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, 2021-Dec.-28

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

2021年12月28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 Conductive heat transfer in a heat sink

13.2 Unsteady cooling process of a steel ball

13.3 Flow and heat transfer in a micro-channel

13.4 Flow and heat transfer in chip cooling

13.5 Liquid cooling of photovoltaic panel

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 散热装置导热问题

13.2 非稳态圆球冷却问题

导热问题

13.3 微通道内流动换热问题

13.4 芯片冷却流动换热问题

13.5 太阳能光伏板水冷问题

对流传热问题

13.6 肋片强化相变材料融化

相变传热

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

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

1. Read mesh
2. Scale domain
3. Choose model
4. Define material
5. Define zone condition
6. Define boundary condition
7. Solution
8. Initialization
9. Run the simulation.
10. Post-processing

13.4 Flow and heat transfer in a micro-channel

微通道内流动换热问题

Focus: **compared with previous examples, the focus of this example is about pressure-out boundary condition and 'two-side-wall" boundary condition.**

13.1 Single-phase fluid flow and heat transfer in micro channel with rectangular ribs (MC-RR)

Known: Cold water at T_f=20°C flows into the inlet of a MC-RR with velocity u=0.1m/s. The side wall of MC-RR is heated with a uniform heat flux q = 30W/cm^2.

Assumption: (1) steady state, (2) laminar flow, (3) incompressible fluid, (4) constant fluid properties, (5) negligible radioactive heat transfer from the micro channel heat sink.

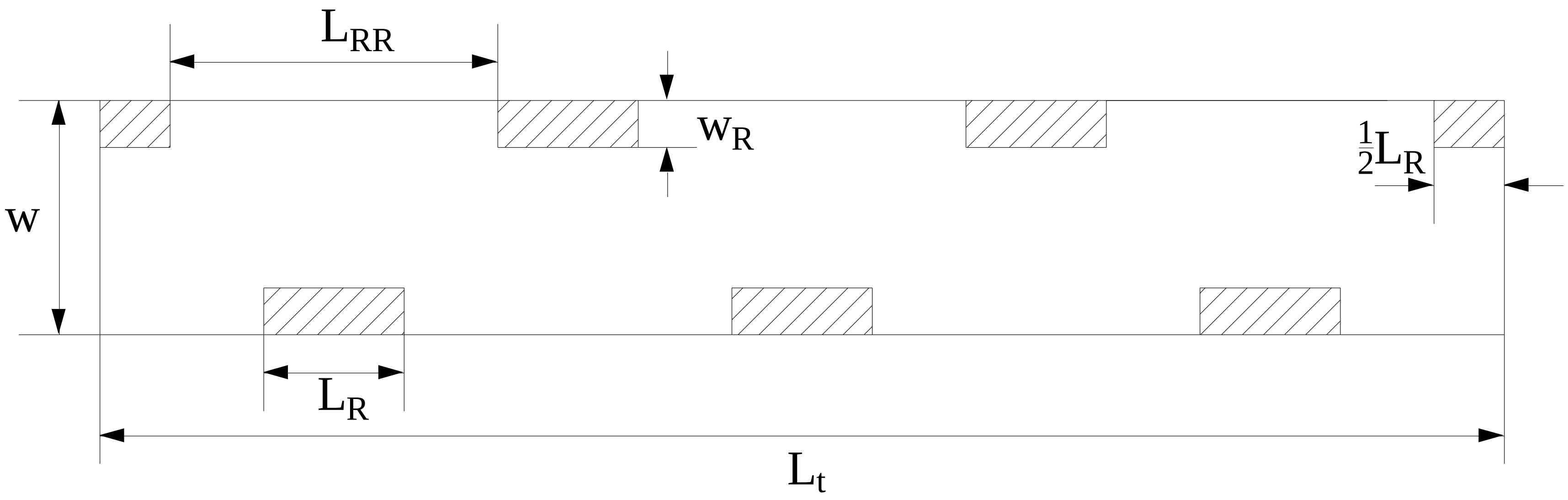


Fig.1 Computational domain

Table .1 Geometrical parameters of MC-RR

Geometrical Parameters	W	L_{RR}	W_R	L_R	L_t
Value/mm	0.5	0.7	0.1	0.3	3

Find: Temperature distribution and pressure distribution in the domain.

Governing equations:

Continuity equation:

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

Momentum equations:

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

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

Energy equation:

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

where T_f is the coolant's temperature, c_{Pf} is fluid specific heat, and k_f is fluid thermal conductivity.

Energy equation for the solid region:

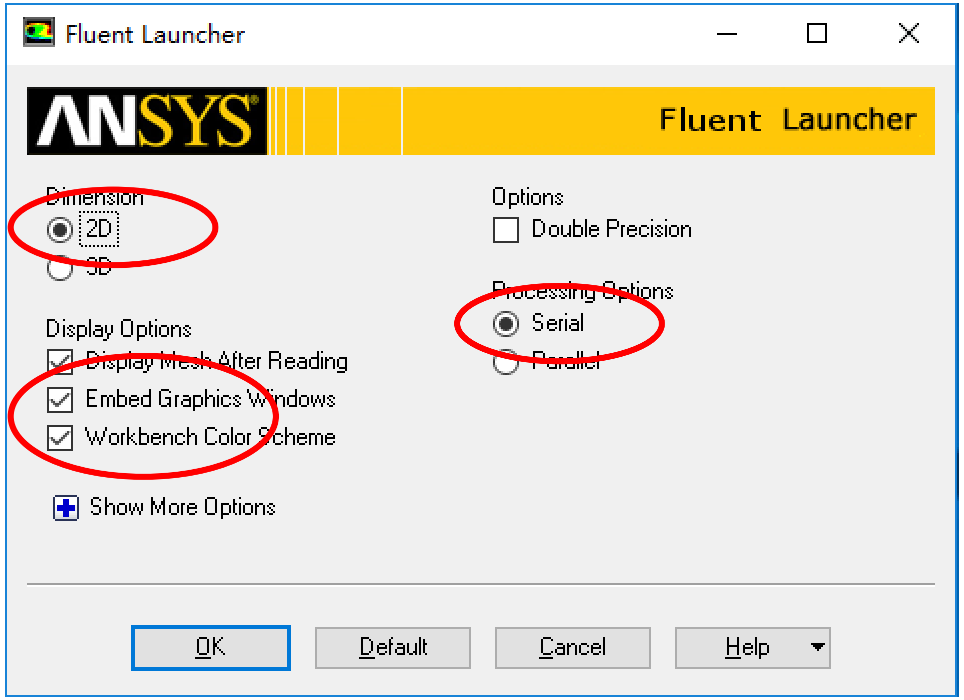
$$0=k_s\left(\frac{\partial^2 T_s}{\partial x^2}+\frac{\partial^2 T_s}{\partial y^2}\right)$$

where T_s is solid temperature and k_s is solid thermal conductivity

Boundary condition:

1.		channel inlet $x=0$ $u=u_f$ For fluid $T_f=T_{in}=293.15\text{K}$
2.		Channel outlet $x=3\text{mm}$ $P_f=P_{out}=1\text{atm}$ For fluid $-k_f(\frac{\partial T_f}{\partial x})=0$
3.	$T_s=T_f$	fluid/solid surface $u=v=0$ $-k_s(\frac{\partial T_s}{\partial n})=-k_f(\frac{\partial T_f}{\partial n})$ Where (n) is the coordinate normal to the wall
4.		At side wall $-k_s(\frac{\partial T_s}{\partial y})=q=30\text{W/cm}^2$

Start the Fluent software



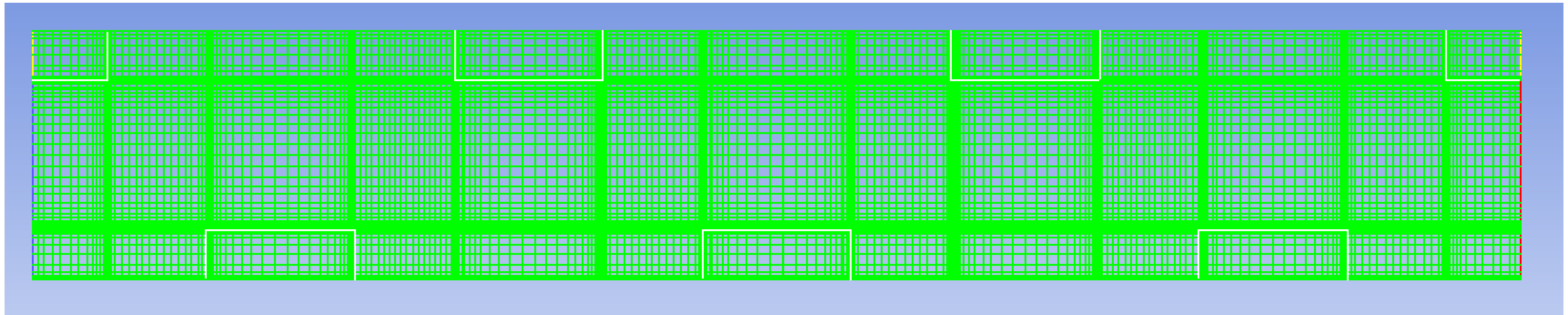
1. Choose 2-Dimension
2. Choose display options
3. Choose Serial processing option

Note: Double precision or Single precision

For most cases the single precision version of Fluent is sufficient. For heat transfer problem, if the thermal conductivity difference between various components are high, it is recommended to use Double Precision Version.

