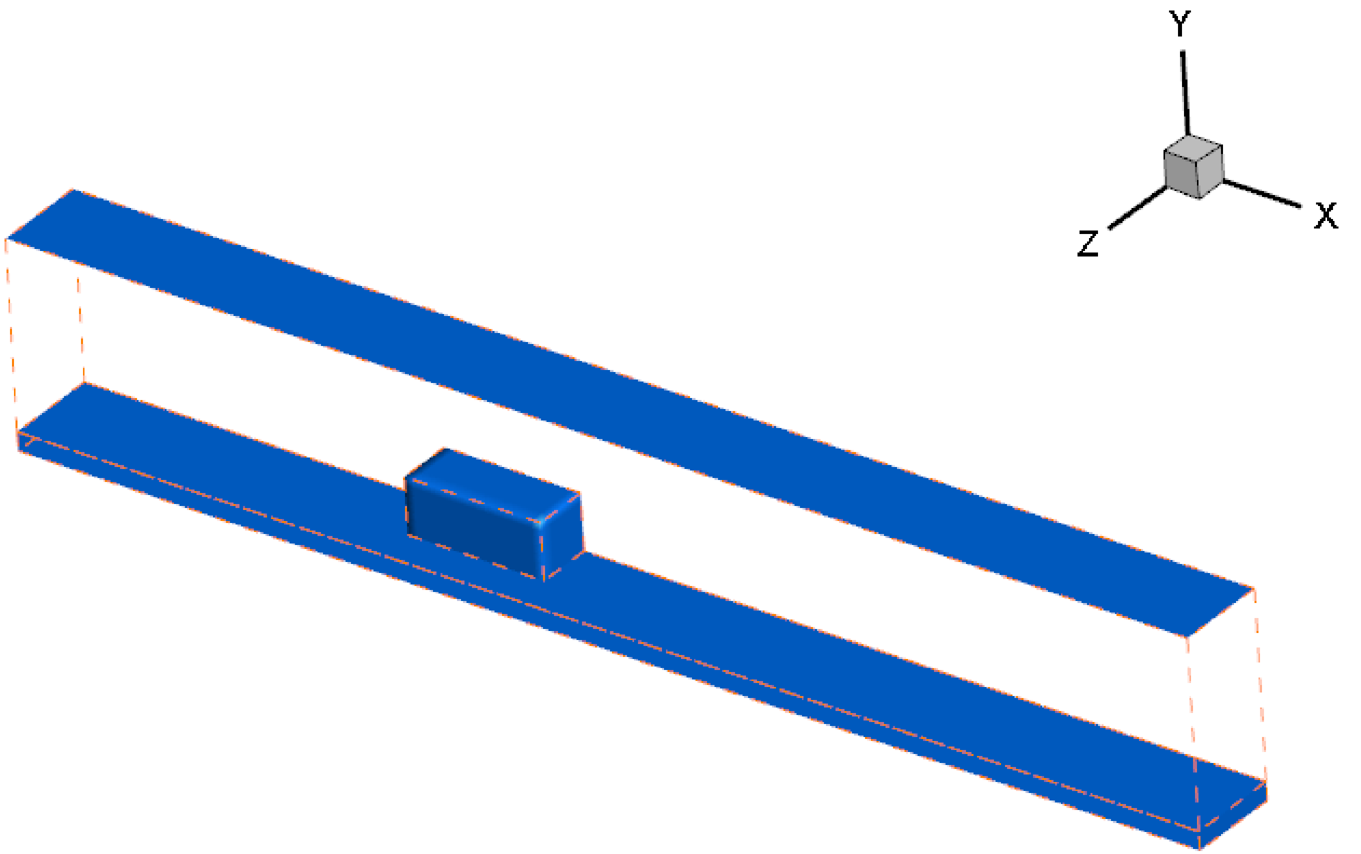
Step 6: Define the boundary condition

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*****.**

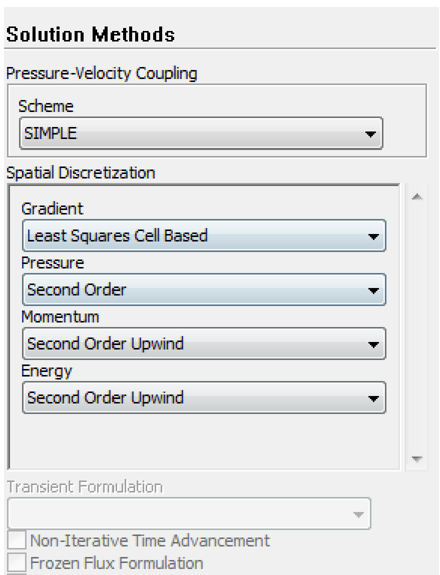


There are many two-sided-wall in this Example.



