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.-25

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20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

13.2 非稳态圆球冷却问题

13.3 顶盖驱动流动换热问题

13.4 微通道内流动换热问题

13.5 芯片冷却流动换热问题

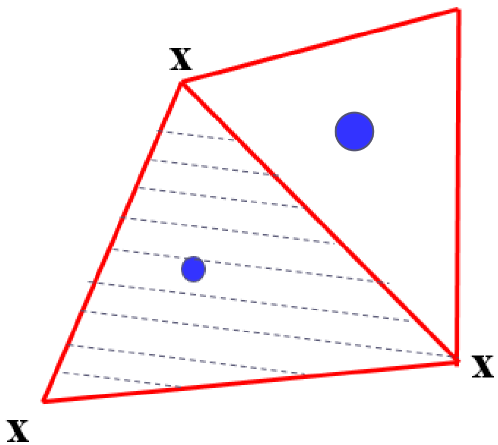
13.6 多孔介质流动换热问题

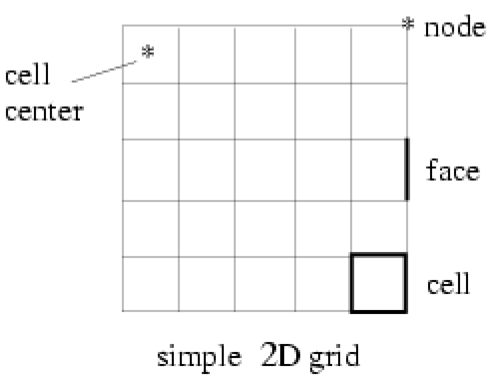
13.7 气膜冷却流动换热换热问题

Review

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.

Remark: Difference between the terminology in our NHT (Practice B) and Fluent software about the mesh information.



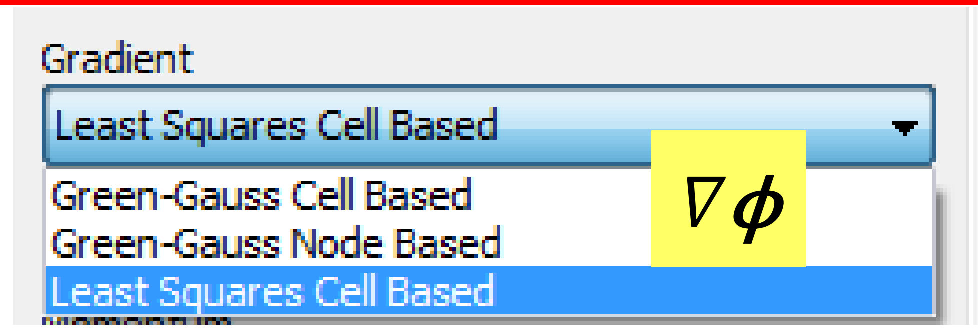


	Our NHT	Fluent
	Node/cell center	Cell center
x		Node
	Interface	Interior face

Interface in Fluent is particularly used for the face between different materials.

Gradient calculation,

There are three schemes.



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:

$$<\nabla\phi>=\frac{1}{V_C}\int_{V_C}\nabla\phi dV$$

The problem of calculating gradient is transferred into the following equation:

How to determine ϕ_f at the face?

Least-Squares Cell Based 基于单元体的最小二乘法

It is the default scheme for gradient calculation.

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

Pressure calculation： to calculate the pressure value at the interface using centroid value.

1. Linear scheme
2. Standard scheme
3. Second Order
4. Body Force Weighted scheme
5. PRESTO

The difference between Hybrid initialization (混合初始化) and Standard initialization.

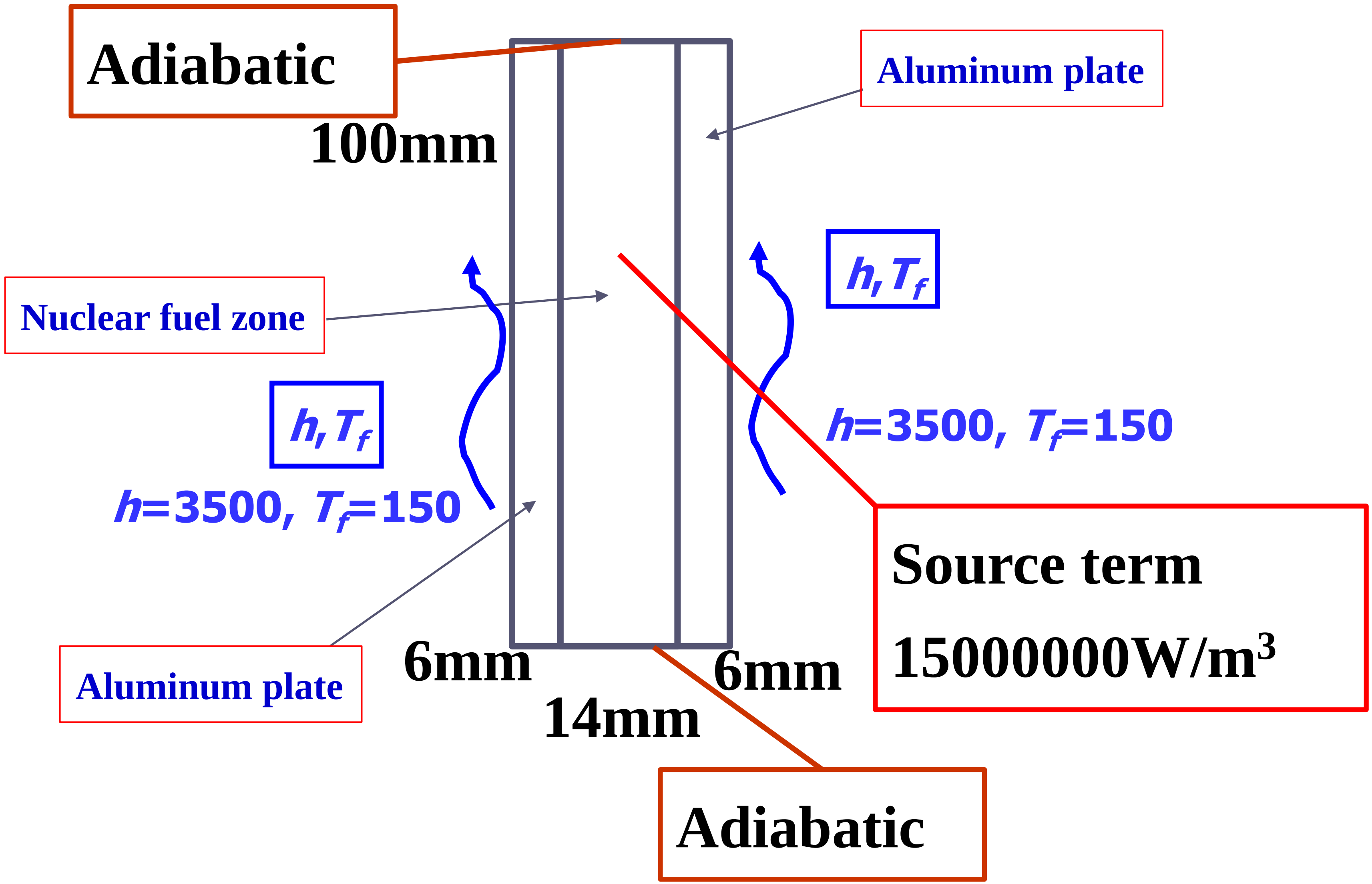


Fig.1 Computational domain

13.2 Unsteady cooling process of a steel ball

非稳态圆球冷却问题

Focus: compared with previous example, the focus of this example is about **"unsteady problem"**.

13.2 Unsteady cooling process of a steel ball

Known:

A steel ball with initial uniform temperature of 723 K was placed in air of 303K.

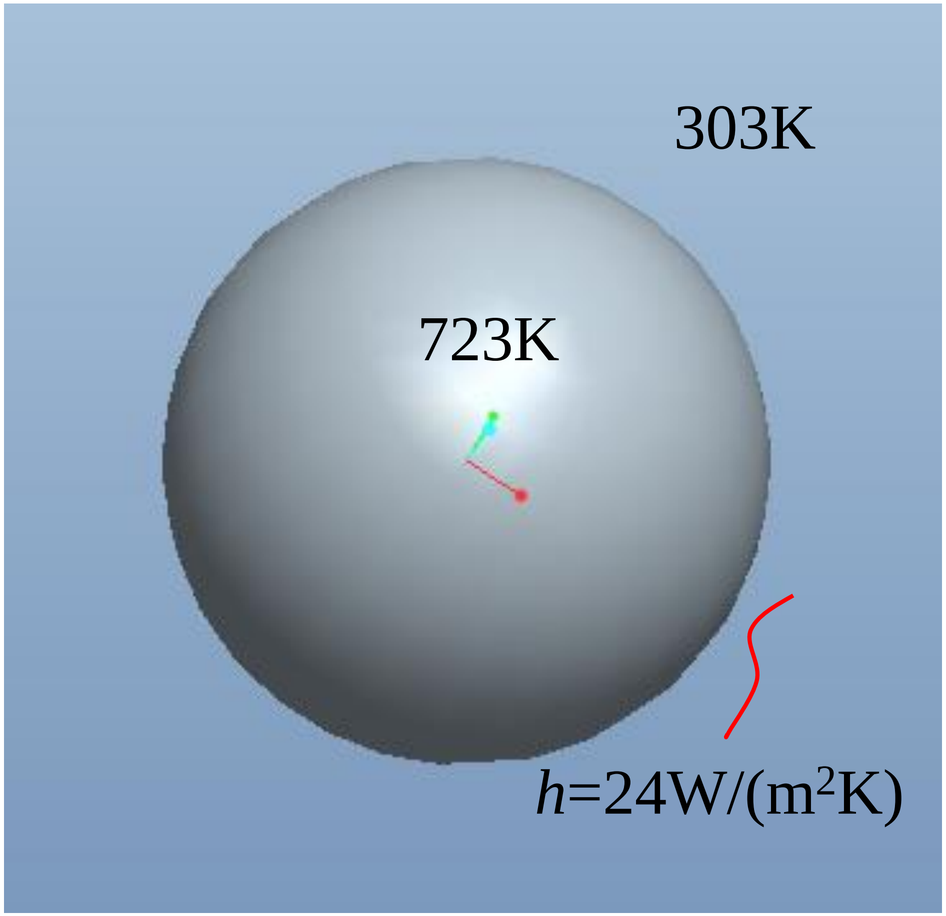
(D=5cm, density is 7735kg/m^3, heat capacity is 480 J/(kg K), conductivity is 33W/(m K)).

- **Outside boundary condition : convective BC**

  fluid temperature: 303K

  Heat transfer coefficient: h=24W/(m^{2}K) .

- **Inside :initial temperature is 723K .**



3rd kind of boundary condition.

h=24, T_f=303

Fig.1 Computational domain

Find: **temperature evolution in the steel ball.**

Solution:

Energy:

$$\frac{\partial\left(\rho C_p T\right)}{\partial t}=div(\Gamma_T gradT)$$

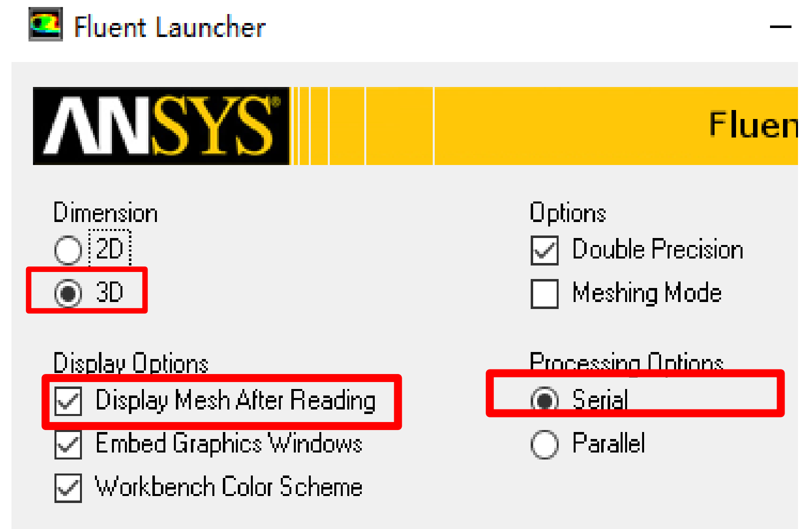
It is a unsteady heat conduction problem with given GAMA.

Remark: here we write the energy governing equation in the improved form with nominal density ρC_p. The improved form is adopted in our general teaching code as well in Fluent.

Start the Fluent software

1、Select 3D dimension as it is a 3D problem.

2、If "display mesh after reading" is selected, after the Fluent is launched, the mesh will automatically shown in the interface.



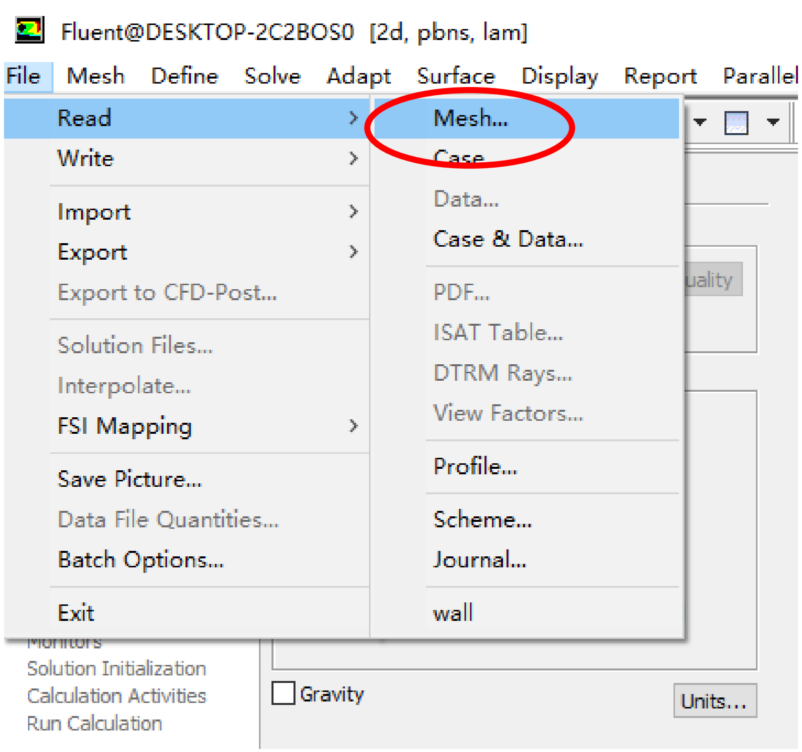
3、For most cases the single precision version of Fluent is sufficient. For heat transfer problem, if the thermal conductivity between different components are high, it is recommended to use Double precision version.

Step 1: Read and check the mesh

- The mesh is generated by pre-processing software such as **ICEM** and **GAMBIT. The document is with suffix (后缀名） "xx.msh"**

- This step is similar to the Grid subroutine (UGRID, Setup1) in our general code.

Mesh→Read

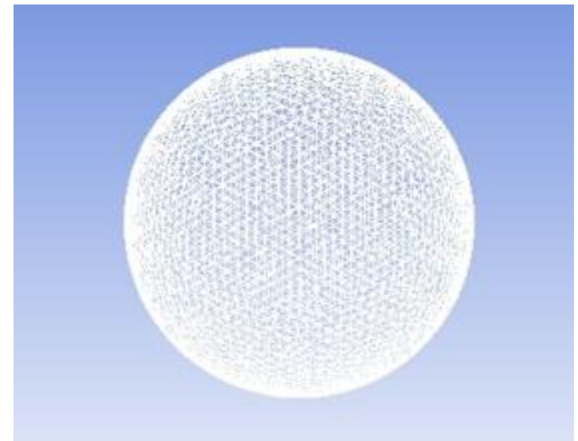



```
> Reading "C:\Users\lichennht\Desktop\陈黎文件管理\a1
Done.
  114545 tetrahedral cells, zone  5, binary.
  225844 triangular interior faces, zone  6, binary.
    6492 triangular wall faces, zone  7, binary.
   20774 nodes, binary.
   20774 node flags, binary.

Building...
     mesh
     materials,
     interface,
     domains,
        mixture
     zones,
        wall
        int_created_material_3
```

Mesh→Check

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

```
Mesh Check

 Domain Extents:
   x-coordinate: min (m) = -2.499196e-02, max (m) = 2.497915e-02
   y-coordinate: min (m) = -2.500000e-02, max (m) = 2.500000e-02
   z-coordinate: min (m) = -2.498061e-02, max (m) = 2.496219e-02
 Volume statistics:
   minimum volume (m3): 1.441216e-10
   maximum volume (m3): 1.394640e-09
     total volume (m3): 6.519246e-05
 Face area statistics:
   minimum face area (m2): 3.881175e-07
   maximum face area (m2): 2.646230e-06
 Checking mesh.........................
Done.
```