Step 1: Read and check the mesh

- The mesh is generated by pre-processing software such as ICEM, GAMBIT or MESHING. The document is with suffix (后缀名） “xx.msh”



```
> Reading "C:\Users\lichennht\Desktop\ansys\4fluent_
Done.
   14340 quadrilateral cells, zone 15, binary.
    2124 quadrilateral cells, zone 16, binary.
   28270 2D interior faces, zone 17, binary.
    3987 2D interior faces, zone 18, binary.
      44 2D velocity-inlet faces, zone 19, binary.
      44 2D pressure-outlet faces, zone 20, binary.
     177 2D wall faces, zone 21, binary.
     321 2D wall faces, zone 22, binary.
```

```
     411 2D wall faces, zone 23, binary.
      24 2D symmetry faces, zone 24, binary.
     321 shadow face pairs, binary.
   17129 nodes, binary.
   17129 node flags, binary.
```

Step 1: Read and check the mesh

Mesh→Check

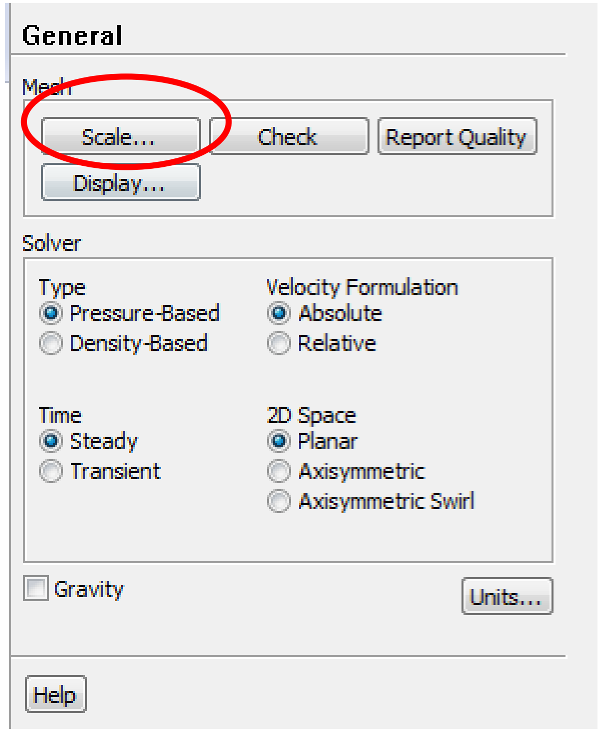
- Check the quality and topological information of the mesh

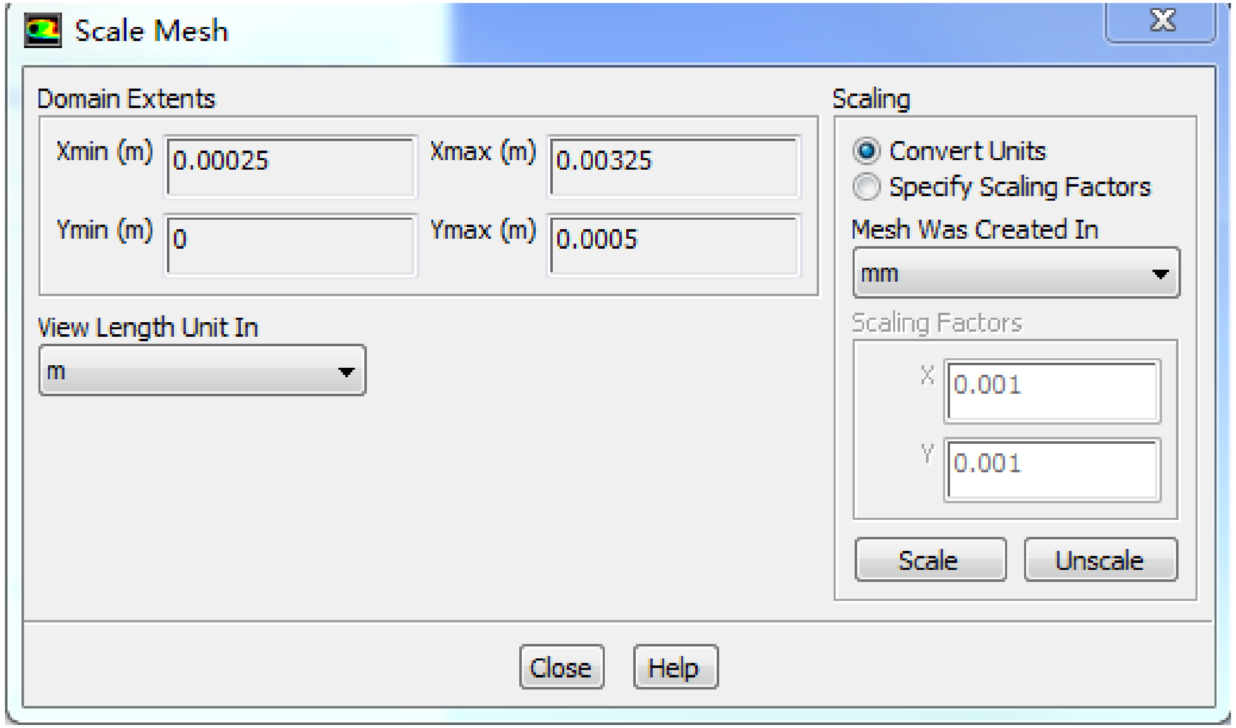
```
Mesh Check

 Domain Extents:
   x-coordinate: min (m) = 2.500000e-04, max (m) = 3.250000e-03
   y-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-04
 Volume statistics:
   minimum volume (m3): 9.997533e-13
   maximum volume (m3): 5.455531e-10
     total volume (m3): 1.500000e-06
 Face area statistics:
   minimum face area (m2): 9.997748e-07
   maximum face area (m2): 2.495997e-05
 Checking mesh.............................
Done.
```

Step 2: Scale the domain size

General→Scale

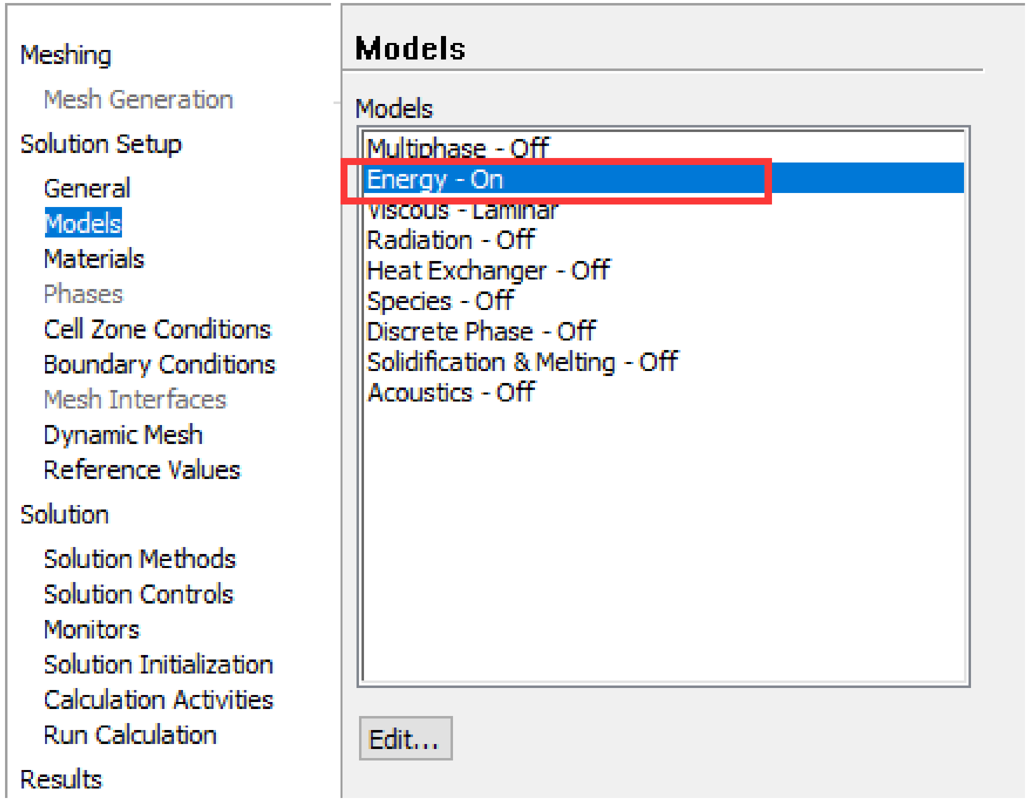




The mesh is generated in Fluent using unit of mm. Fluent import it as unit of m. Thus, "Convert units" is used.

Step 3: Choose the physicochemical model

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



Example 4 is about single-phase laminar flow and heat transfer, so Energy model should be activated.

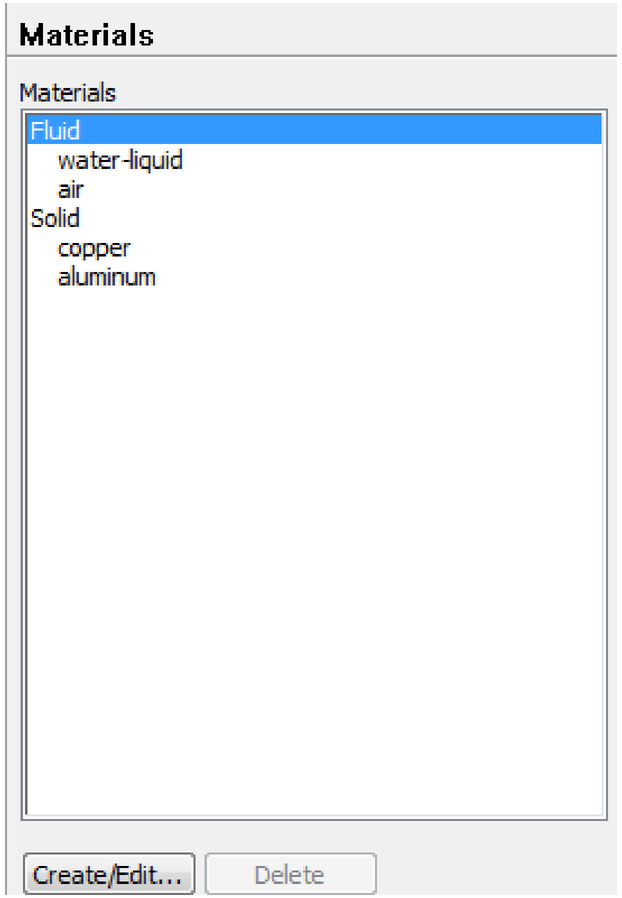
Step 4: Define the material properties

Define the material properties required for modeling! For heat conduction problem studied here, ρ, Cp and λ should be defined.