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.

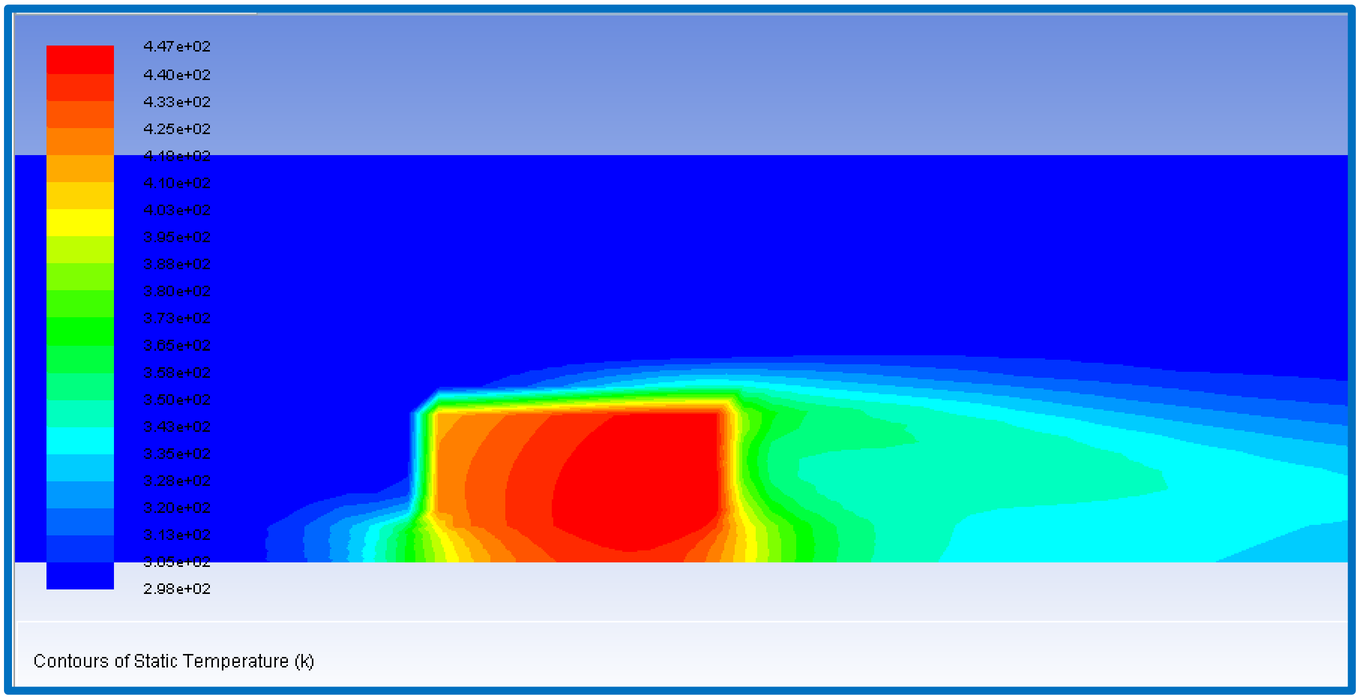
8st step: Initialization

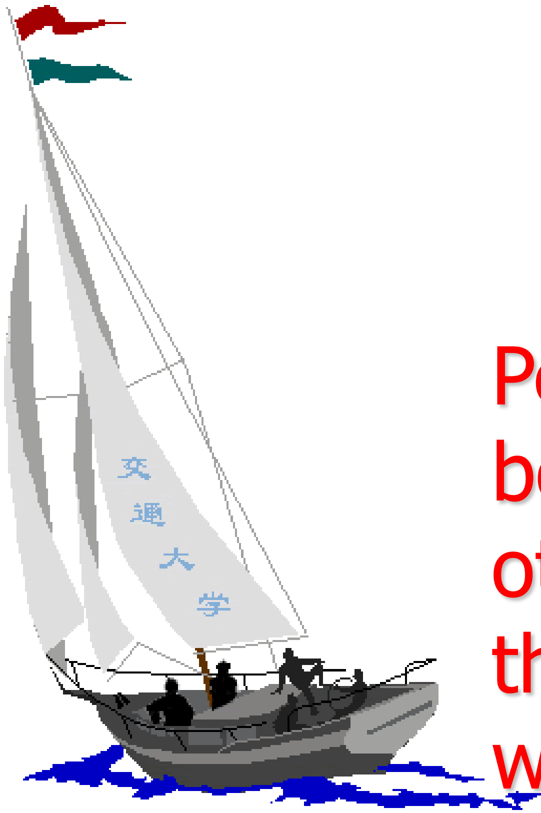
Use the standard initialization, for more details of Hybrid initialization, refer to Example 1.

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….