- **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
```

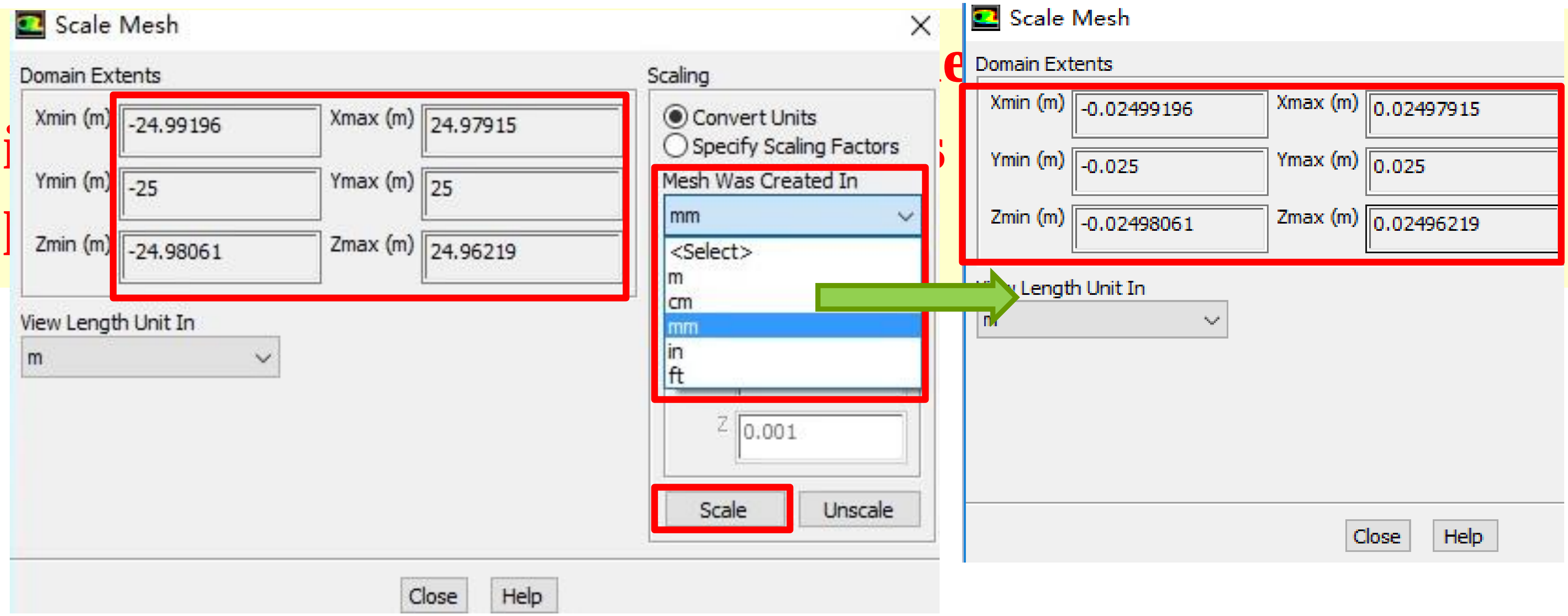
```
WARNING: Mesh check failed.
```

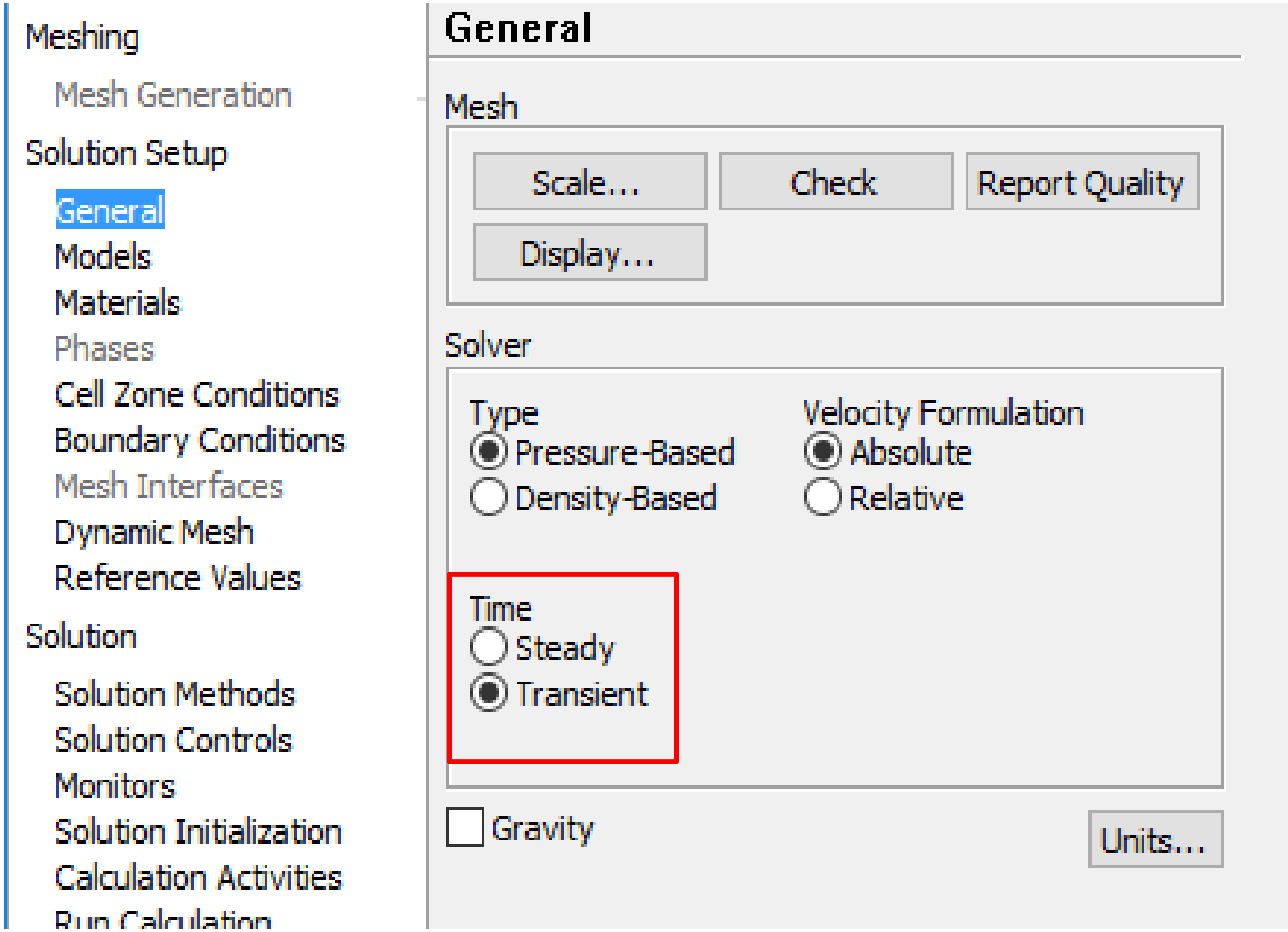
```
WARNING: The mesh contains high aspect ratio quadrilateral,
hexahedral, or polyhedral cells.
```

2st step: Scale the domain size

General→Scale

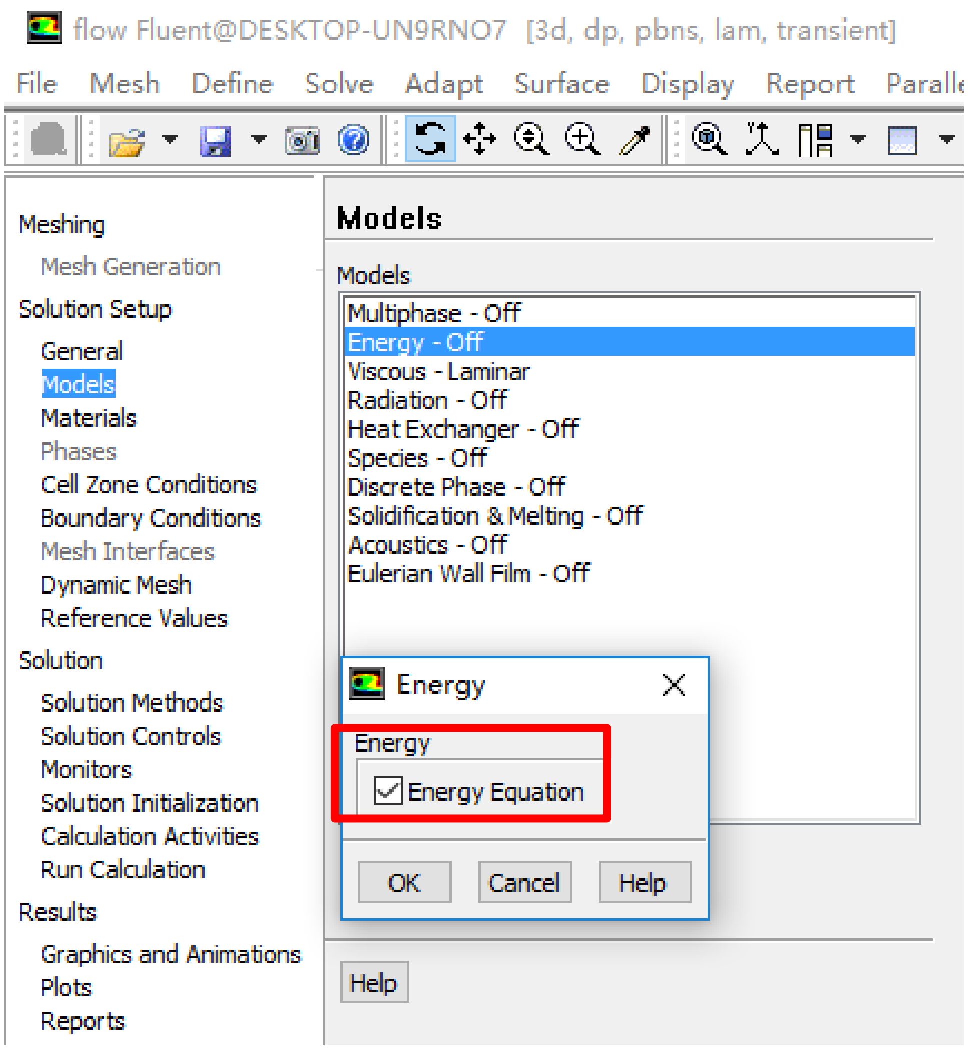
In Example 2, the mesh was created in ICEM in the length unit of “mm”. The diameter of the steel ball is 50mm.





Choose “transient” for a unsteady problem!

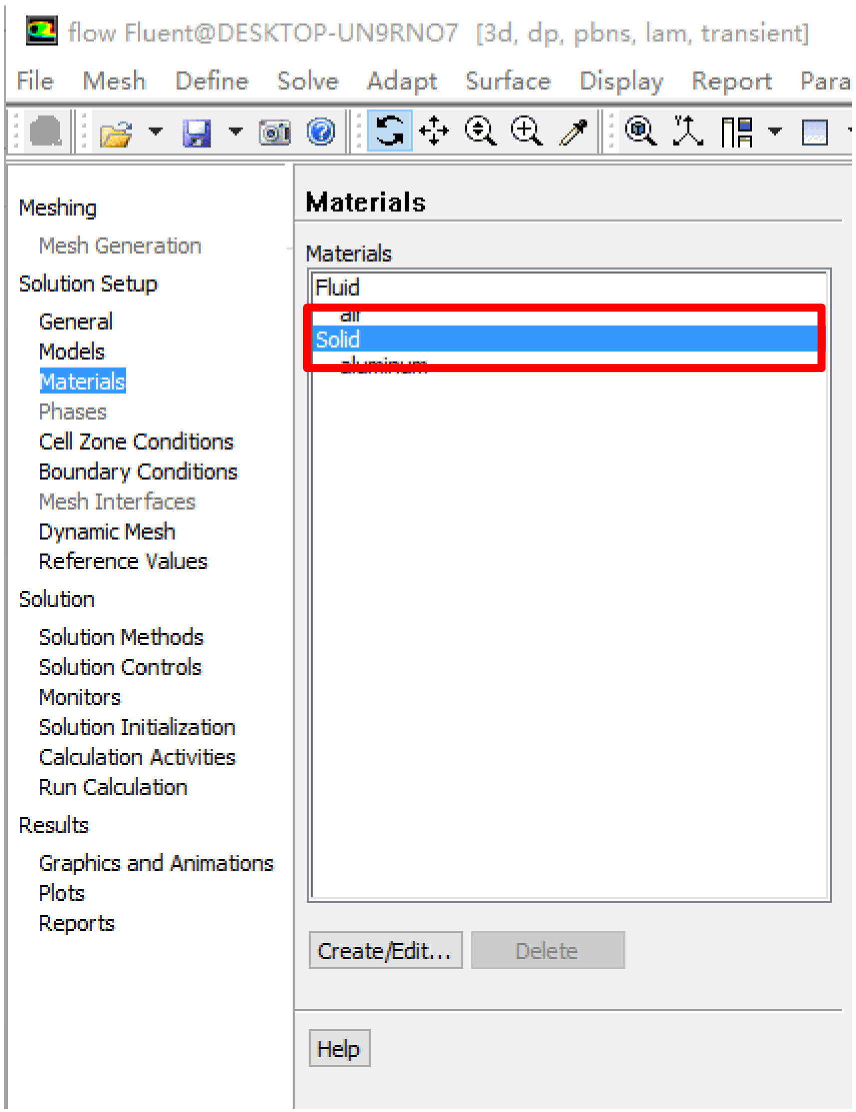
Step 3: Choose the physicochemical model



$$\frac{\partial\left(\rho C_p T\right)}{\partial t}=div(\Gamma_T gradT)$$

The energy equation is activated.

Step 4: Define the material properties



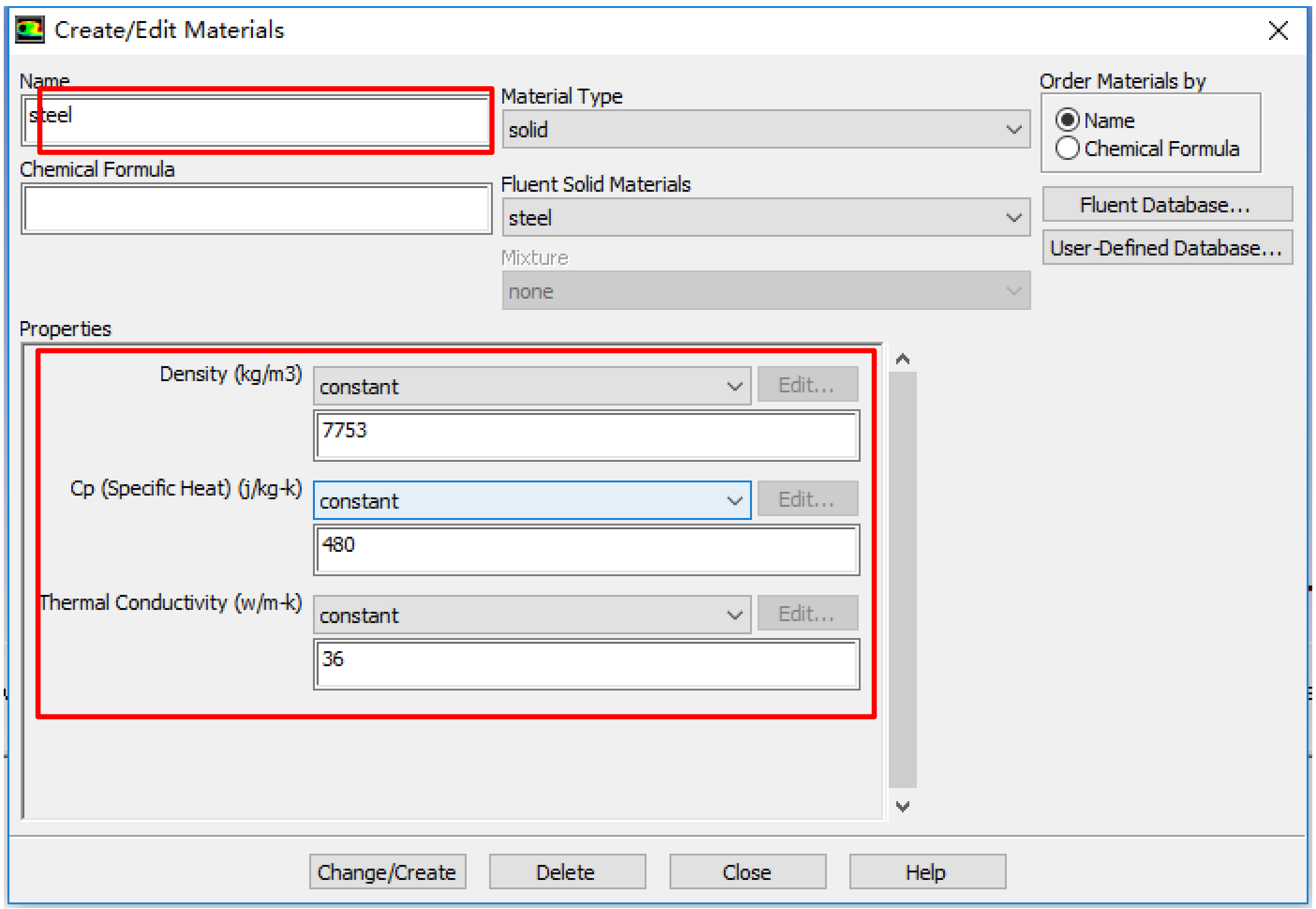
The default fluid in Fluent is air.

The default solid in Fluent is Aluminum.

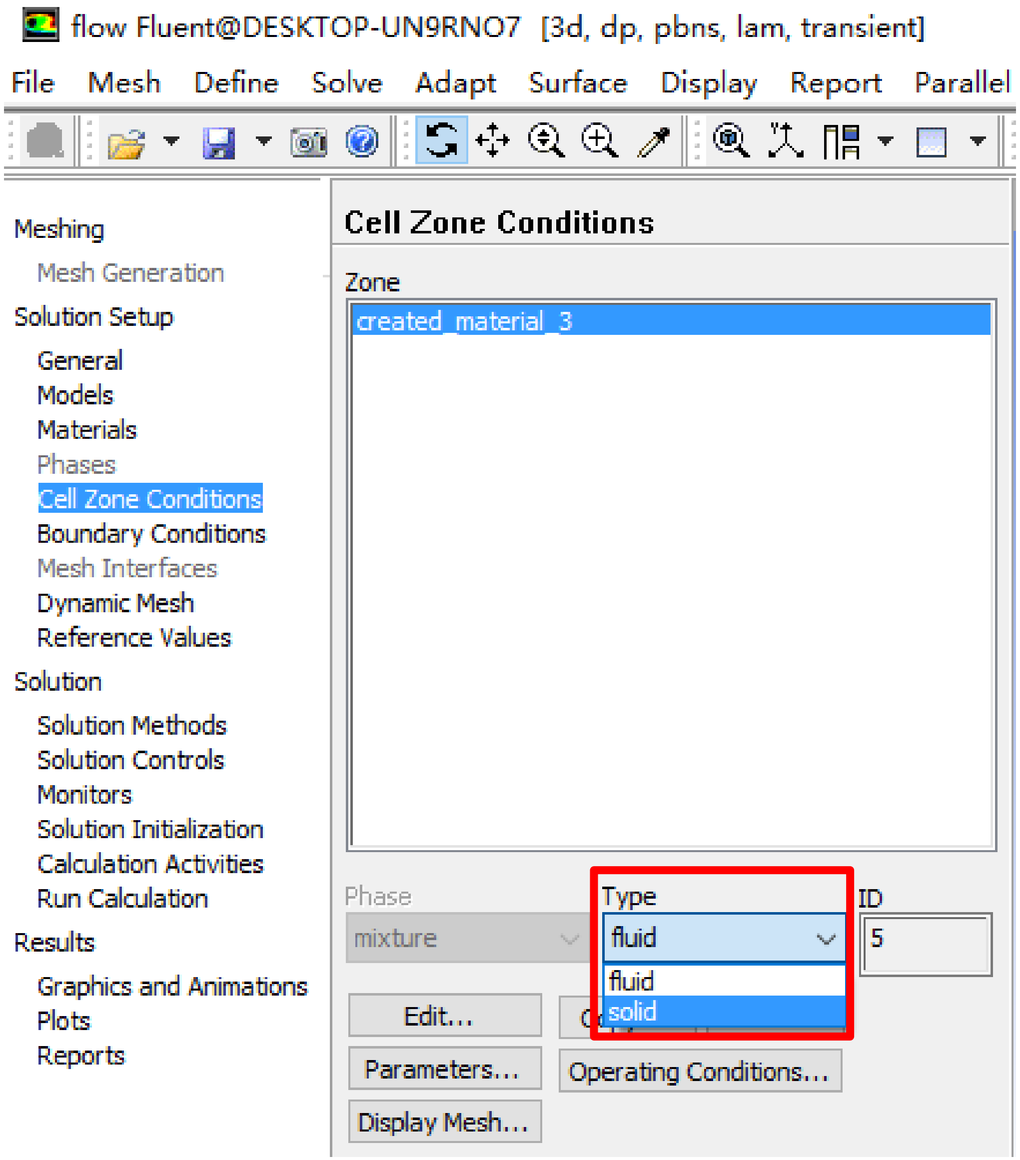
For Example 2, steel material should be added.

The properties of steel are manually inputted.

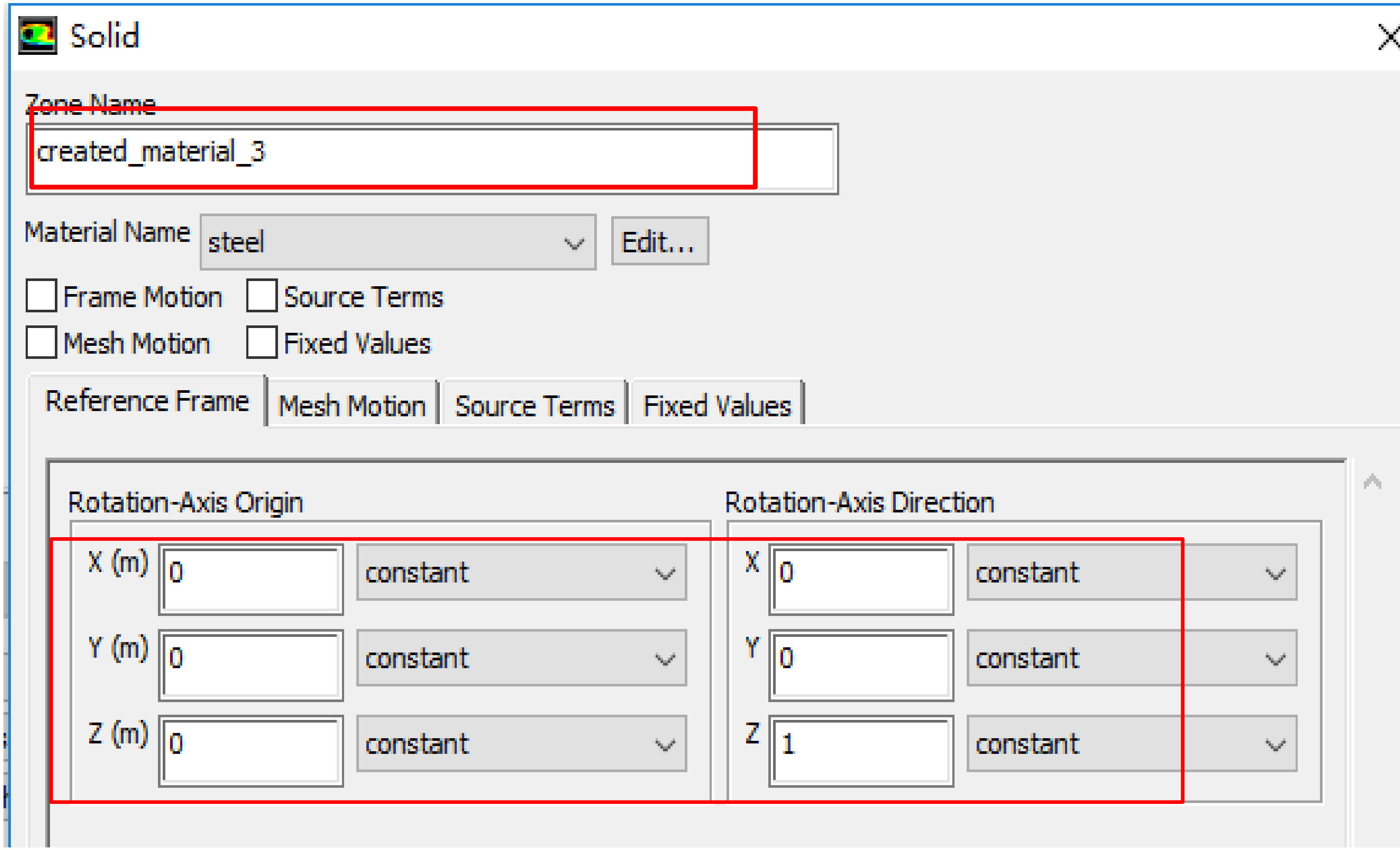
Density is 7735kg/m^3, heat capacity is 480 J/(kg K), conductivity is 33W/(m K)



Step 5: Define zone condition

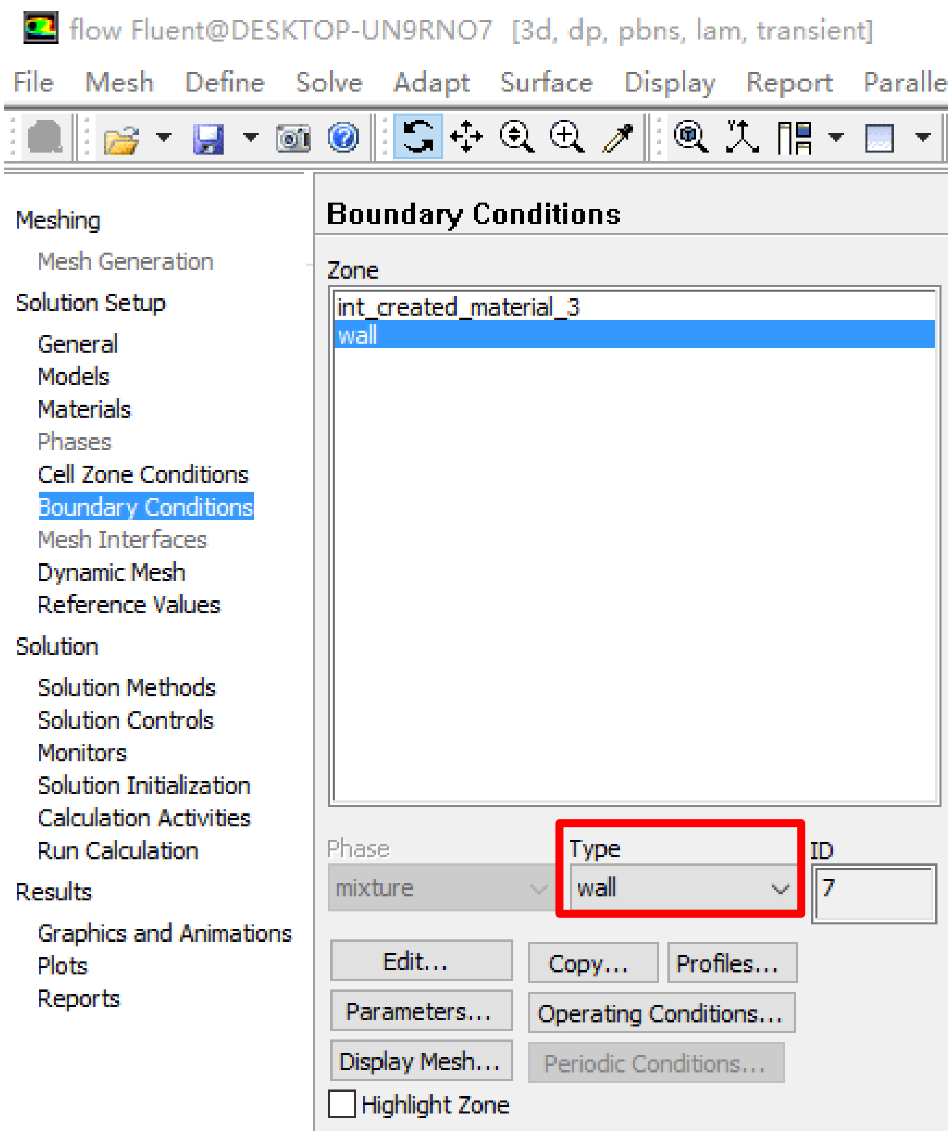


In this step, we define the cell zone conditions. , the cell zone is a ball made of steel, so you should choose the type “solid”.



Be sure the material is steel and others keep as default.

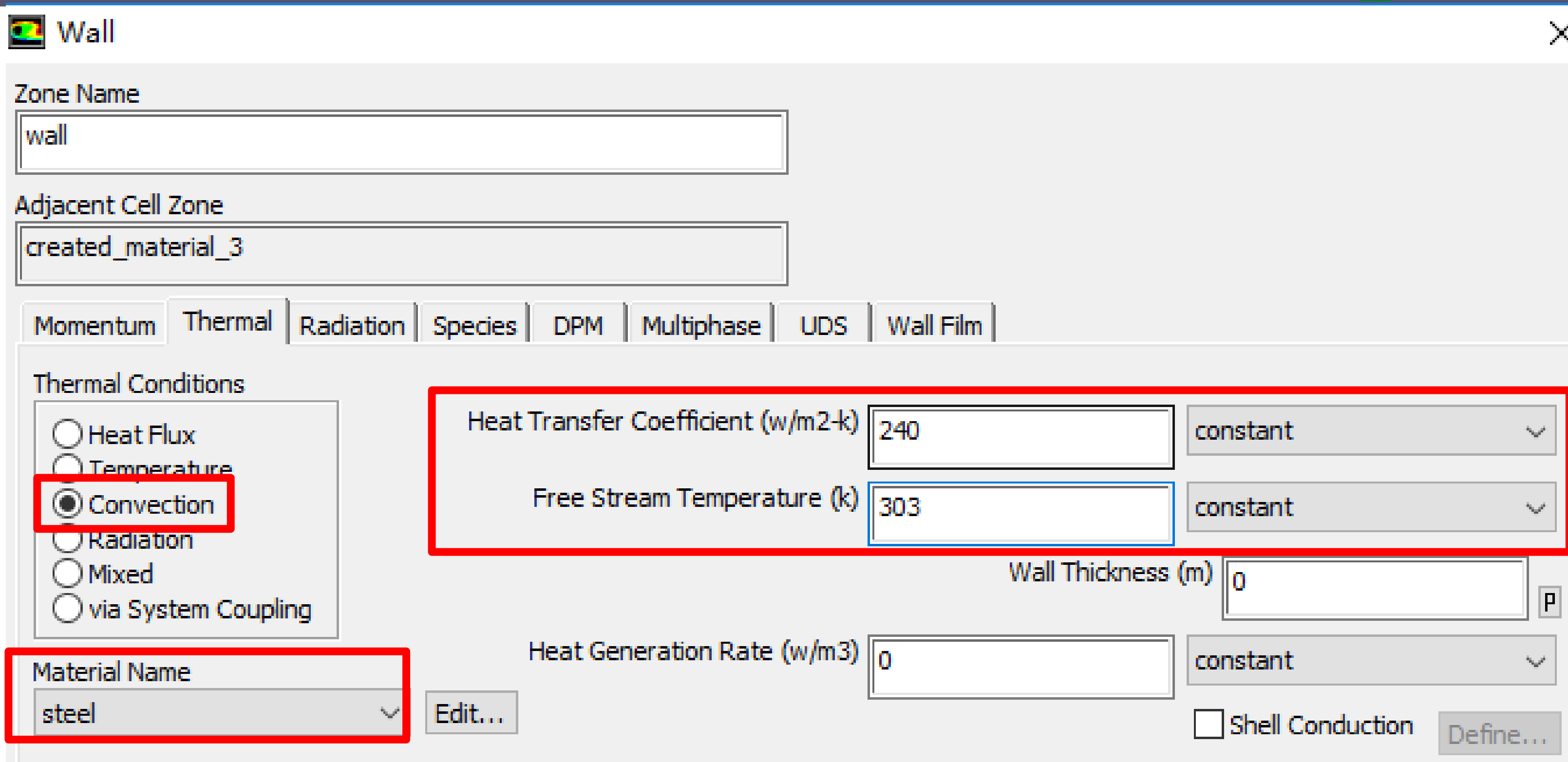
Step 6: Define the boundary condition



Now, you need to define the "Boundary conditions"

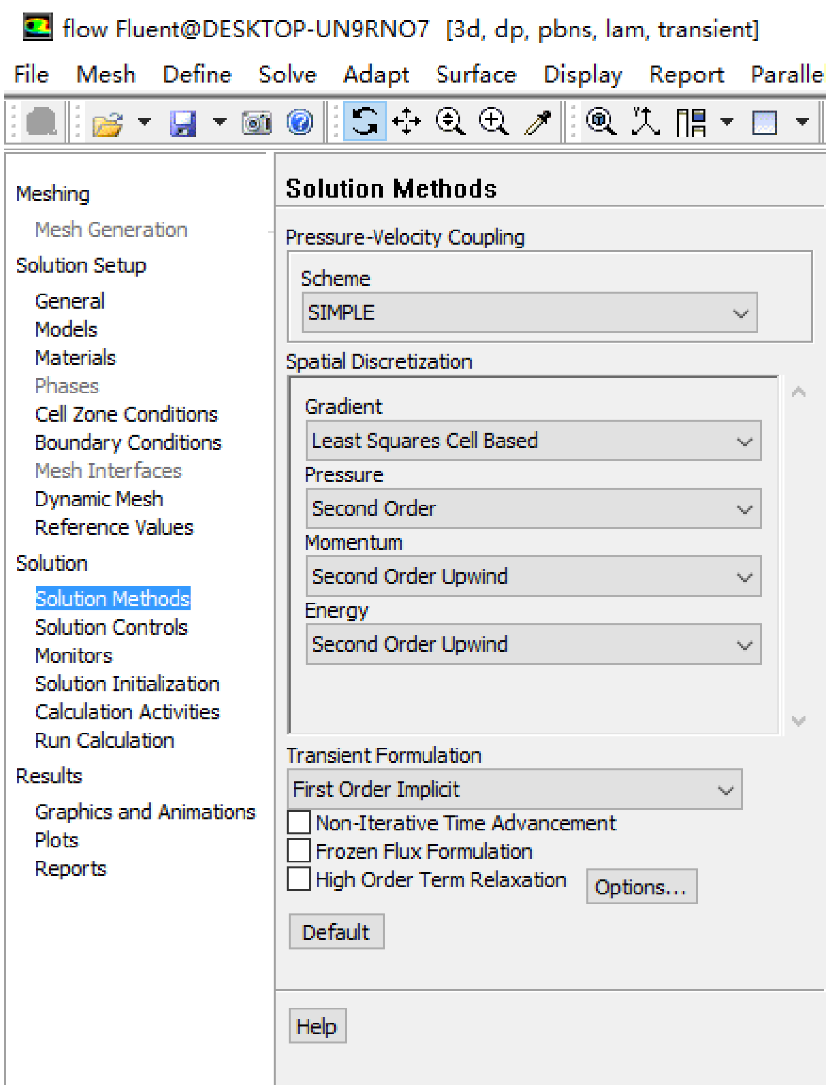
Firstly, Ensure the "type" is "wall".

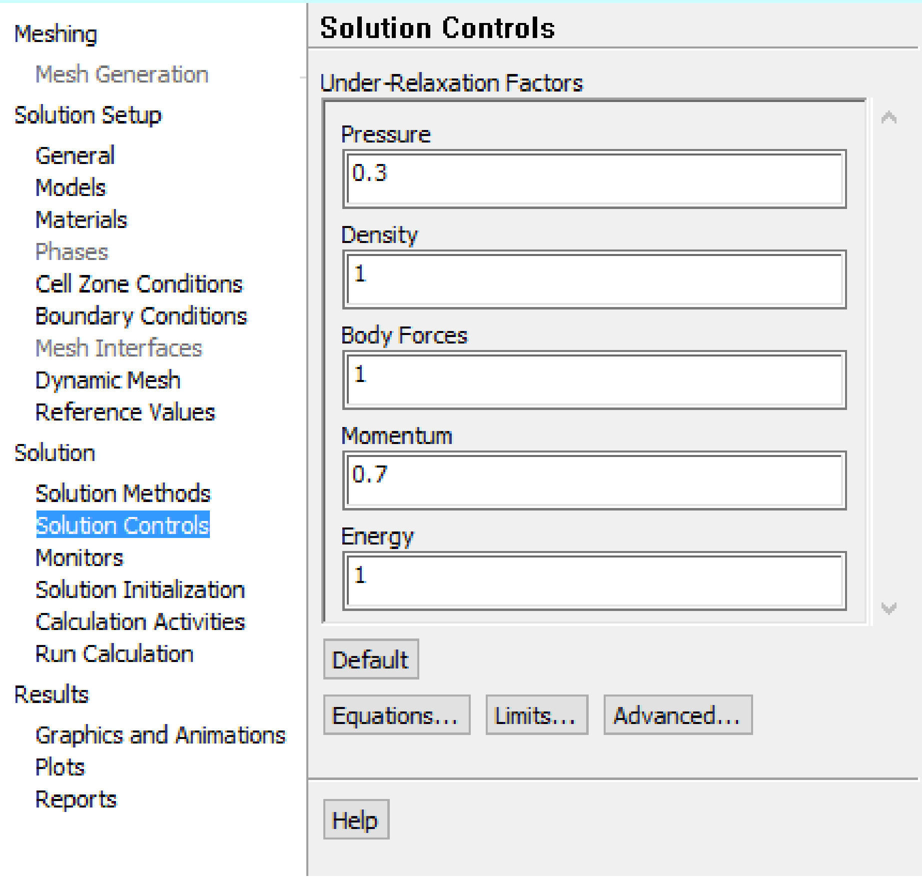
Then click the "edit" to edit the BC.



In this problem, the BC is third kind of boundary condition, so we select "Convection" and input 24 for "Heat Transfer Coefficient", and 303K for the "Free Stream Temperature".

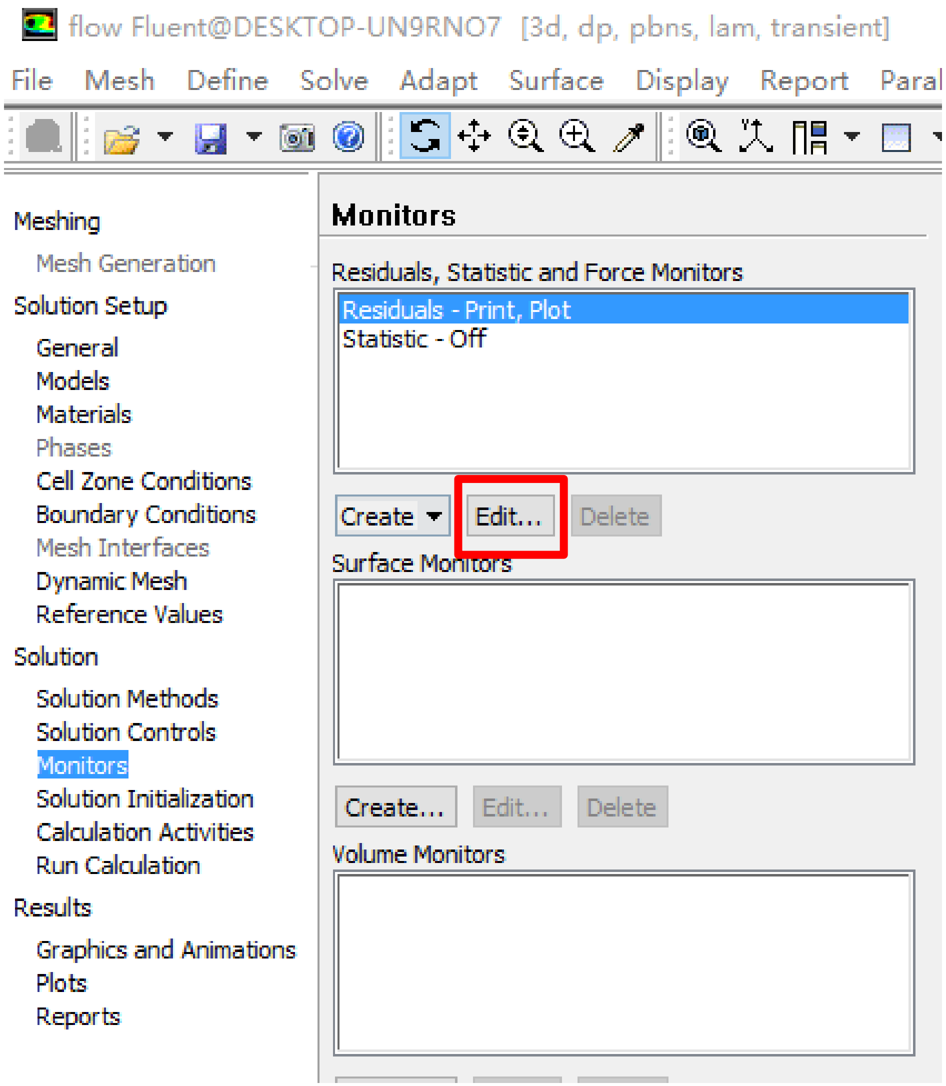
Step 7: Solution setup: algorithm and scheme





The default algorithm, schemes and under-relaxation factors are used.

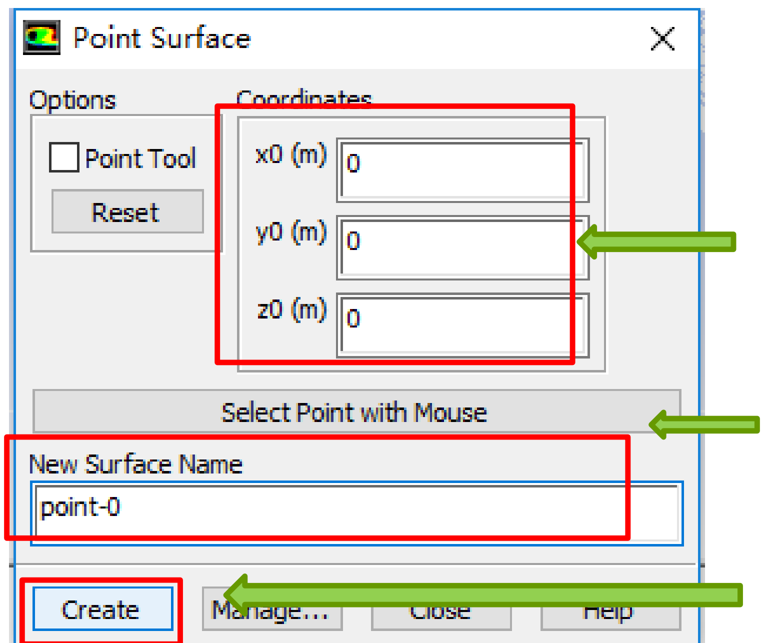
Step 7: Solution setup: monitors

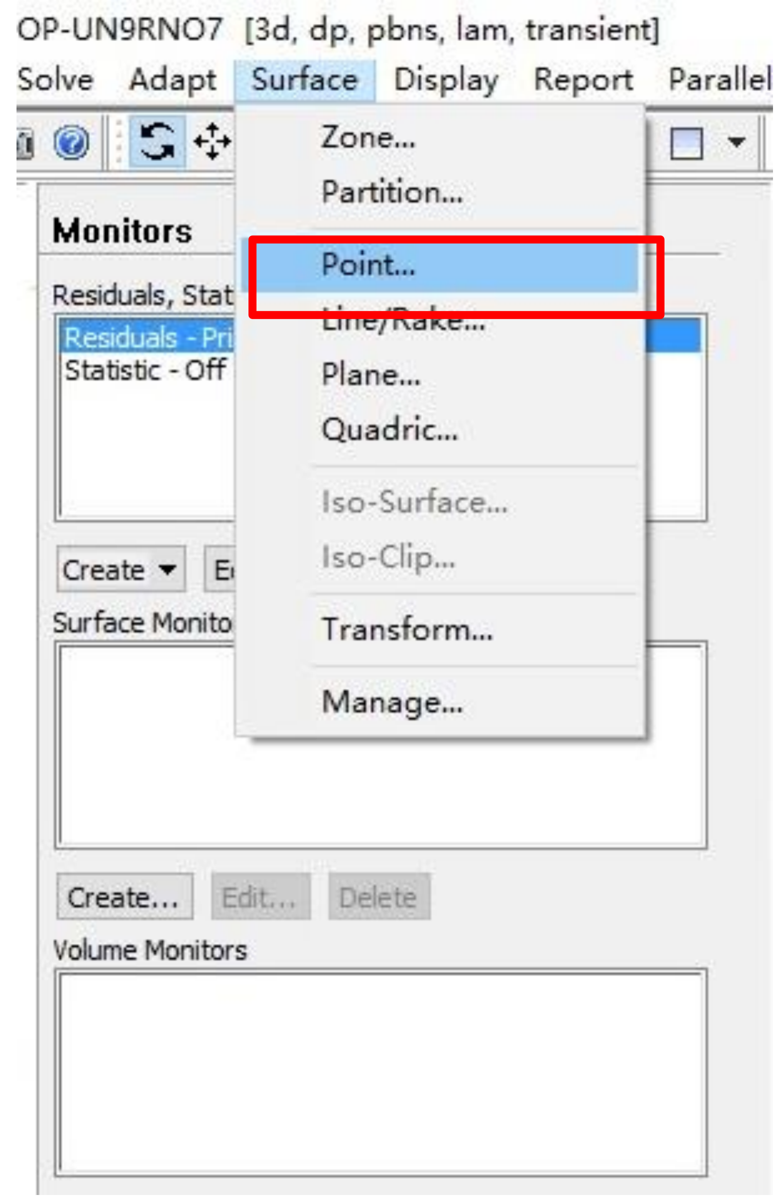


In this step, the residual can be changed.

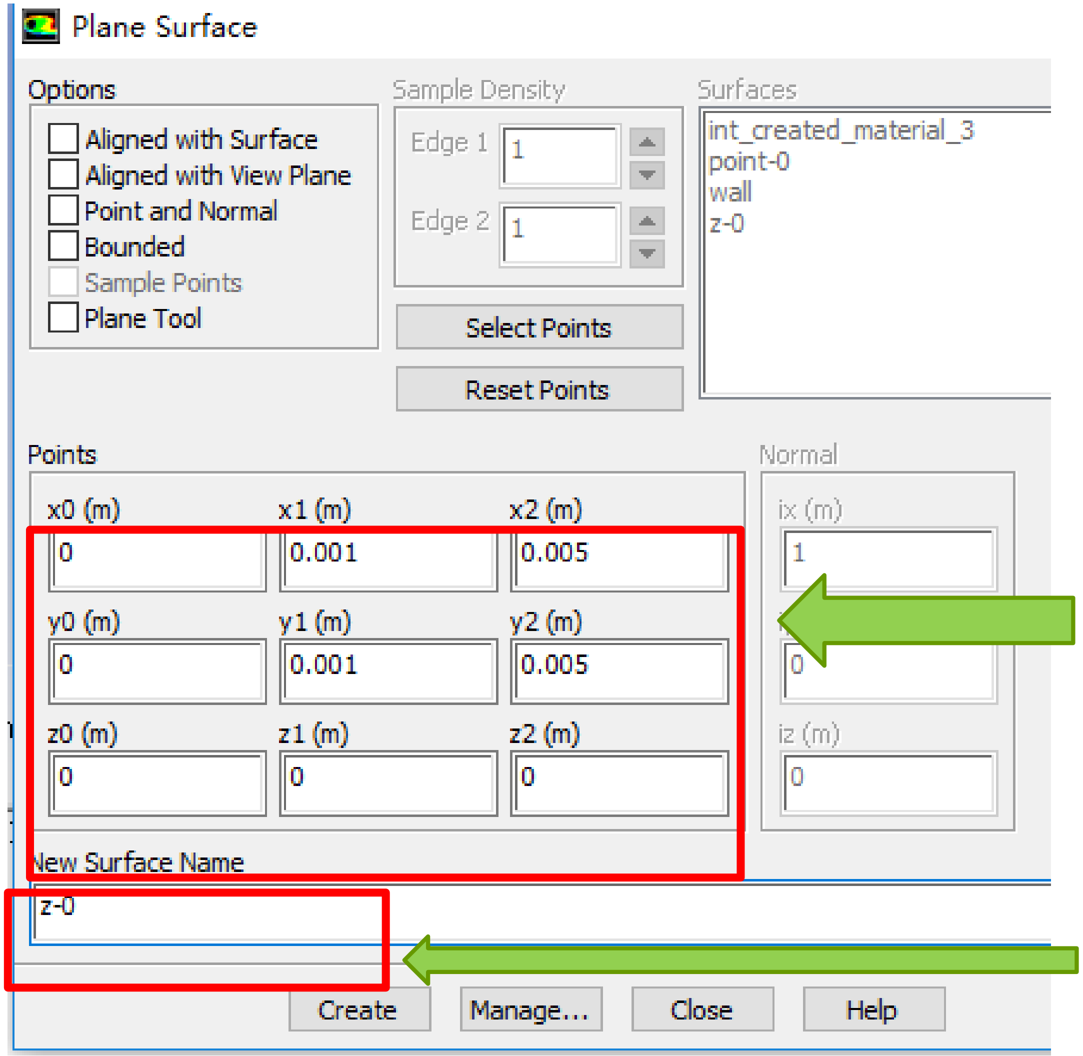
You also can define a point, a line or a surface to monitor related variables.