Solution Setup→Materials

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

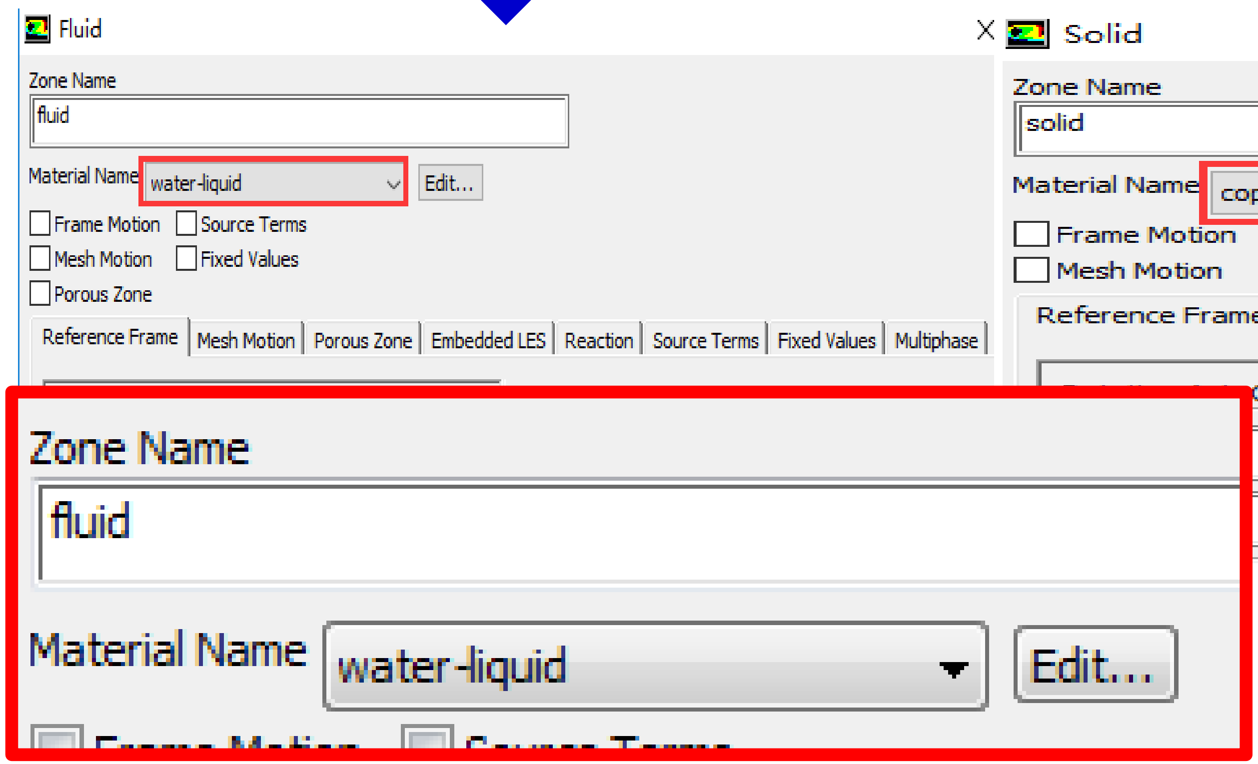
Click the Create/Edit button to add Water and Copper.

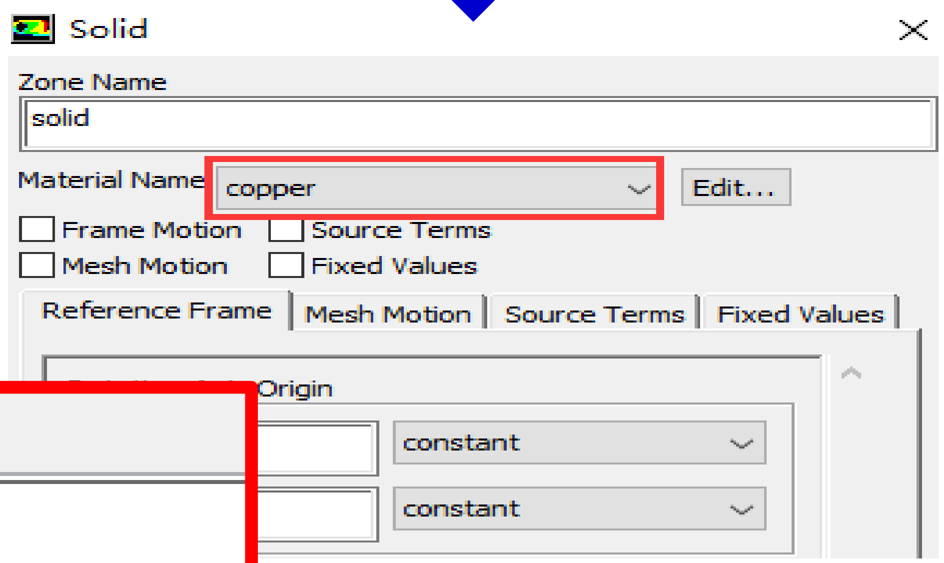


Step 5: Define zone condition

Solution Setup→Cell Zone Condition

Choose water for Fluid zone Choose copper for Solid zone

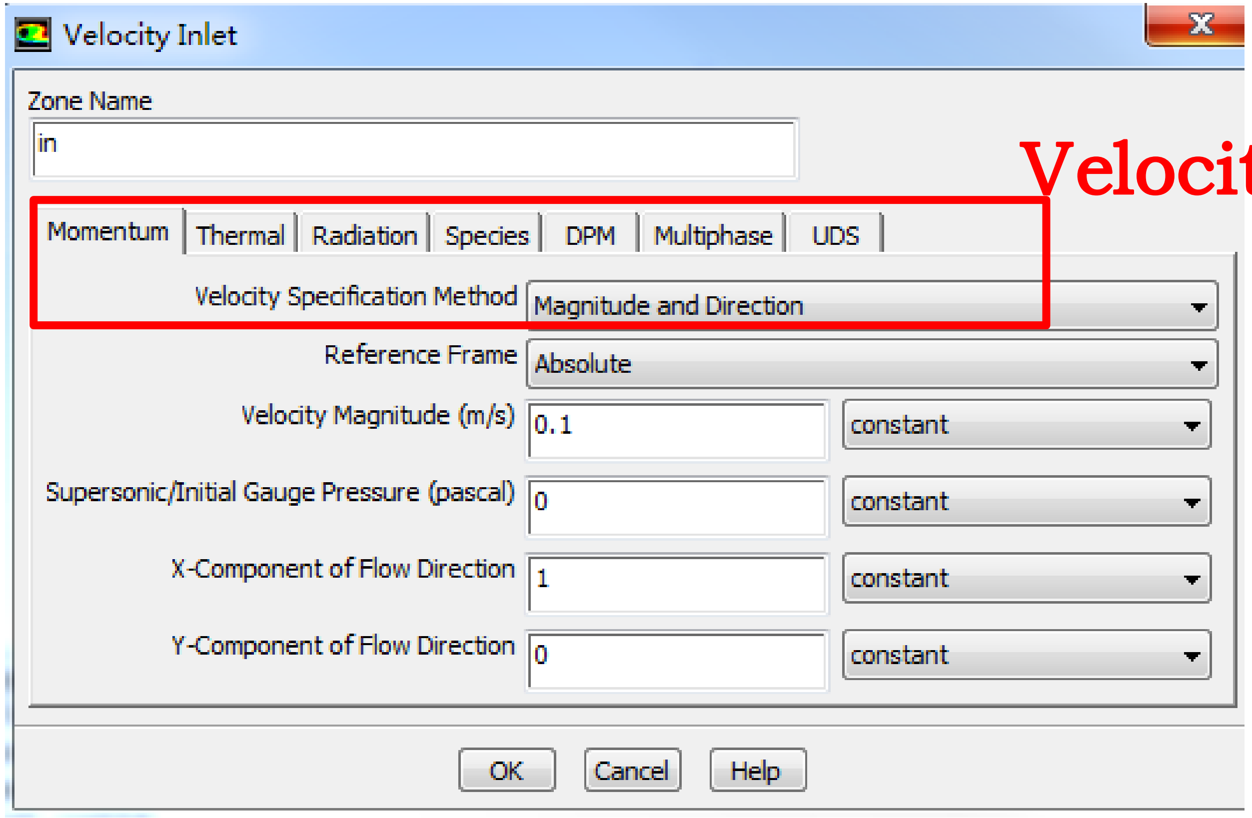




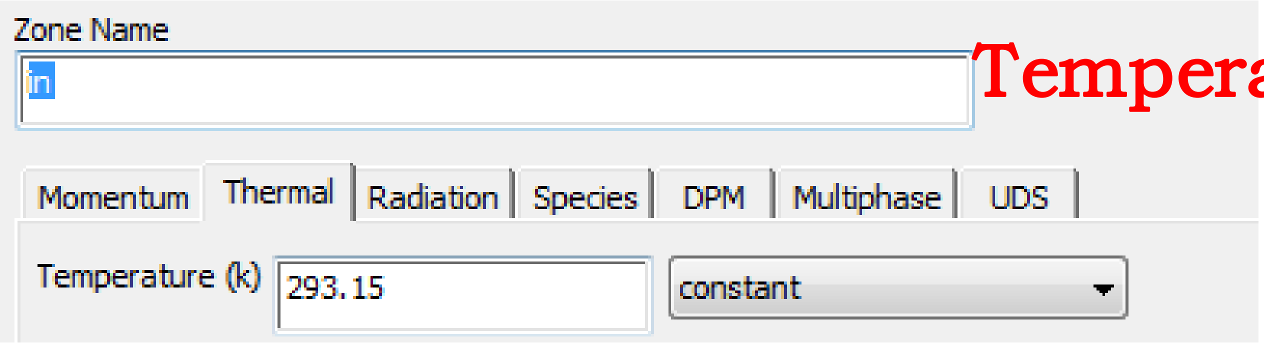
Step 6: Define the boundary condition

Inlet

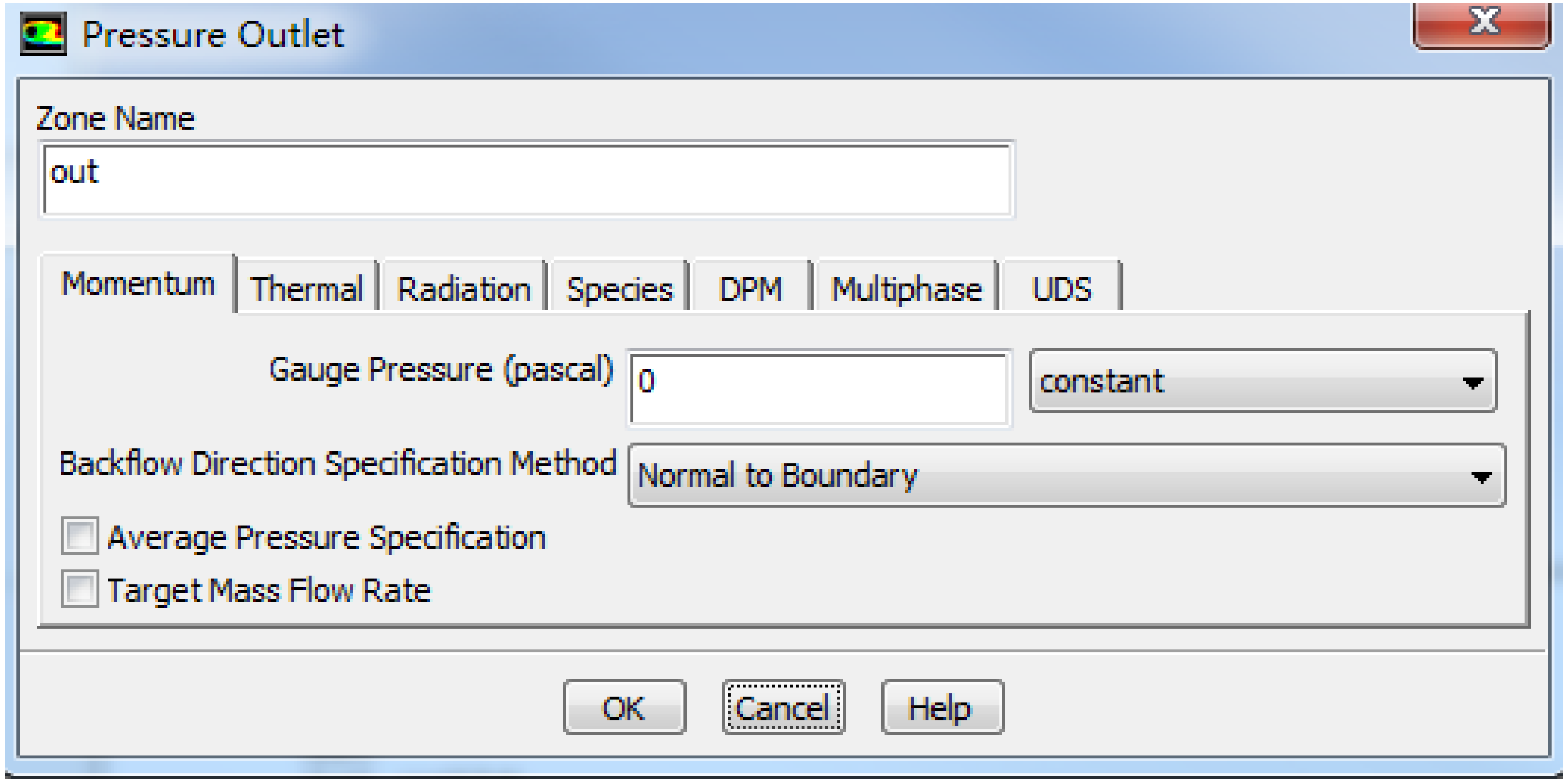
Velocity



Temperature



Outlet: pressure outlet



Gauge Pressure (表压)

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 $0.5\rho U^2$

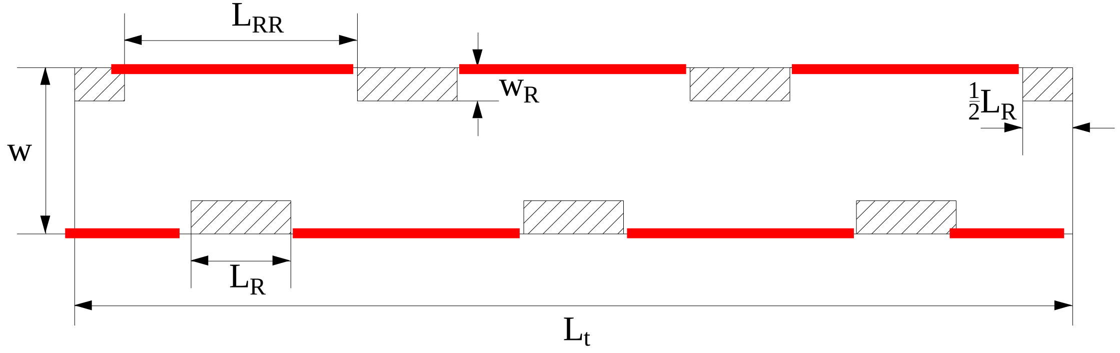
Is related to the velocity.

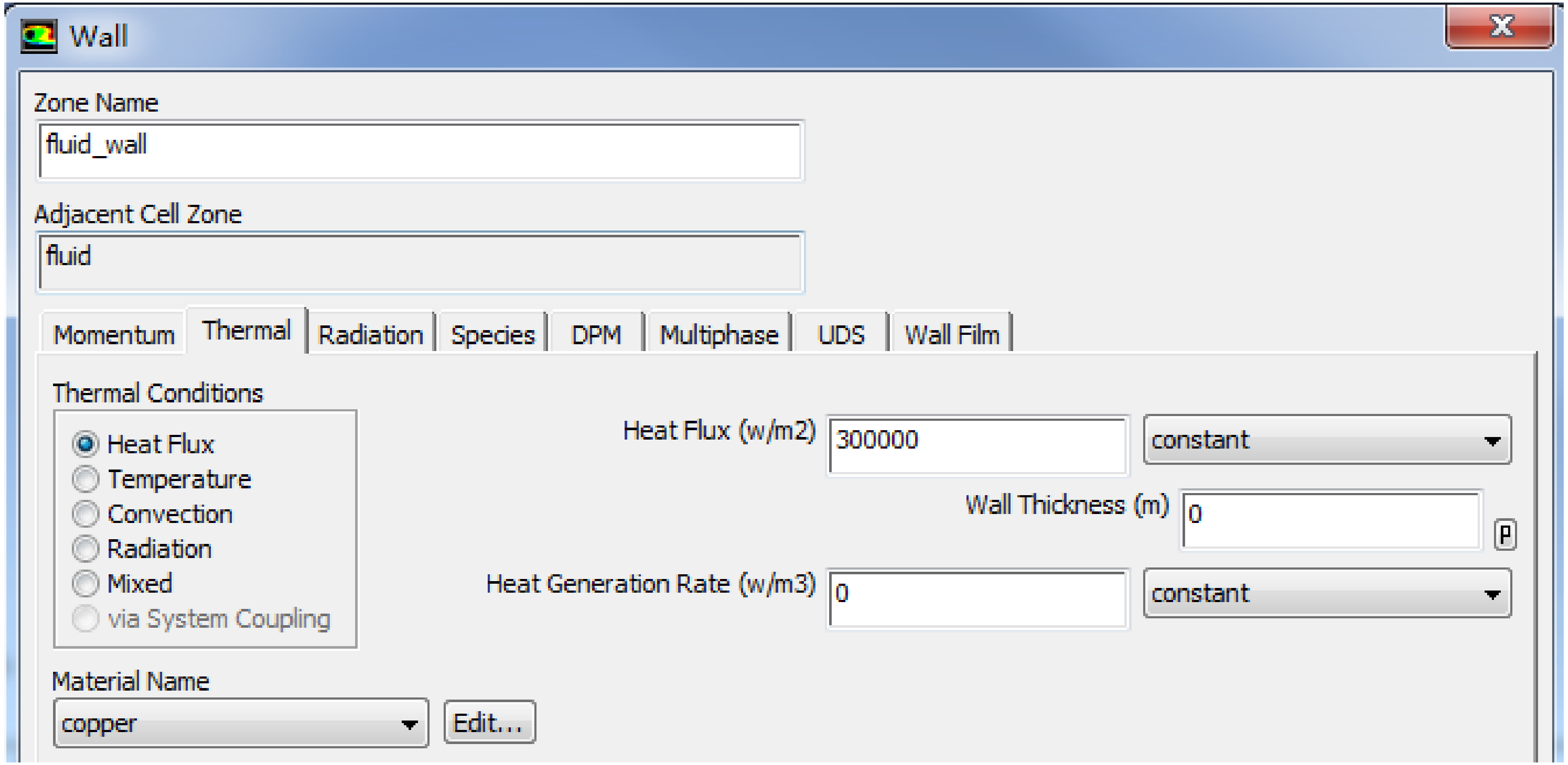
Total pressure (动压):