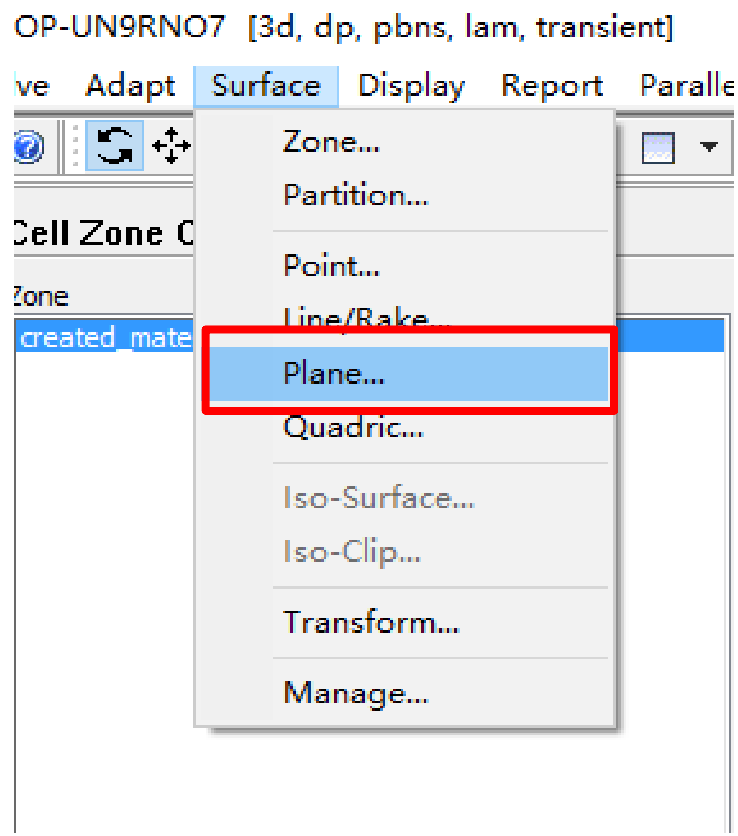
Here, you can create a point by clicking "surface" and choose "point", the "point" dialog will display.

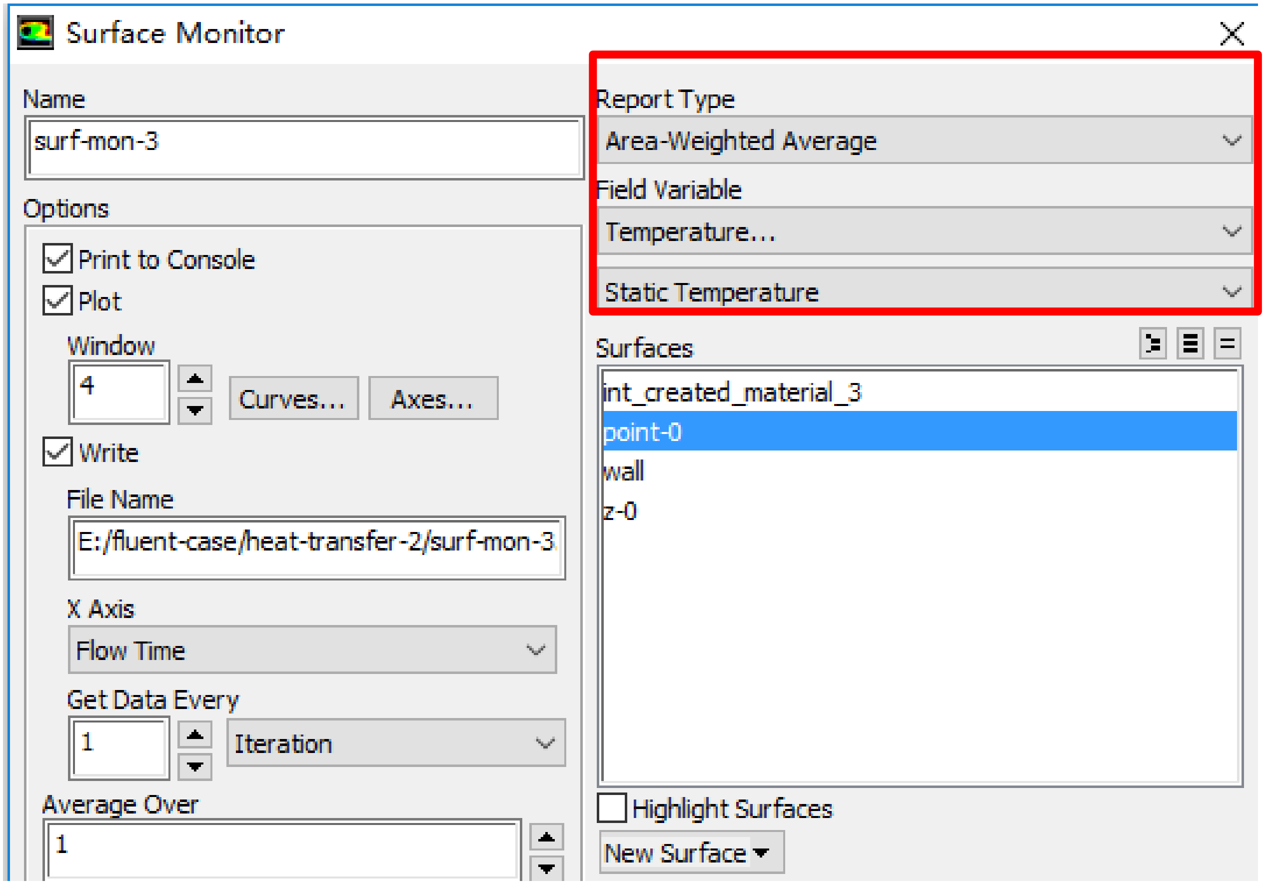




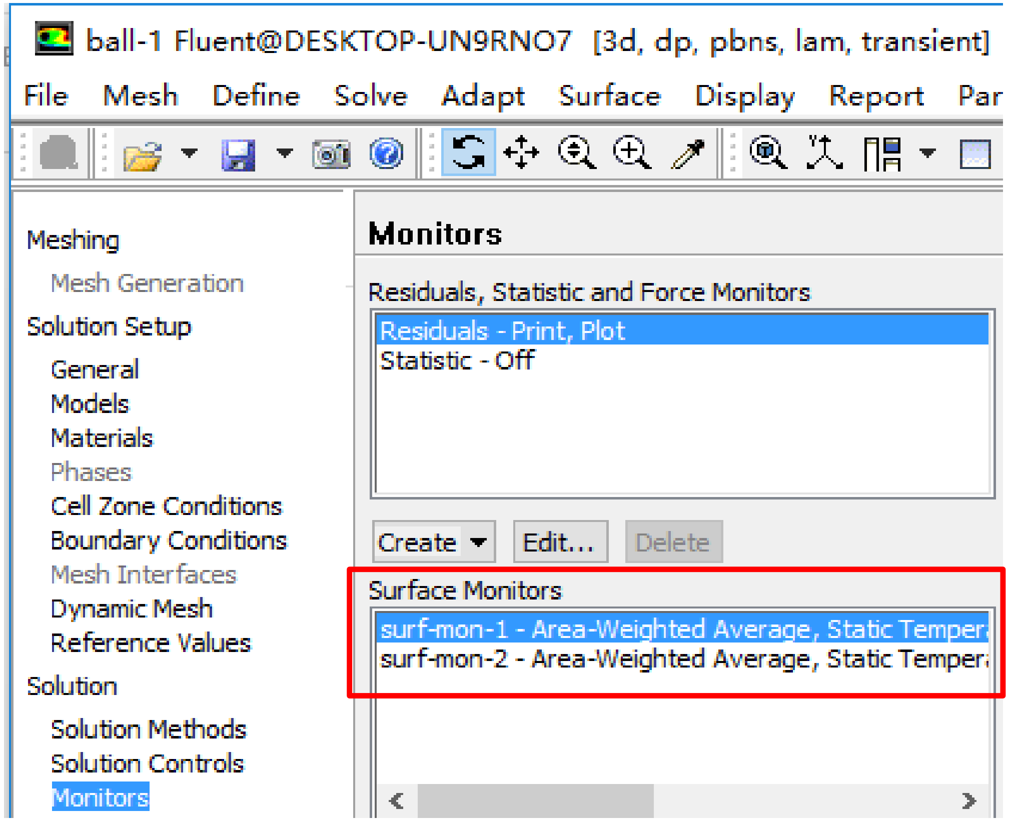
You can also create Plane by defining three points in the surface.







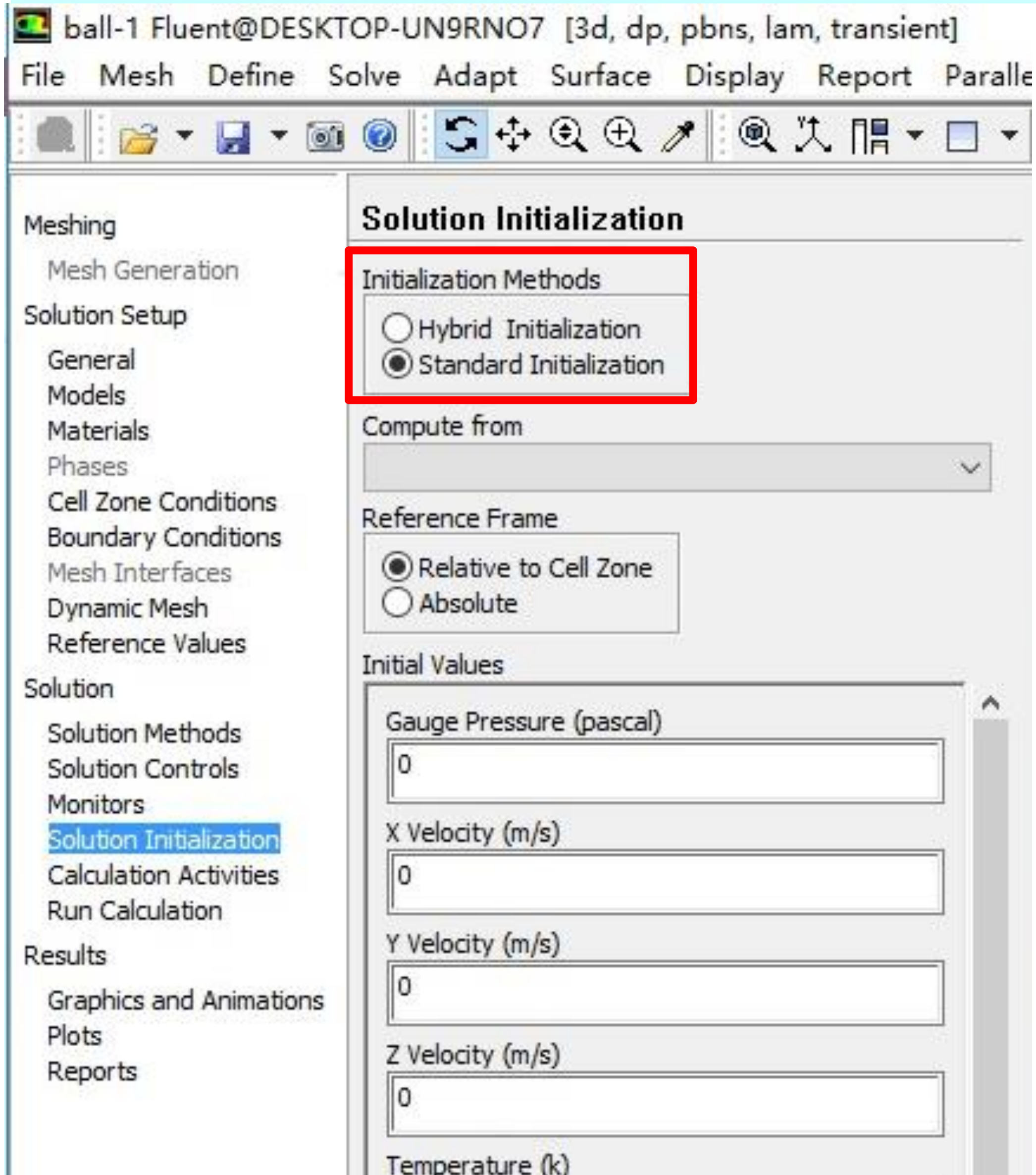
Next, you can create the monitors in the "Monitors" dialog. Select the "Report type", the variable you want to monitor, and the position you want to monitor.



Similarly, you can create a monitor to monitor the average temperature on the surface "z-0".

In the "Surface Monitors", you can see two monitors created.

Step 8: Initialization



For the details of Hybrid initialization and standard initialization, you can refer to Example 1.

Here, the Standard initialization is adopted.

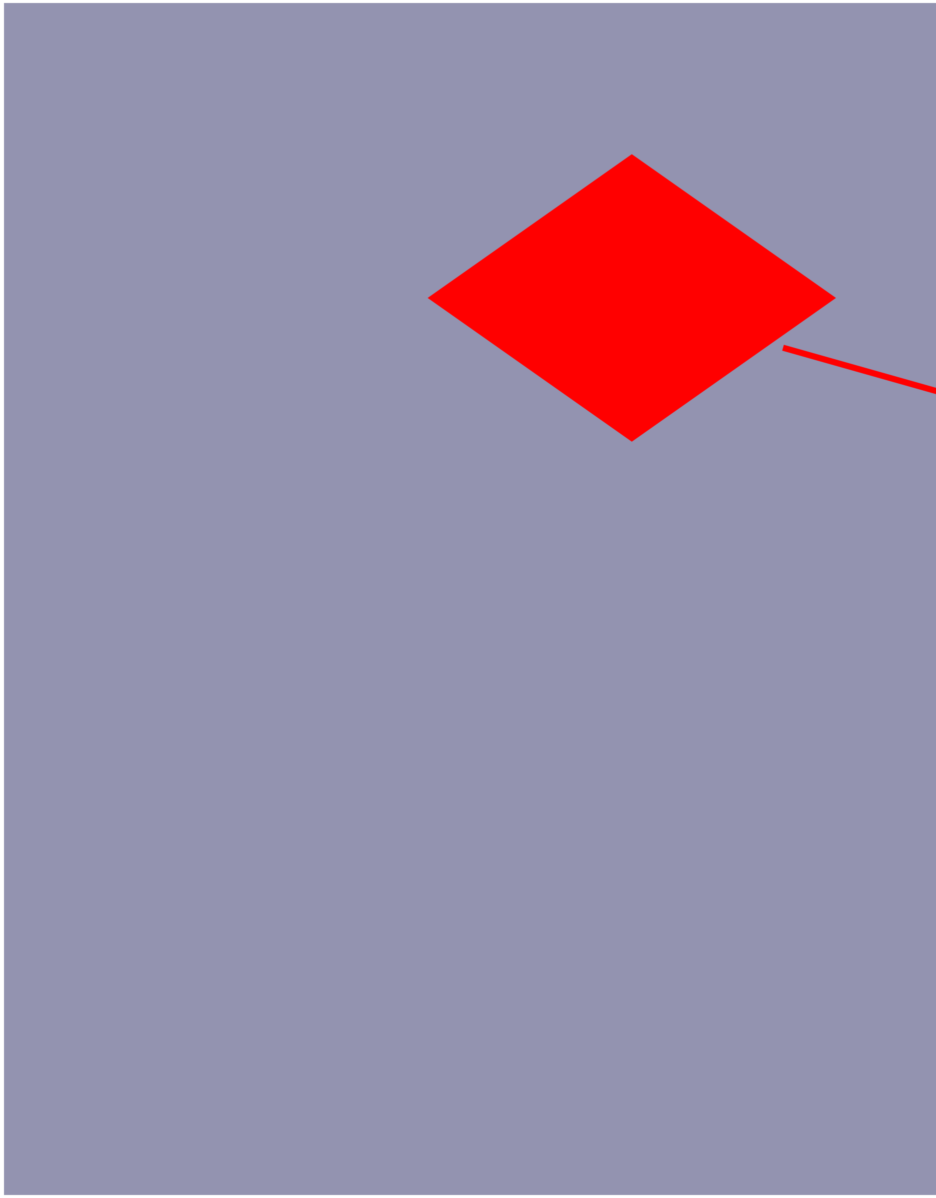
Patching (修补）Values in Selected Cells

After you have initialized the entire domain, you may want to define a different value for a sub-region in the domain.

For multiphase flow, you may also want to define the volume of fraction for a phase in a particular sub-region.

This can be achieved by using the Patch function!

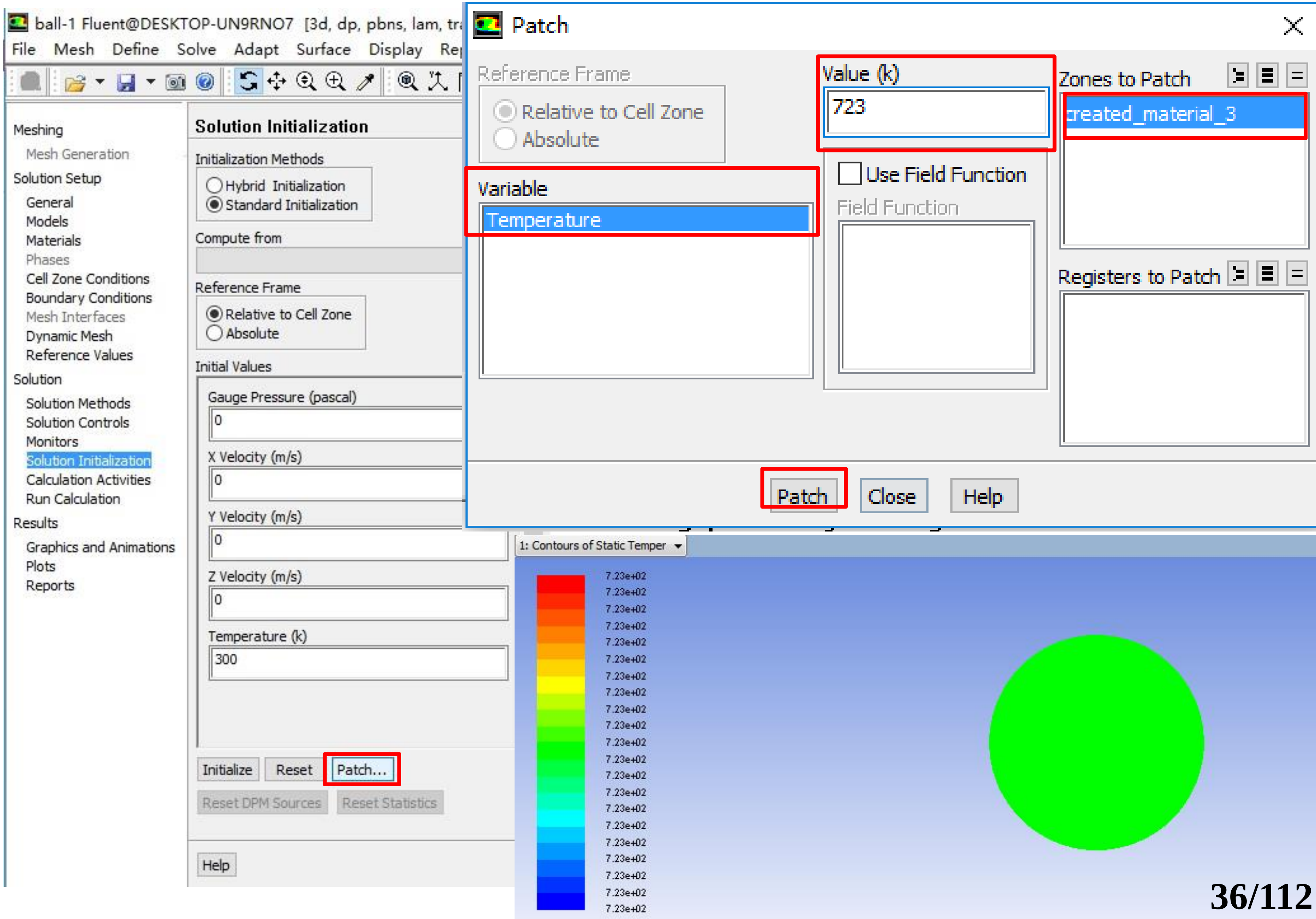
In Example 2, the Patch function is adopted to define the temperature of the entire domain as 723K.



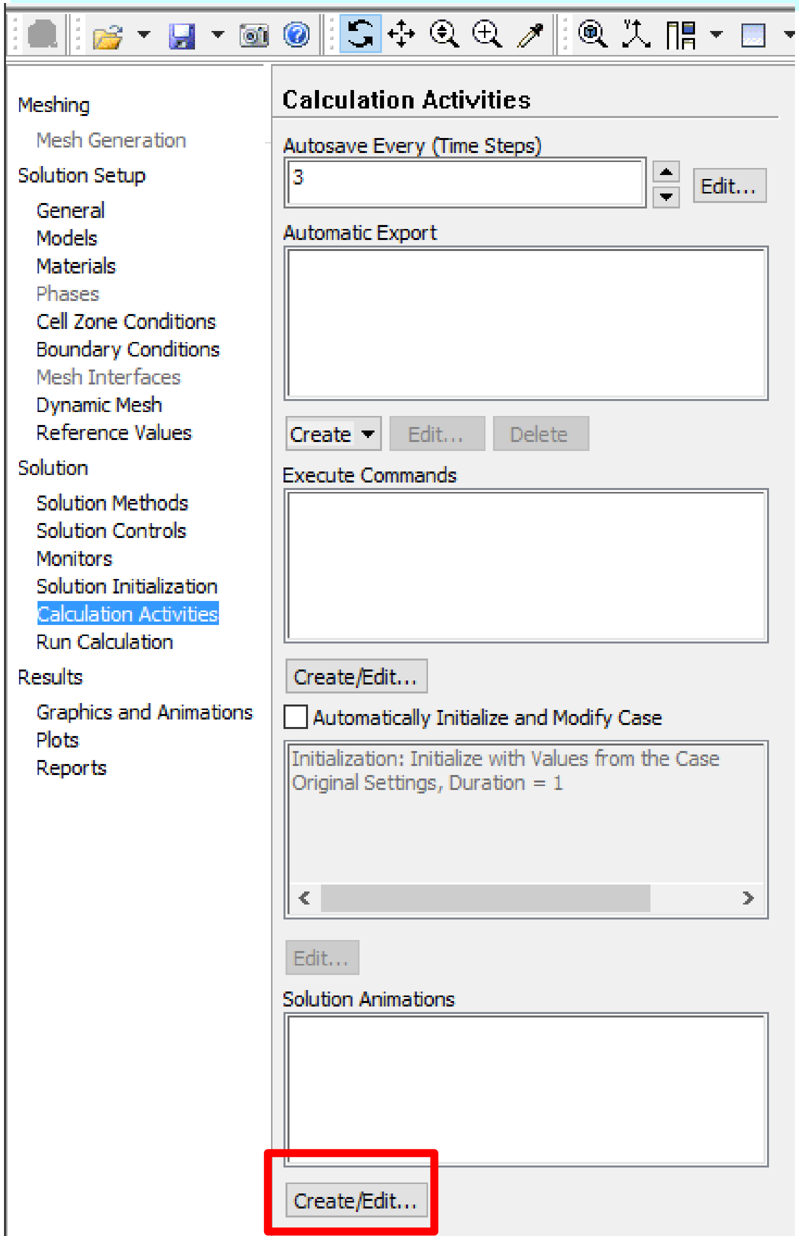
Domain

Sub-region need to Patch

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

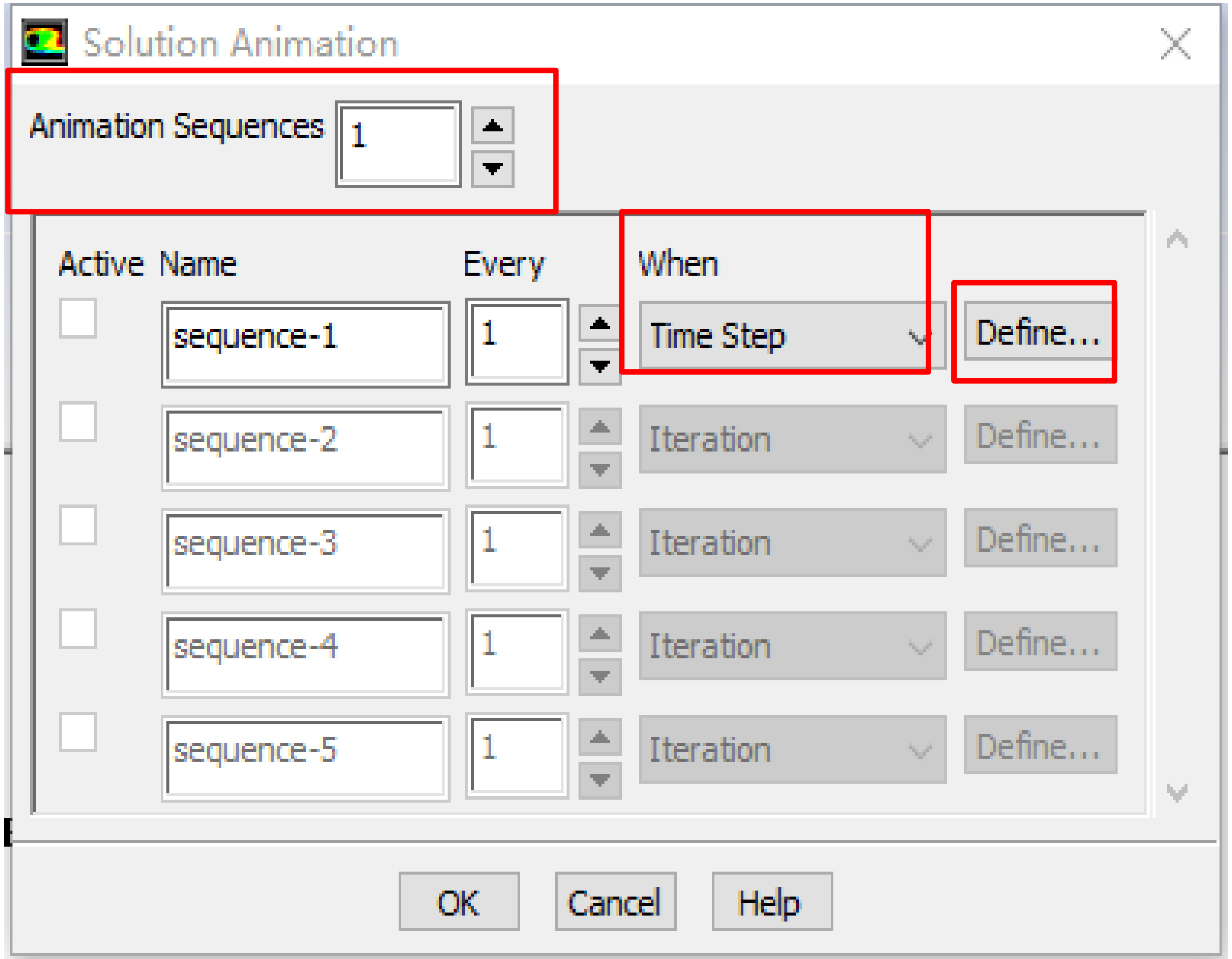

ball-1 Fluent@DESKTOP-UN9RNO7 [3d, dp, pbns, lam, tr
File Mesh Define Solve Adapt Surface Display Re
Meshing
Mesh Generation
Solution Setup
General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
Monitors
Solution Initialization
Calculation Activities
Run Calculation
Results
Graphics and Animations
Plots
Reports
Solution Initialization
Initialization Methods
Hybrid Initialization
Standard Initialization
Compute from
Reference Frame
Relative to Cell Zone
Absolute
Initial Values
Gauge Pressure (pascal)
0
X Velocity (m/s)
0
Y Velocity (m/s)
0
Z Velocity (m/s)
0
Temperature (k)
300
Initialize
Reset
Patch...
Reset DPM Sources
Reset Statistics
Help
Patch
Reference Frame
Relative to Cell Zone
Absolute
Value (k)
723
Zones to Patch
created_material_3
Variable
Temperature
Use Field Function
Field Function
Registers to Patch