= Static pressure + dynamic pressure

Wall

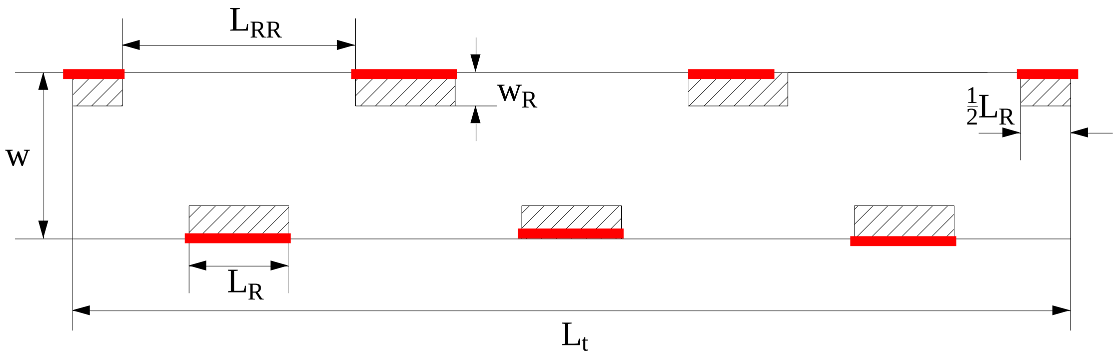
$$q = 30\text{W/cm}^2$$

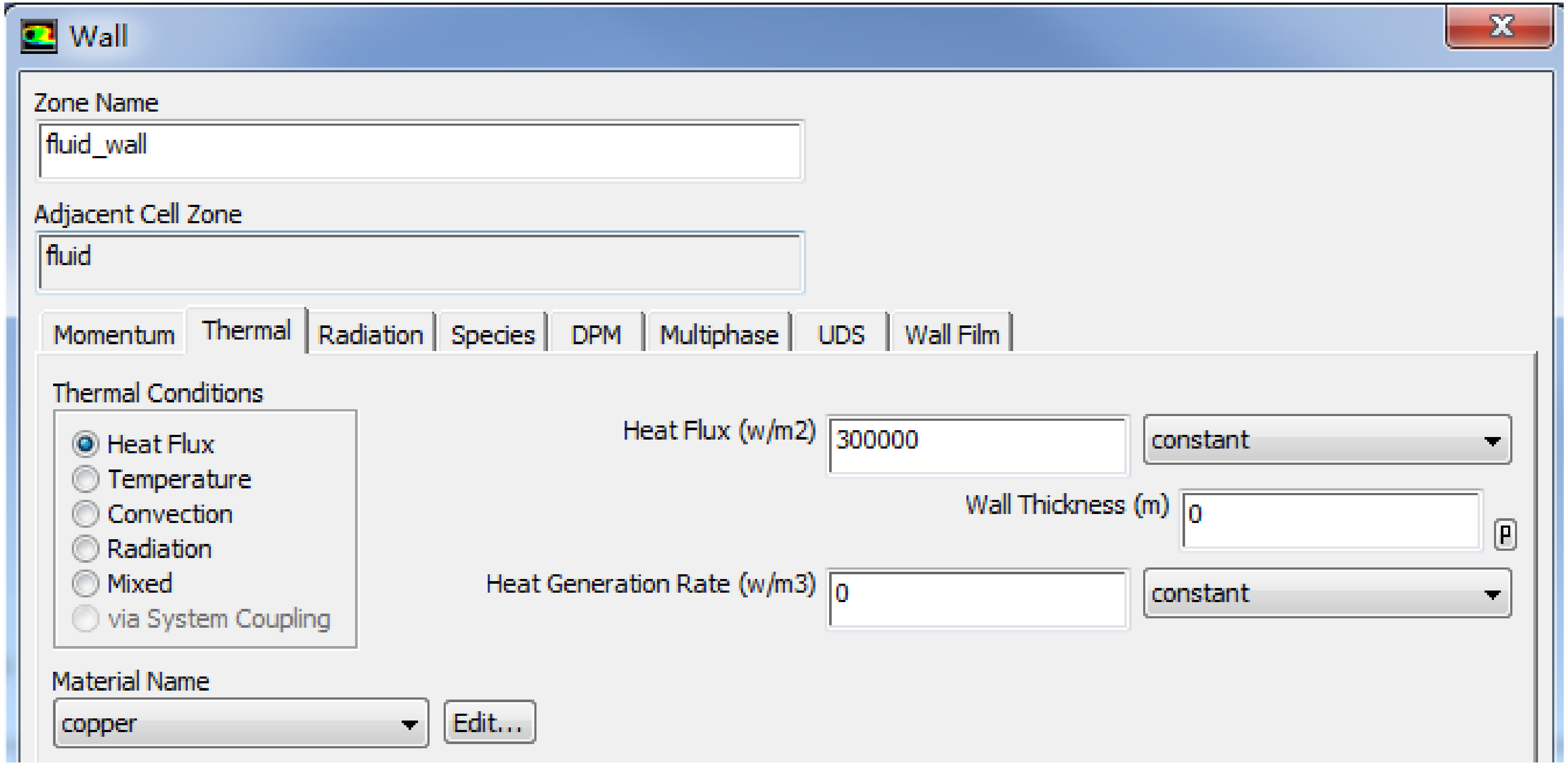




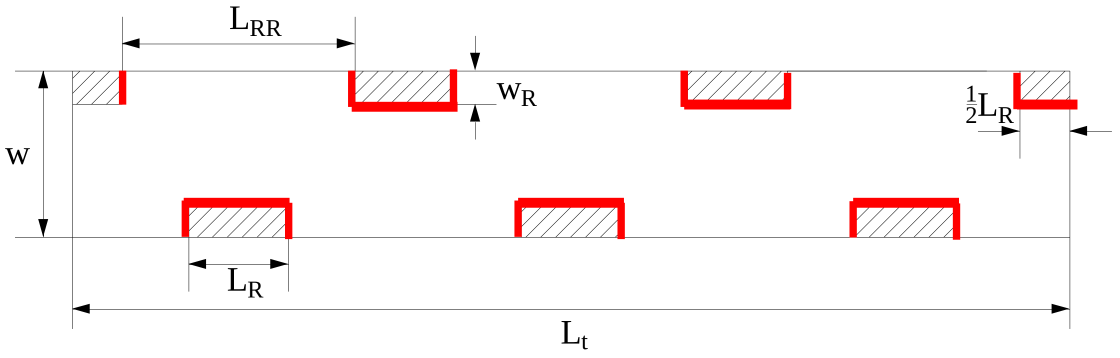
Wall

$$q = 30\mathrm{W/cm}^2$$





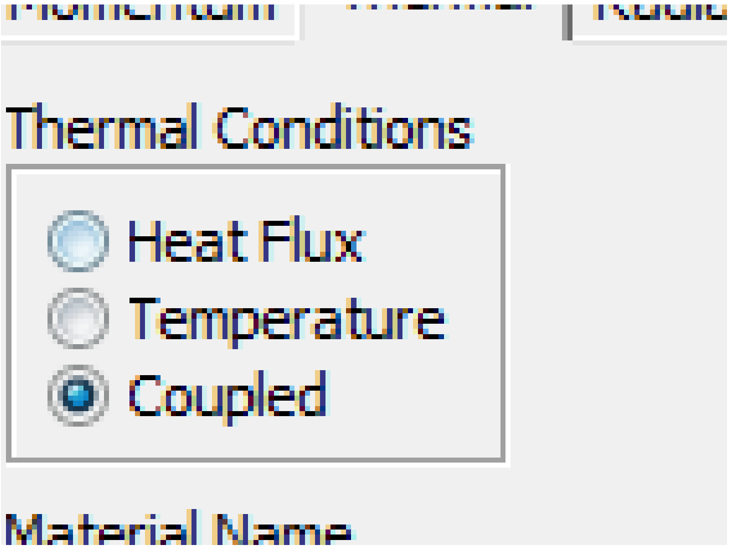
Fluid-solid interface



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.

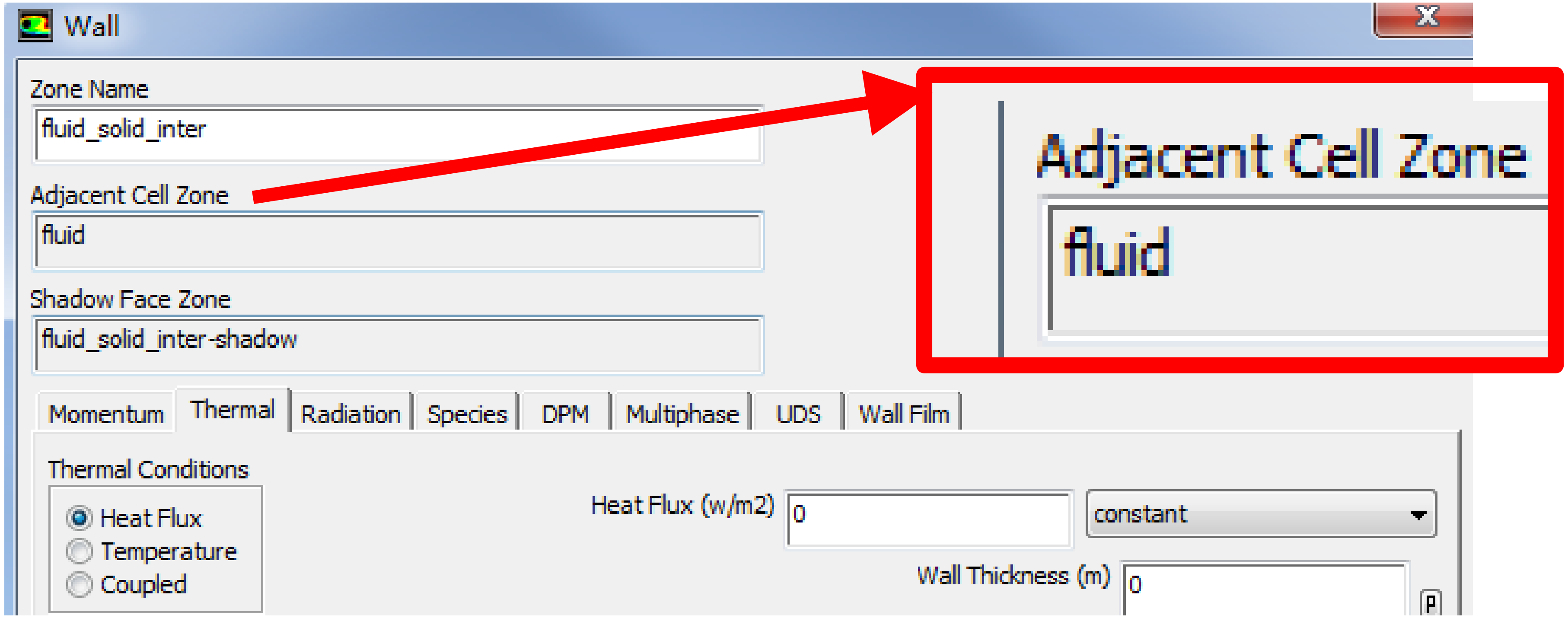
There are three options for the temperature boundary conditions of such "two-sided-wall".



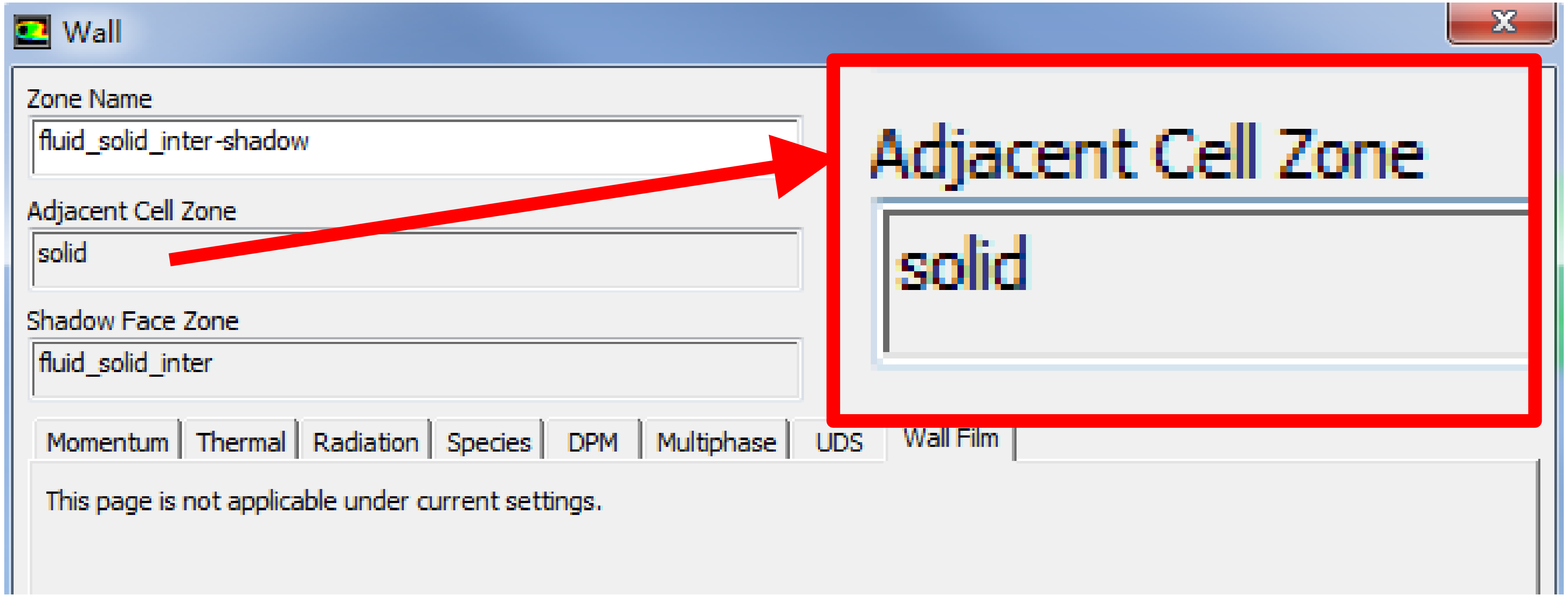
- Heat flux
- Temperature
- Coupled

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.

You also can uncouple the sides of the wall and give different boundary condition for different sides of the wall.



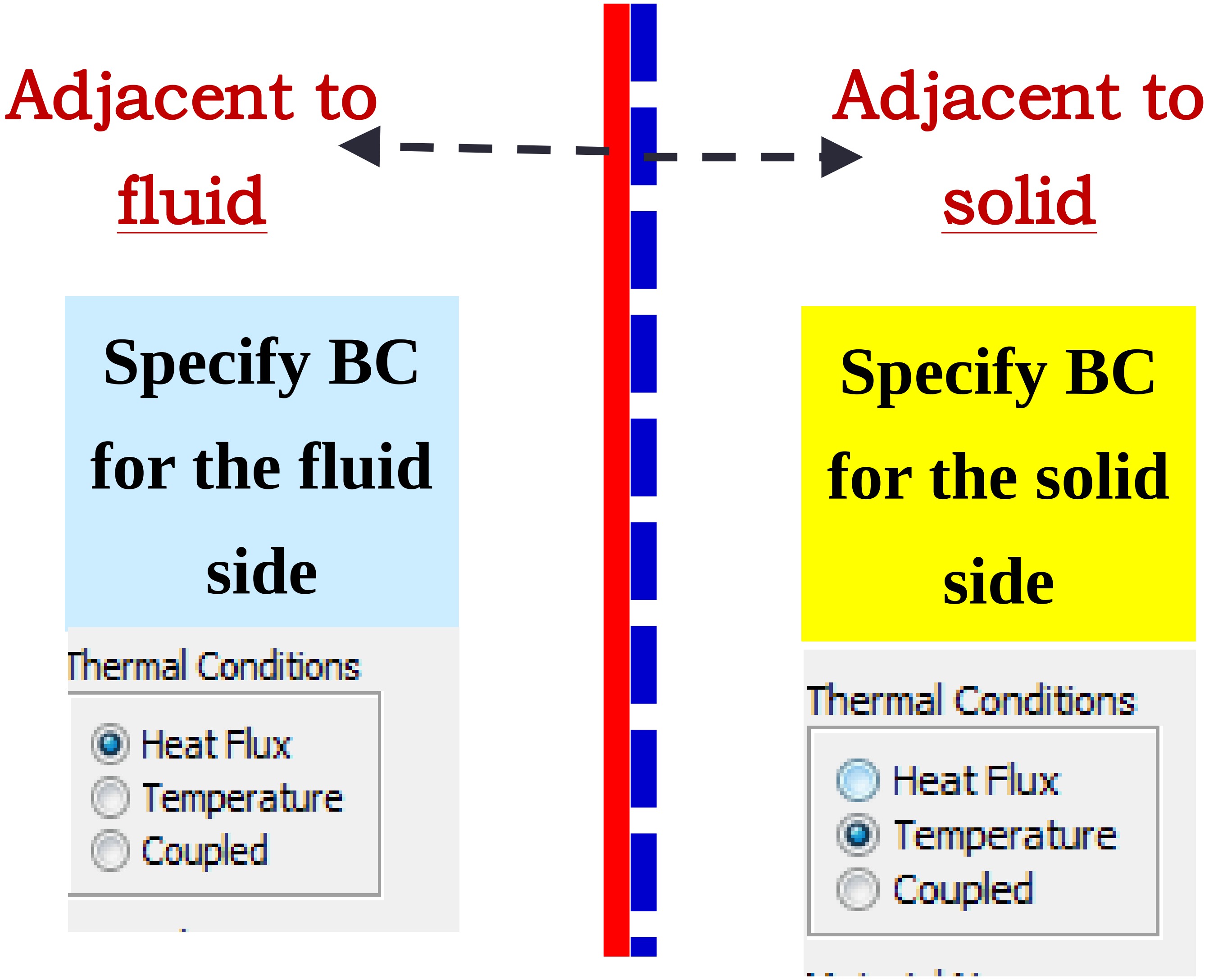
The adjacent cell zone of this wall is fluid!



The adjacent cell zone of this shadow wall is solid!

You can find the wall and its shadow created automatically by Fluent are adjacent to fluid and solid, respectively. So you can specify different BC for different walls.

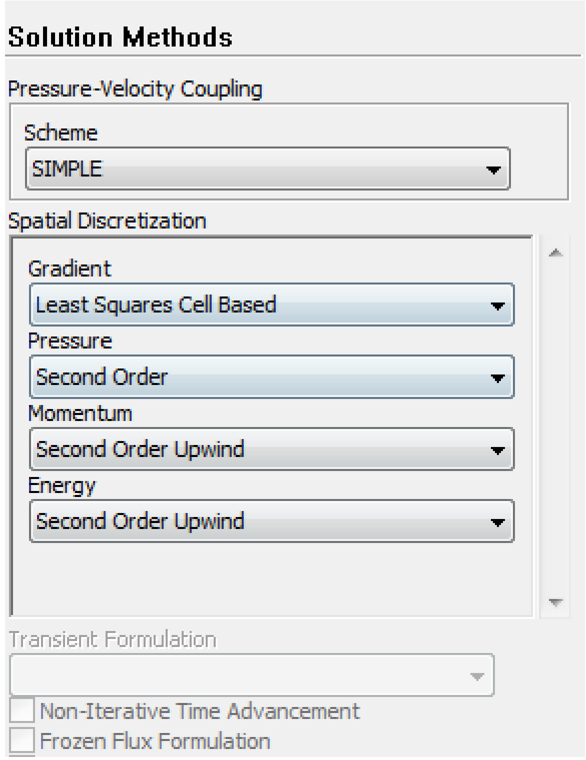
The original two side wall



Its shadow created by Fluent

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.