Patch
Close
Help
1: Contours of Static Temper
7.23e+02

9st step: set animations



We can set animations to monitor the development of temperature in surface: z-0.

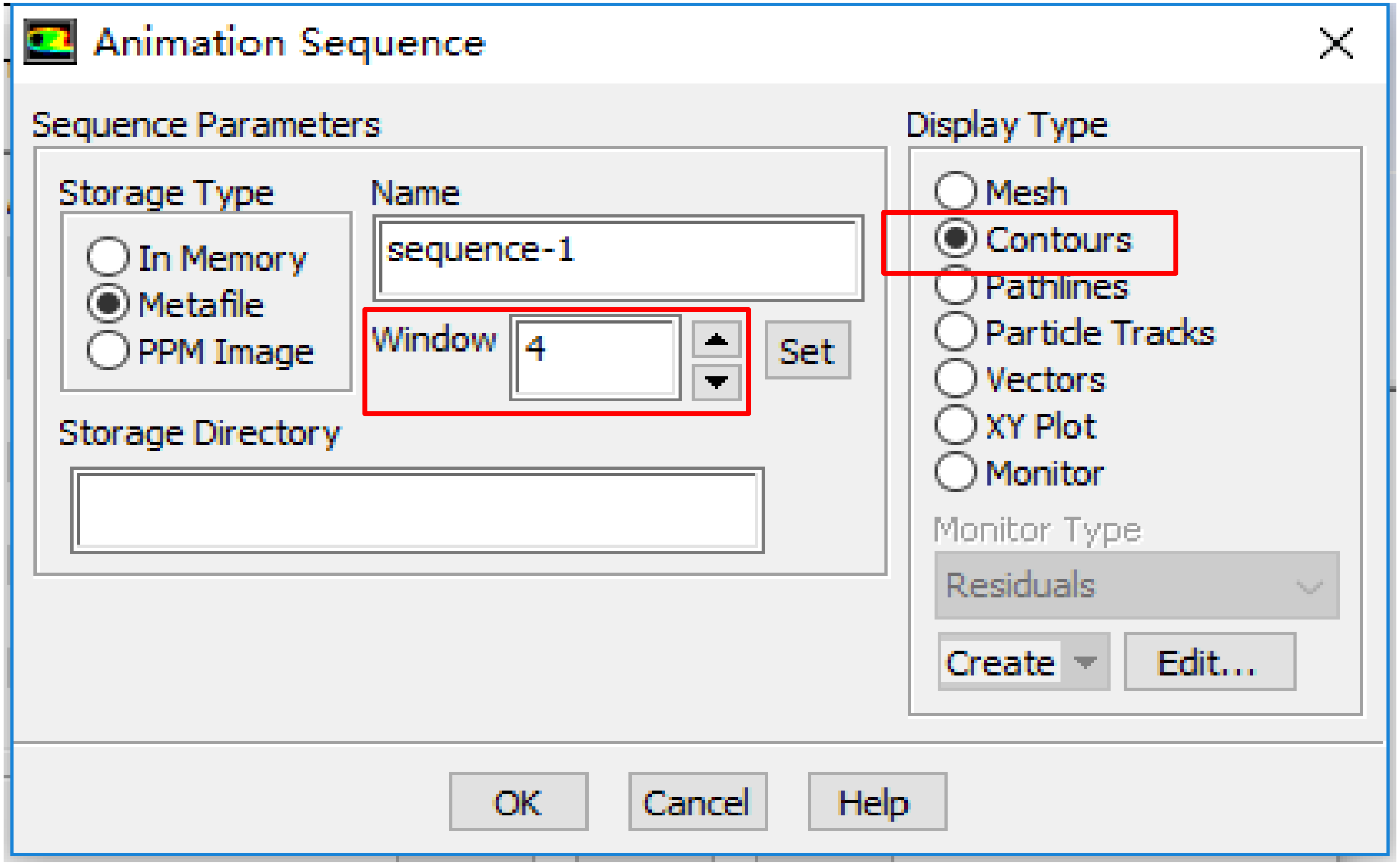
In the "Calculation Activities" dialog, click "Change/Create" in "Solution Animations".



Set the "Animation Sequences" as 1.

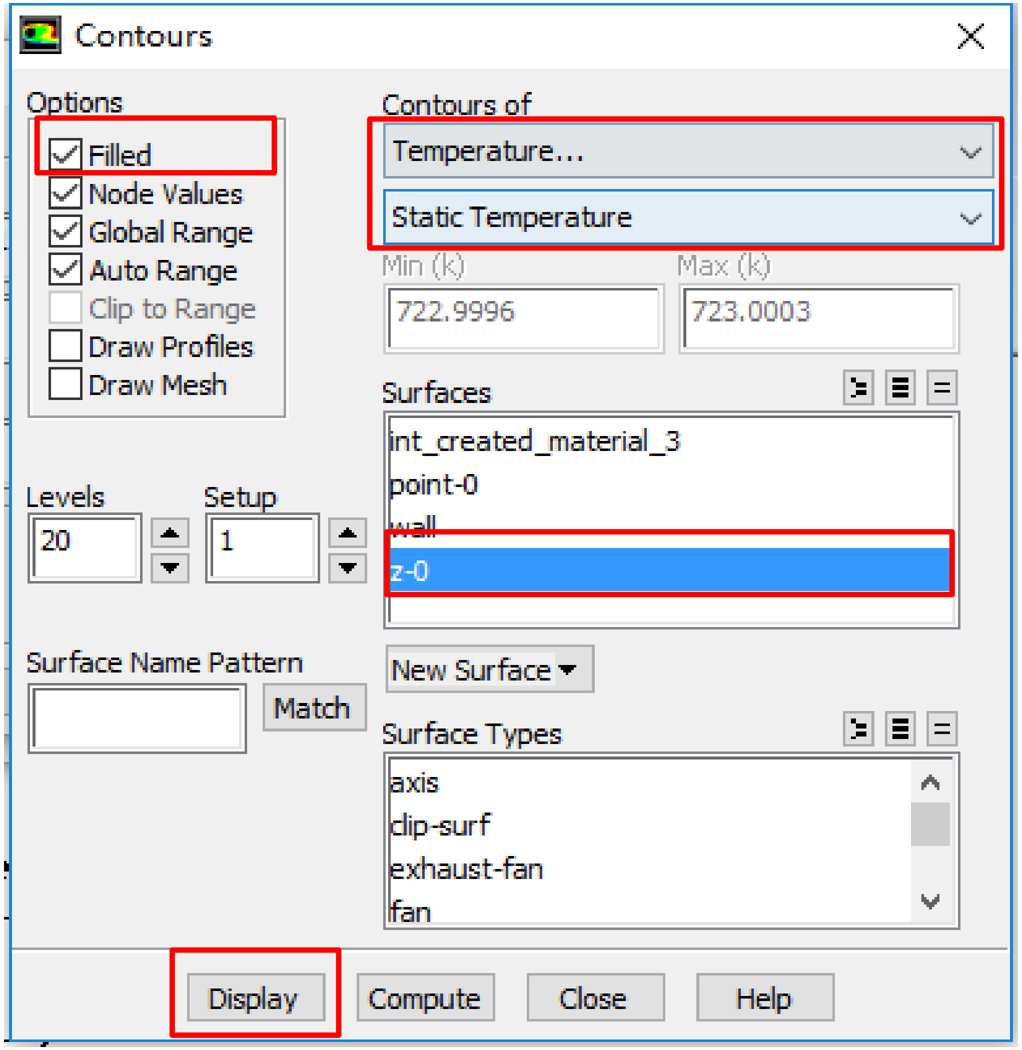
Select "Time Step" in "When".

Click "Define" to set the animation.



Give the "Window" a number and click "Set", we create a window for animation to display.

Select "Contours" to display contours.

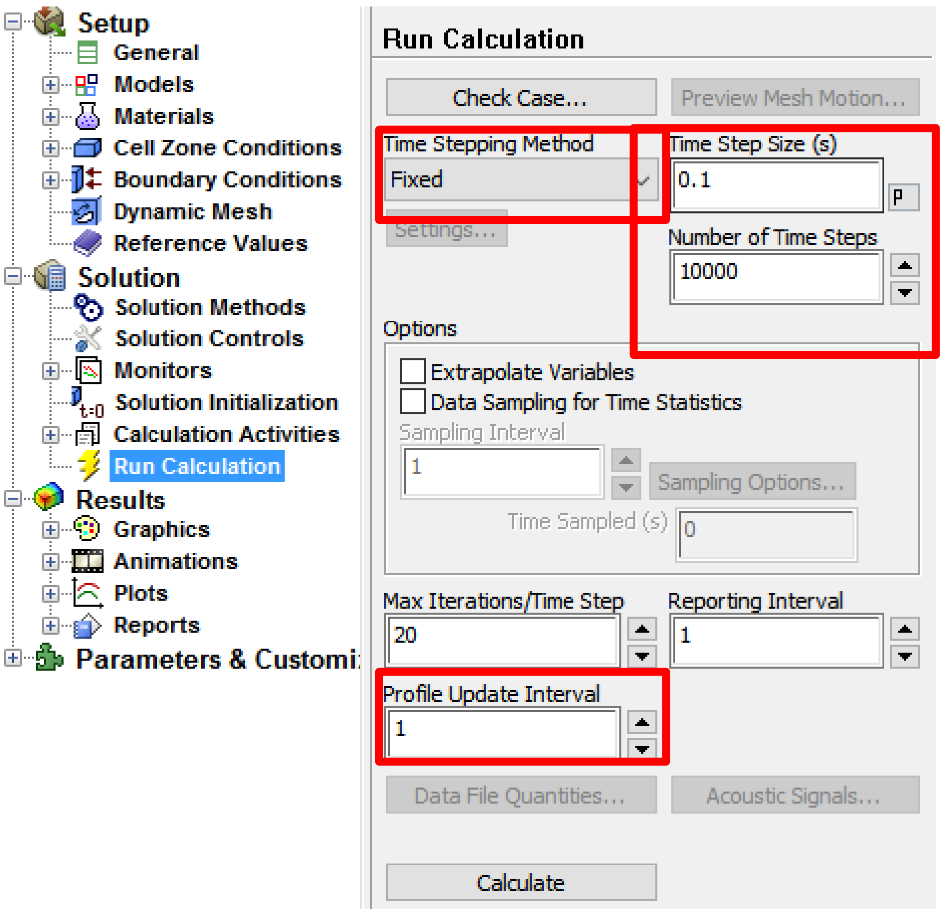


In "Contours" dialog, we choose "Temperature", select "Filled", and choose the surface: z-0.

Click Display, the initial temperature distribution will display in the window we created.

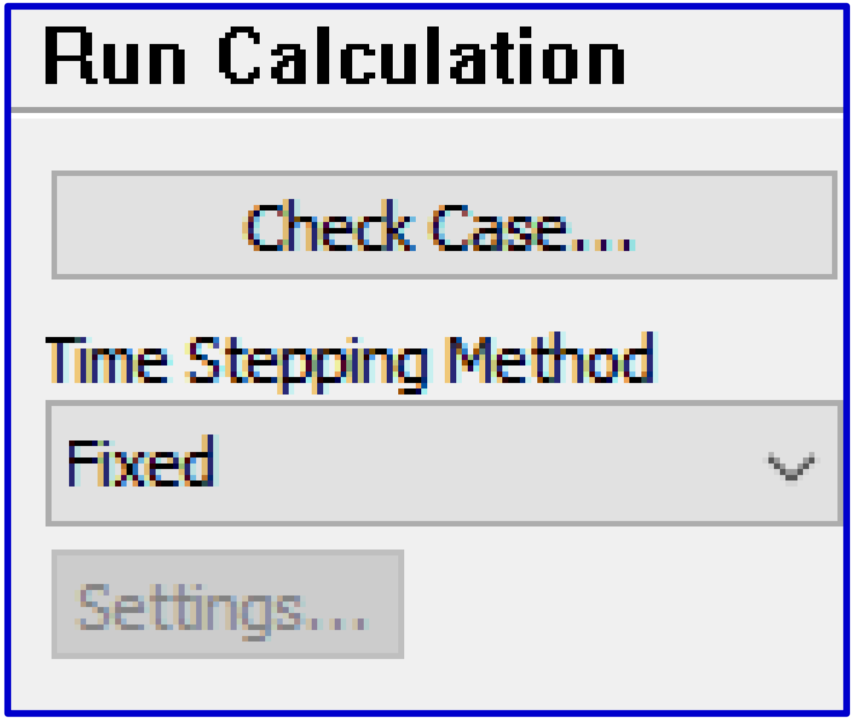
Step 9: Run the simulation

The interface of transient problem is a little complicated compared with steady problem.

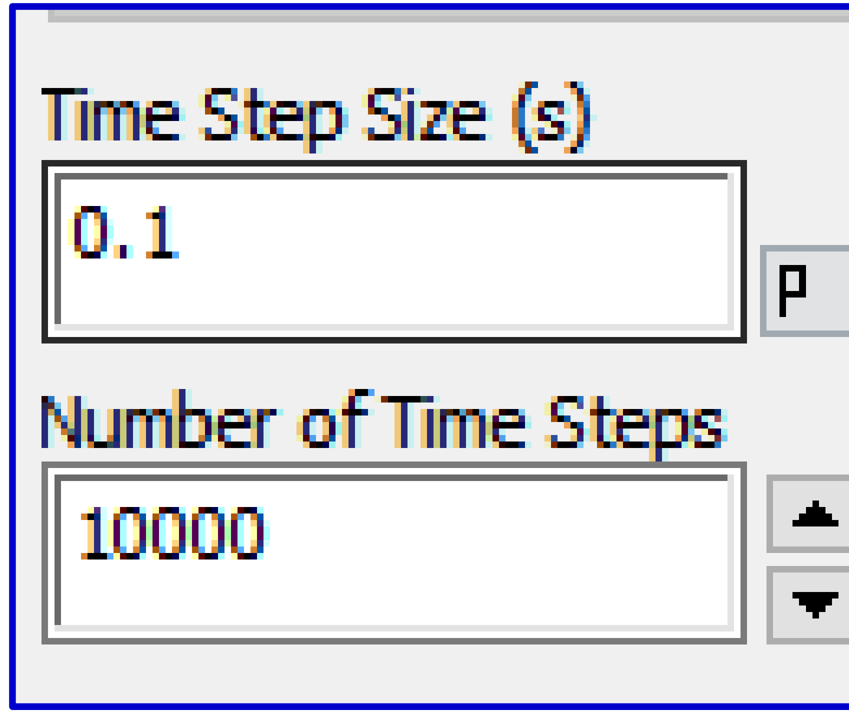


You need to select the time stepping method, set the time step size, and the max iteration per time step.

Time stepping method



Time step size



Iteration per time step

O U T E R I T E R A T I O N

ITER=1

⋮

ITER= ITER+1

⋮

TIME=TLAST

ITER=LAST

Outer Iteration

$A_{p,\ s,n,e,s}$ **does not change**

NT=1

⋮

NT=i

NTIMES(NF)

Two line iterations in x direction

Two line iterations in y direction

Two block corrections in x,y direction

Number of specified iteration cycles

Inner Iteration

Max Iterations/Time Step:

Set the max iterations in each time step to make sure convergence criteria is satisfied. It is the same as the inner iteration in our teaching code. Here it is set as 10.

Time step size

Fully implicit scheme is adopt in Fluent. Therefore, the value of *Δt* will not affect the stability. However, it will affect the accuracy.

$$a_P\phi_P = a_E\phi_E + a_W\phi_W + a_S\phi_S + a_N\phi_N + b$$

$$a_P = a_E + a_W + a_N + a_S + a_P^0 - S_P\Delta V$$

$$b = S_C\Delta V + a_P^0\phi_P^0 \qquad a_P^0 = \frac{\rho_P\Delta V}{\Delta t}$$

7.3.1 Sufficient condition for iteration convergence of Jakob and G-S iteration

1. Sufficient condition－Scarborough criterion

Coefficient matrix is non-reducible (不可约), and is diagonal predominant(对角占优）：