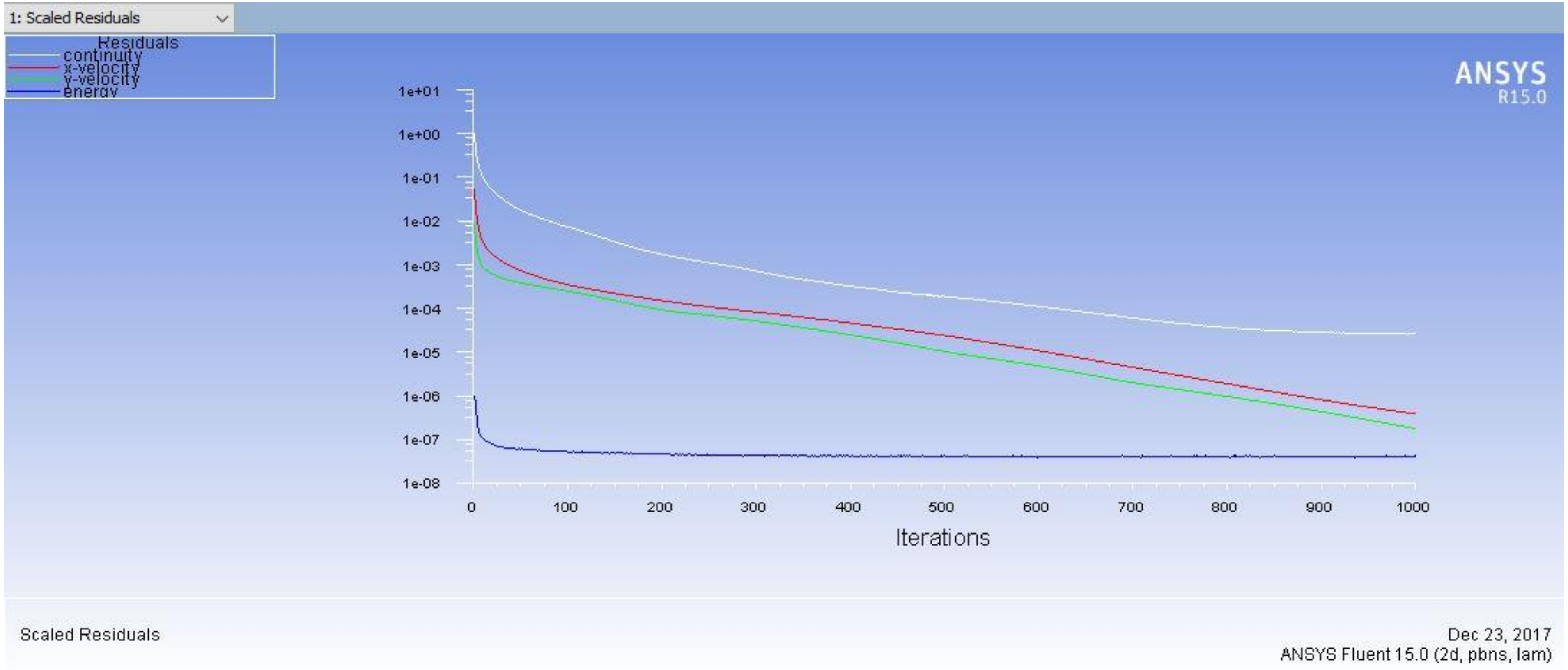
8st step: Initialization

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

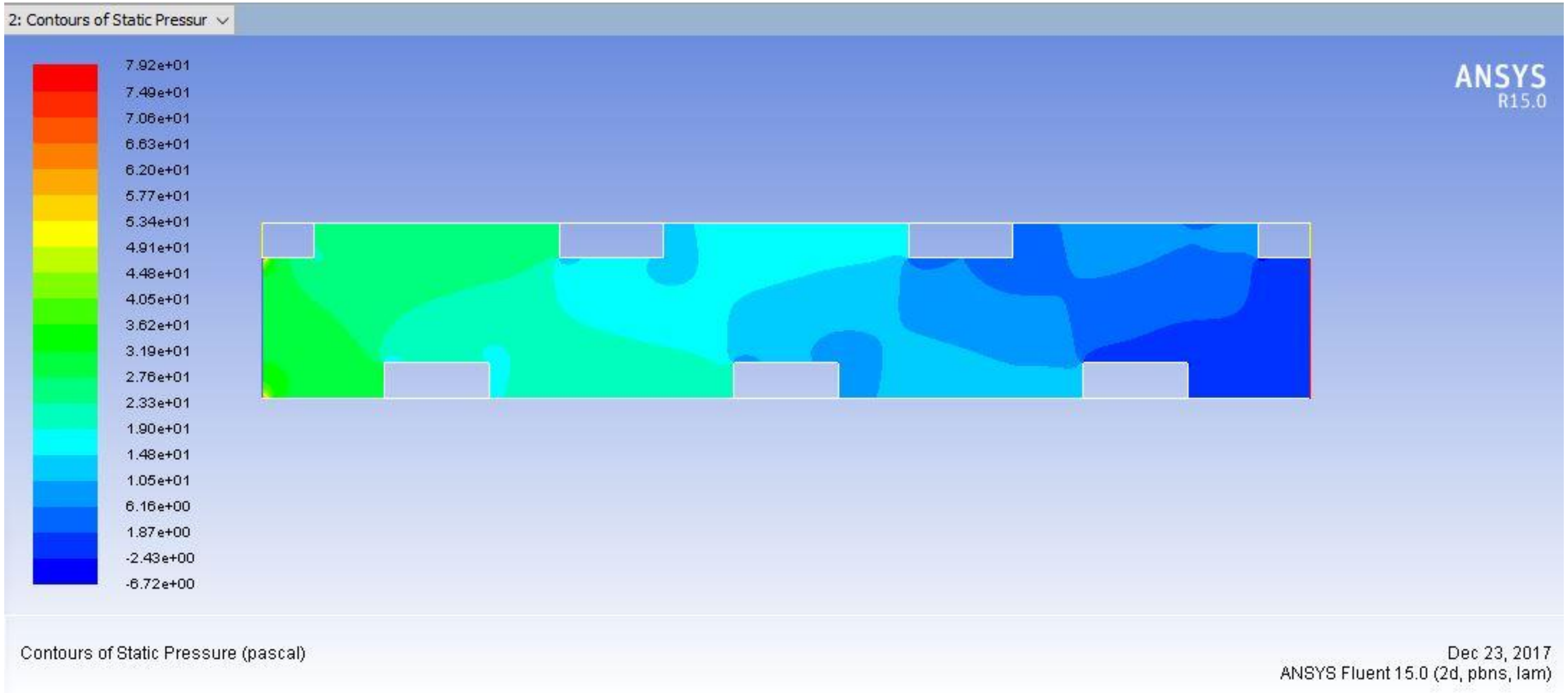
Step 9: Run the simulation

Step 10: Post-processing results

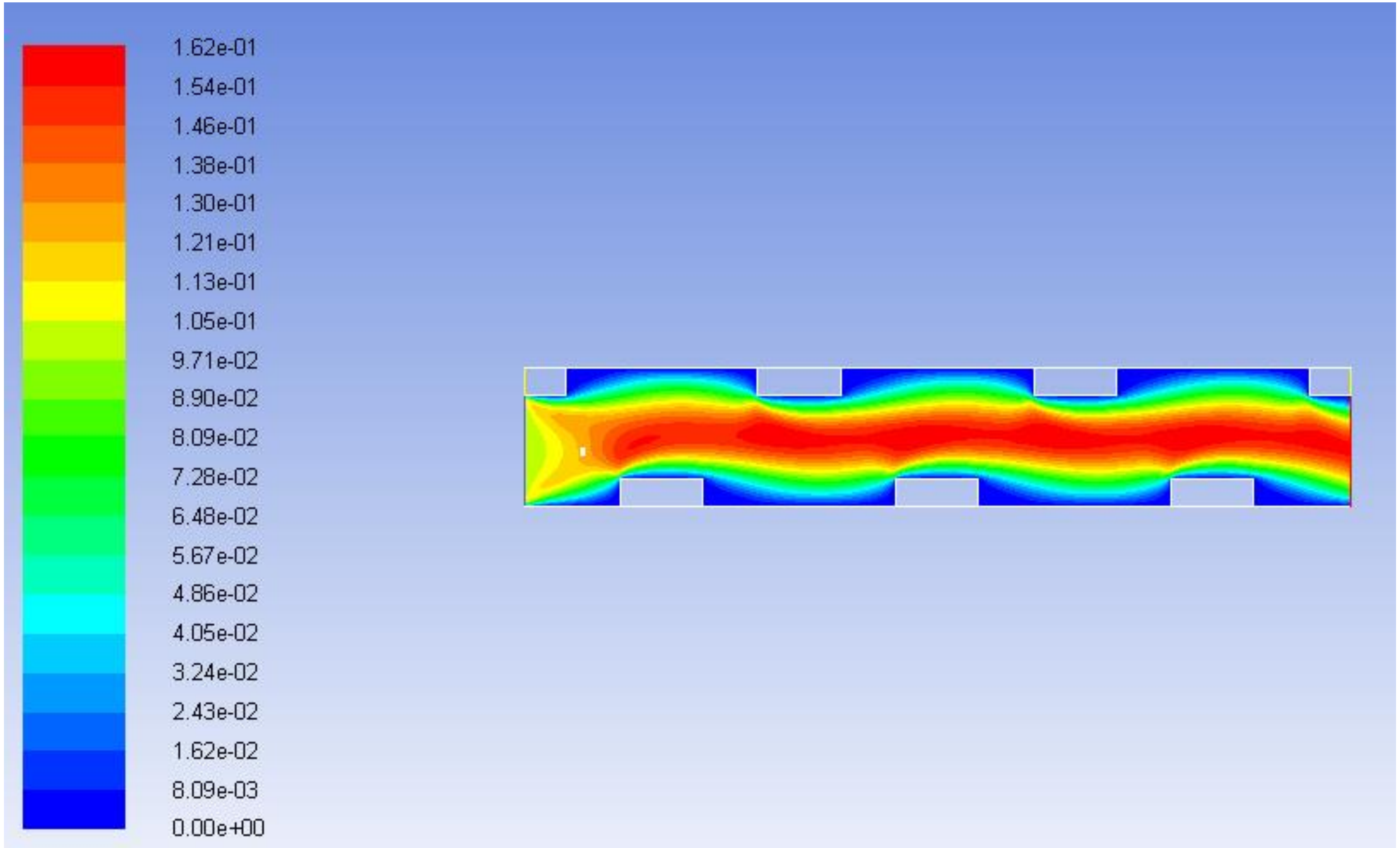
Residuals


1: Scaled Residuals
Residuals
continuity
x-velocity
y-velocity
energy
ANSYS
R15.0
1e+01
1e+00
1e-01
1e-02
1e-03
1e-04
1e-05