$$\frac{\sum |a_{nb}|}{|a_P|} \le 1 \begin{cases} \le 1 & \text{for all equations (a)} \\ < 1 & \text{at least for one equations (b)} \end{cases}$$

$$a_P = a_E + a_W + a_N + a_S + a_P^0 - S_P \Delta V$$

$$b = S_C \Delta V + a_P^0 \phi_P^0 \qquad a_P^0 = \frac{\rho_P \Delta V}{\Delta t}$$

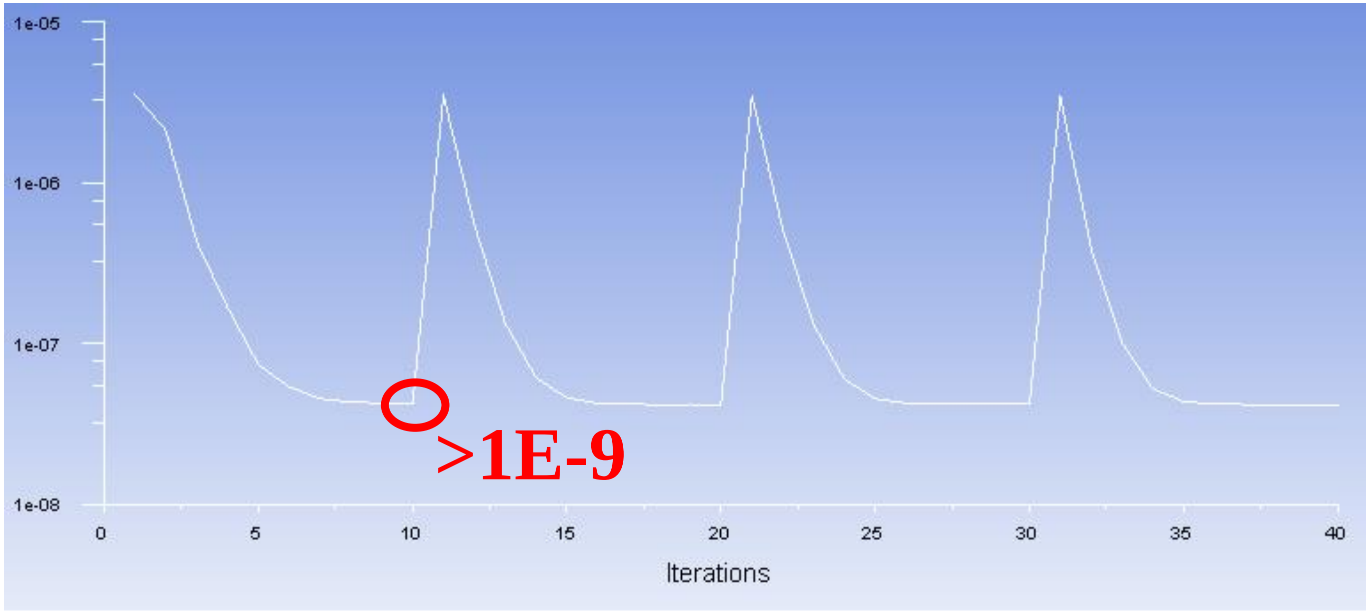
However, Δt will affect the accuracy of the simulation results.

The following way is recommended by Fluent to set Δt:

1. At each time step, the ideal iteration number is 5-10.

2. If Fluent needs more inner iteration step (>10) for convergence at each time step, Δt is too large.

3. If Fluent needs only a few iteration steps, Δt is too small.



Here, the convergence criteria is 1e-9, Fluent needs more than 10 step to achieve the criteria. Thus Δt is too large here.

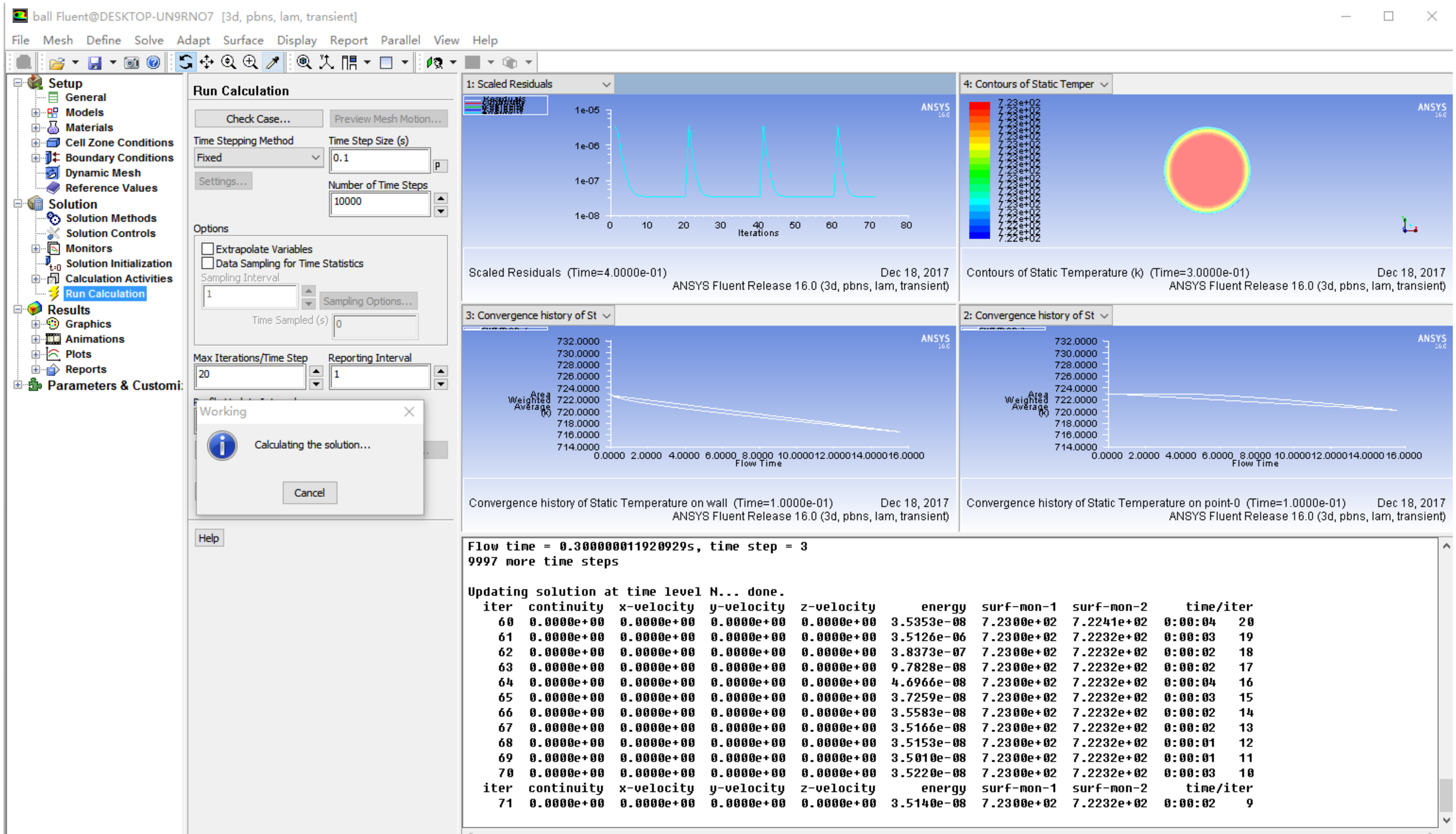
Usually, Δt should be small at beginning and then can be increased after 5-10 time steps.

Time stepping method

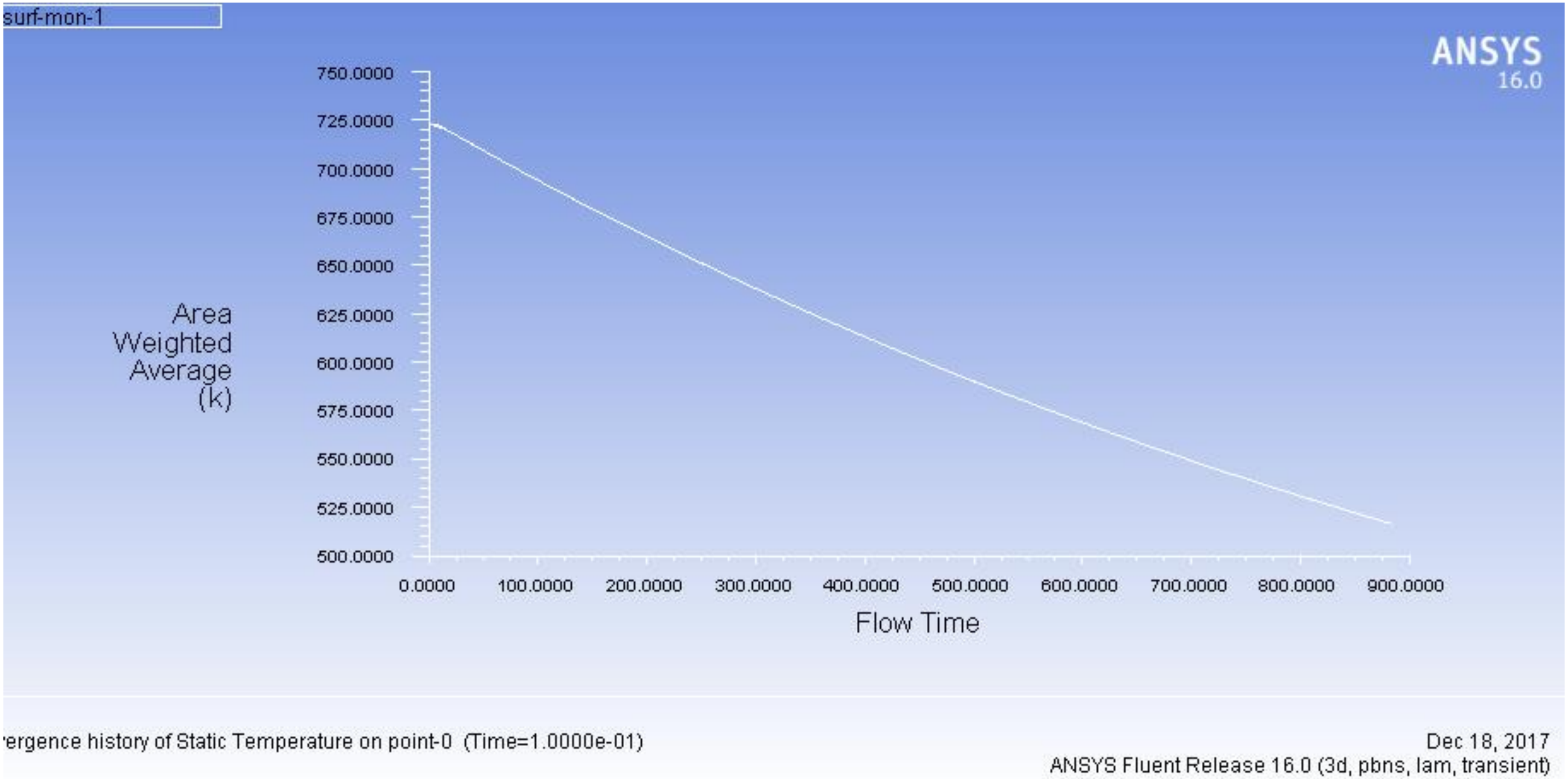
Here for Example 2, you can simply set the time stepping method as fixed, indicating the time step size is not changed during the iteration.

For some problem, it is reasonable to chose Adaptive method in which Δt is dynamically changed. For example, in multiphase flow simulation using VOF, you can use this function to update the phase interface more efficiently.

Run the simulation



The average temperature on "point-0" change by time is as below:



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

Steel: density: 7753 kg/m3; Cp: 480J/(kg.K)

Thermal conductivity: 33W/(m.K)

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.-25

序号	学号	姓名	第一次作业	第二次作业	第三次作业	第四次作业	第五次作业	第六次作业	第七次作业
1	3117001033	朱星星	√	√		√	√	√	√
2	3117004015	万明佳	√	√		√	√	√	√
3	3117007047	侯岳显	√	√	√	√	√	√	
4	3117009028	万震		√	√	√	√	√	√
5	3117009037	毛柳浩		√	√		√		√
6	3117009038	盖博	√	√		√	√	√	√
7	3117009053	徐海涛	√	√	√	√	√	√	
8	3117009061	周王峥	√	√		√	√	√	√
9	3117011004	王冉	√		√	√	√	√	√
10	3117013020	方佳旗	√	√	√				√
11	3117016014	占涛涛	√	√	√		√	√	√
12	3117016020	徐华志	√	√	√	√	√	√	
13	3117016040	邓世培	√	√	√		√	√	√
15	3117017017	陈曦	√						
16	3117022005	戢坤池	√	√	√	√	√		√
17	3117307015	姜梦雨	√	√	√	√	√		√

18	3117307033	曾令午	√	√	√	√	√		√
19	3117307088	马彦达		√	√	√	√	√	√
20	3117307100	杨尚升	√	√	√	√	√		
21	3117307112	匡也	√	√	√	√	√		√
22	3117323003	梁楠	√		√	√	√	√	√
23	3117323021	李可	√	√	√		√		√
24	3417009005	崔天依	√	√	√	√	√		√
25	4117003145	徐丹	√	√	√			√	√