1e-06
1e-07
1e-08
0
100
200
300
400
500
600
700
800
900
1000
Iterations
Scaled Residuals
Dec 23, 2017
ANSYS Fluent 15.0 (2d, pbns, lam)

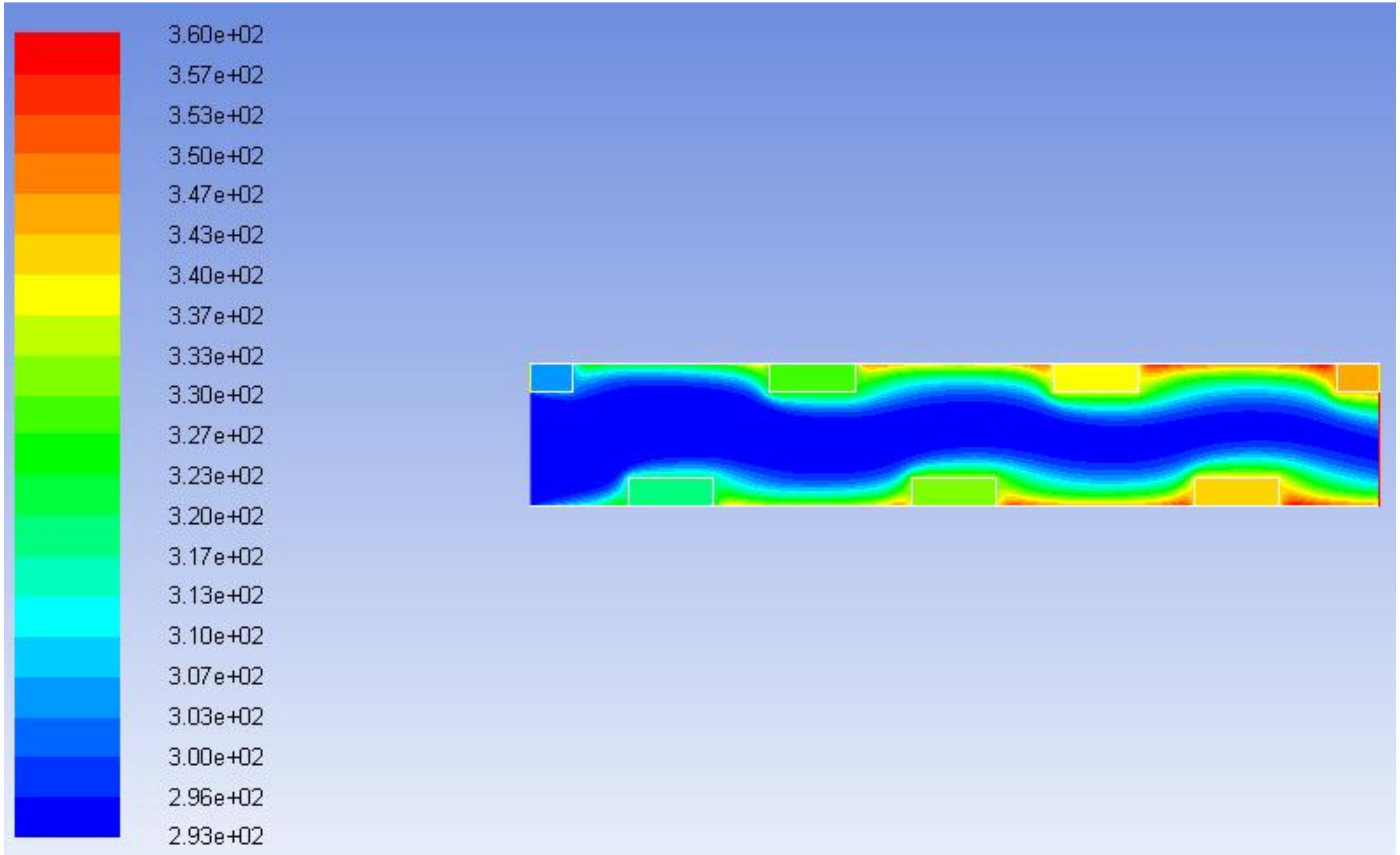
Contours of static pressure (Pa)



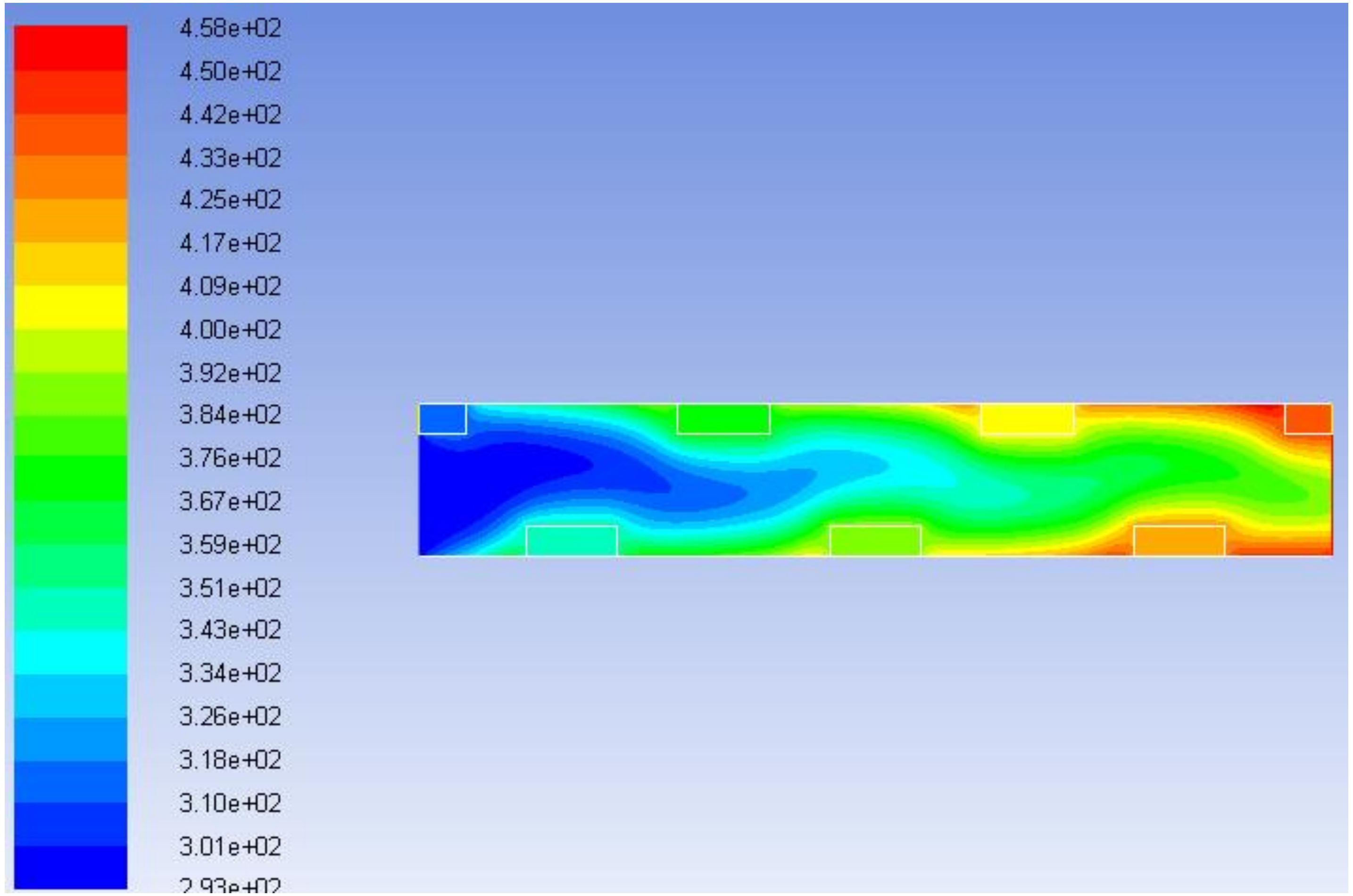
Velocity magnitude



Temperature (K)



Temperature (K) of velocity as 0. 01 m/s



同舟共济
渡彼岸！

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