26	4117003151	毛红威	√	√	√	√	√		√
27	3117307039	王珊	√	√		√	√	√	√
28	3117307006	周尧	√	√	√	√	√	√	
29	3417009006	吴家荣	√	√	√				
30	4117016014	余亚雄		√	√			√	√
31	4117999084	KHUBAIB SYED MUHAMMAD		√	√	√	√	√	√
32	4117003147	王晓红	√	√	√	√	√	√	

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
辅讲 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25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 气膜冷却流动换热换热问题

湍流

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

顶盖驱动流动换热问题

Focus: **compared with previous examples, the focus of this example is that fluid flow is further considered and moving wall boundary condition is adopted.**

13.3 Lid-driven flow (homework of chapter 1)

Known:

An long solid plate with uniform temperature $T_{w1} = 80°C$ is moving with velocity u=0.1m/s at the top of a square cavity. The left and right walls of the cavity are adiabatic (绝热), while the temperature of bottom wall is fixed at $T_{w2} = 100°C$. The effect of gravity is neglected.

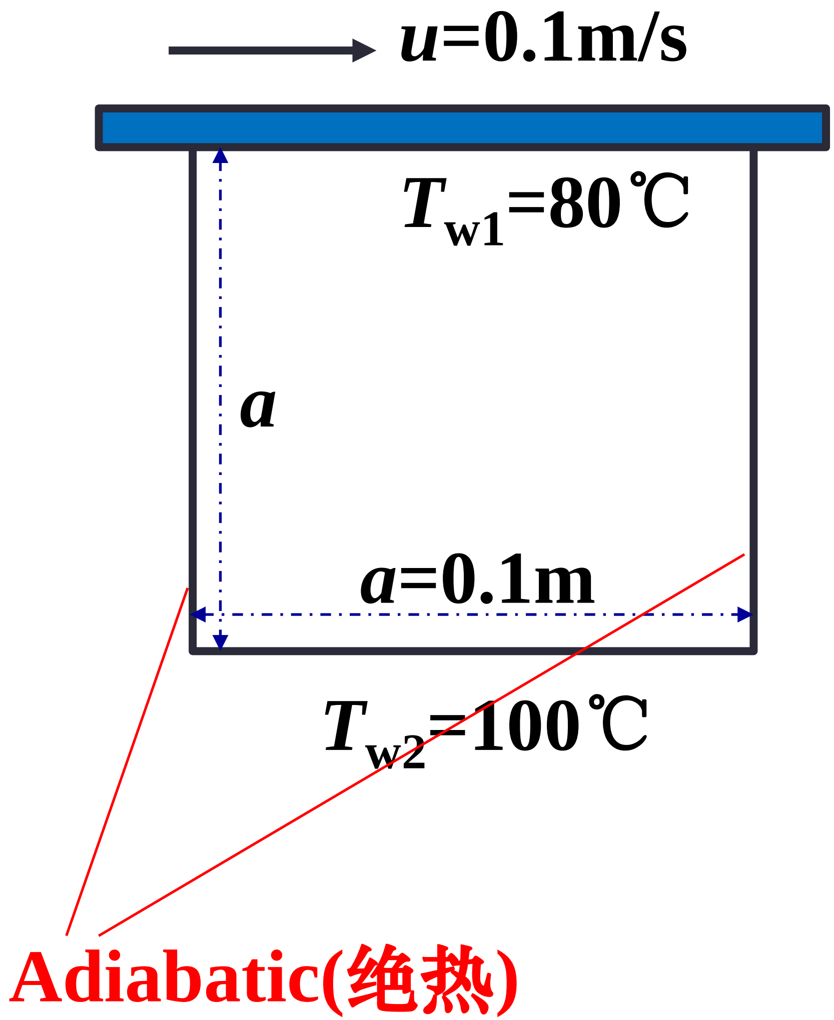


Fig.1 Computational domain

Find: velocity and temperature distribution

Solution:

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

Momentum: $$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)$$

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

Energy: $$\frac{\partial(\rho C_p uT)}{\partial x}+\frac{\partial(\rho C_p vT)}{\partial y}=\lambda\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)$$

We should estimate Re to determine laminar or turbulent state.

Know:

$u_{max} = 0.1\mathrm{m/s}$,l= 0.1m, $\nu = 22.1\mathrm{E}-6\mathrm{m^2/s}$

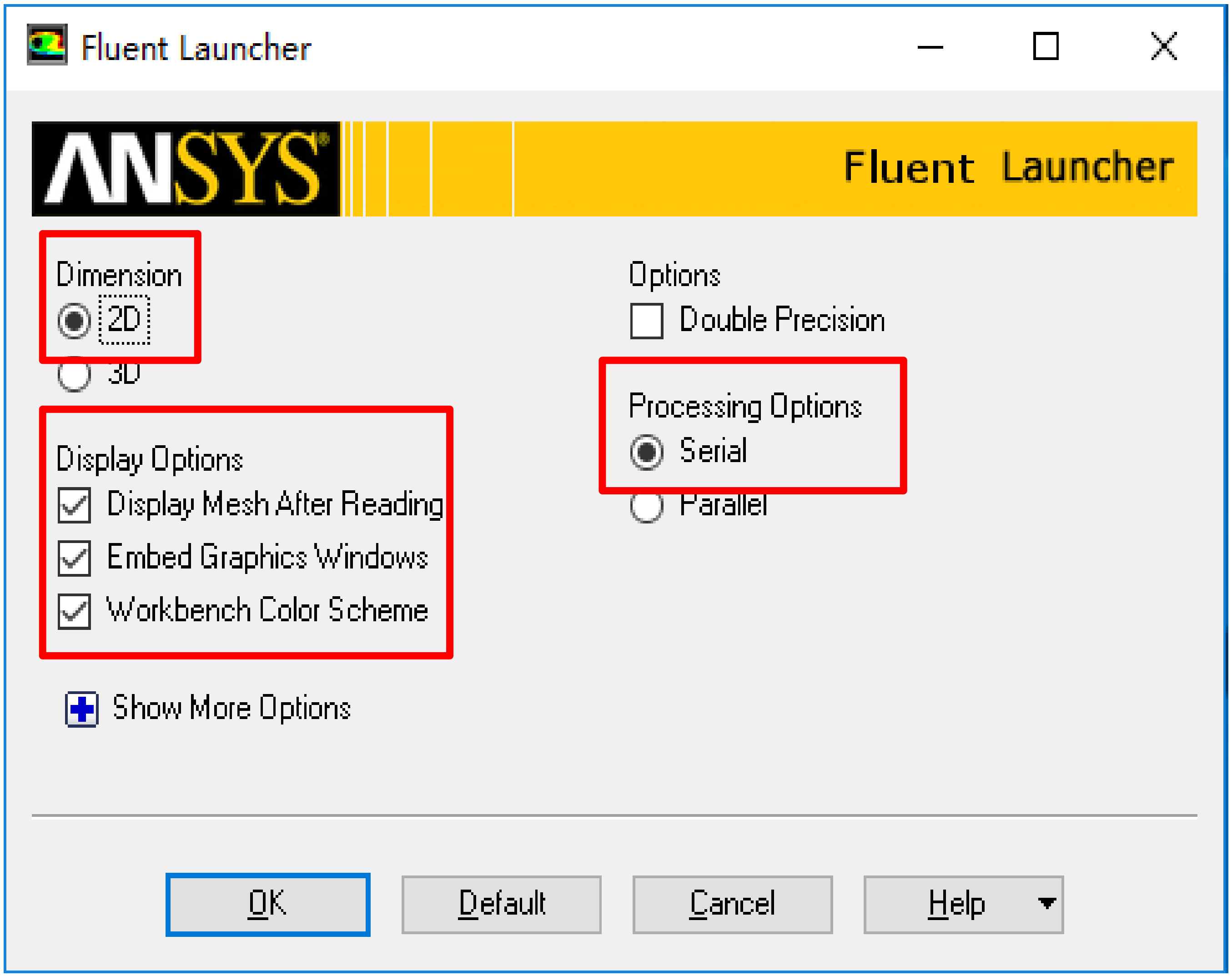
$$Re = \frac{ul}{\nu} = 452$$

Laminar flow

Remark: in this problem, we just take into account the forced convection. Nature convection is neglected. You can further study the effects of nature convection!

Start the Fluent software

Choose 2-Dimension



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）“.msh”

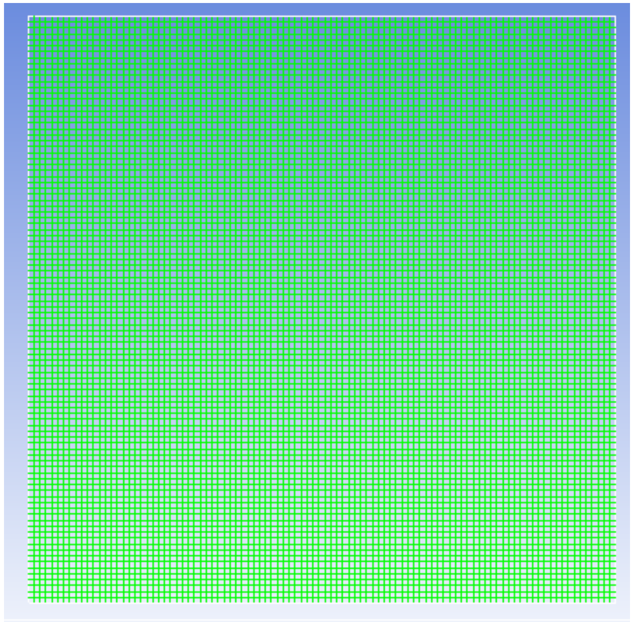
```
> Reading "E:\fluent-case\flow-5\flow2.cas"...
Done.
     9801 quadrilateral cells, zone  8, binary.
    19404 2D interior faces, zone  9, binary.
       99 2D wall faces, zone 10, binary.
       99 2D wall faces, zone 11, binary.
      198 2D wall faces, zone 12, binary.
    10000 nodes, binary.
    10000 node flags, binary.

Building...
     mesh
     materials,
     interface,
     domains,
        mixture
     zones,
        fixed-wall
        bottom-wall
        move-wall
        int_solid
        fluid
Done.
```

1st step: Read and check the mesh

Mesh→Check/Report quality

- Check the quality and topological information of the mesh

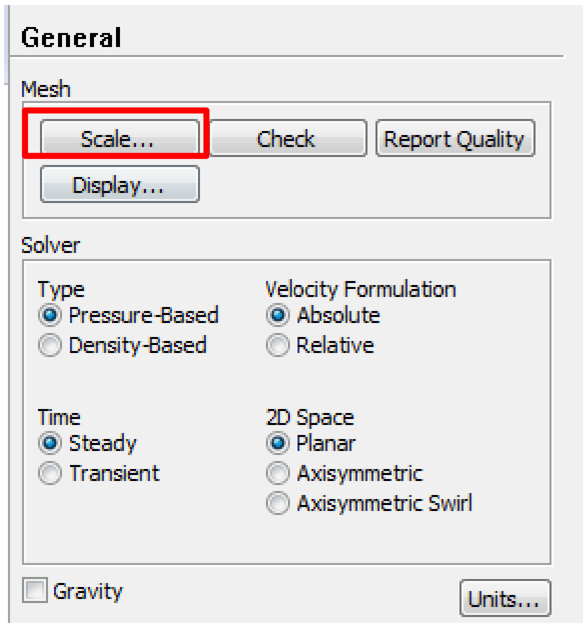
```
Domain Extents:
  x-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
  y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
Volume statistics:
  minimum volume (m3): 1.020304e-06
  maximum volume (m3): 1.020304e-06
    total volume (m3): 1.000000e-02
Face area statistics:
  minimum face area (m2): 1.010101e-03
  maximum face area (m2): 1.010101e-03
Checking mesh..........................
Done.
```

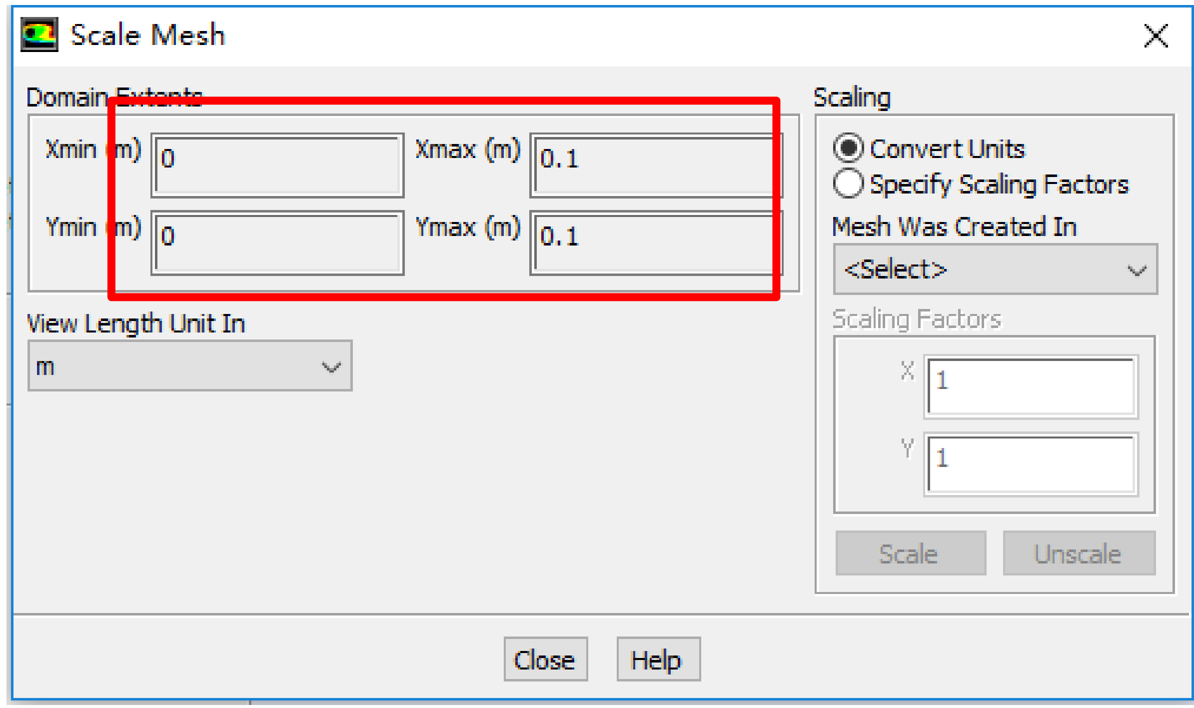


```
Mesh Quality:
 Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality =  1.00000e+00
Maximum Aspect Ratio =  1.41422e+00
```

2st step: Scale the domain size

General→Scale

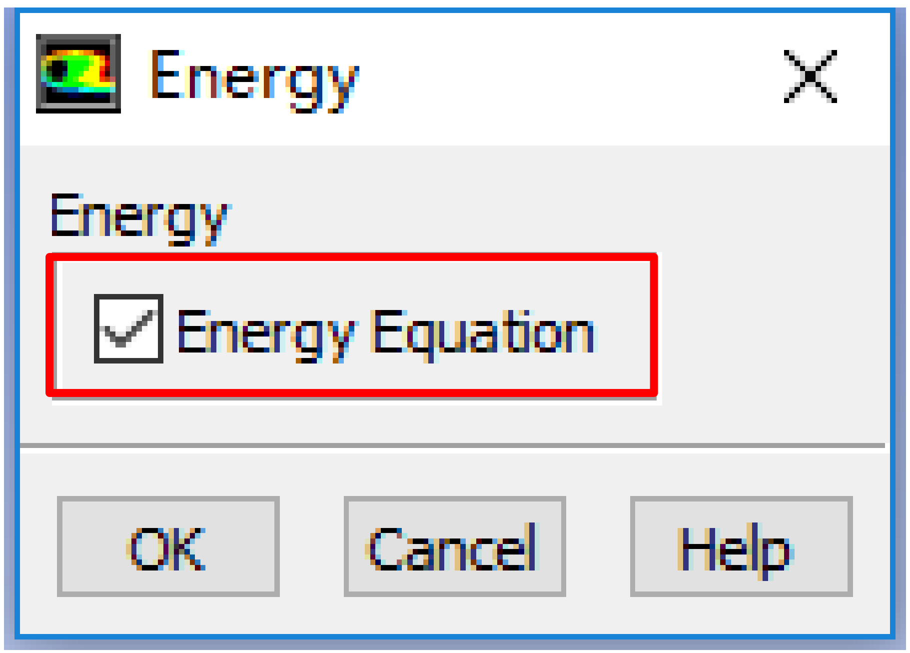


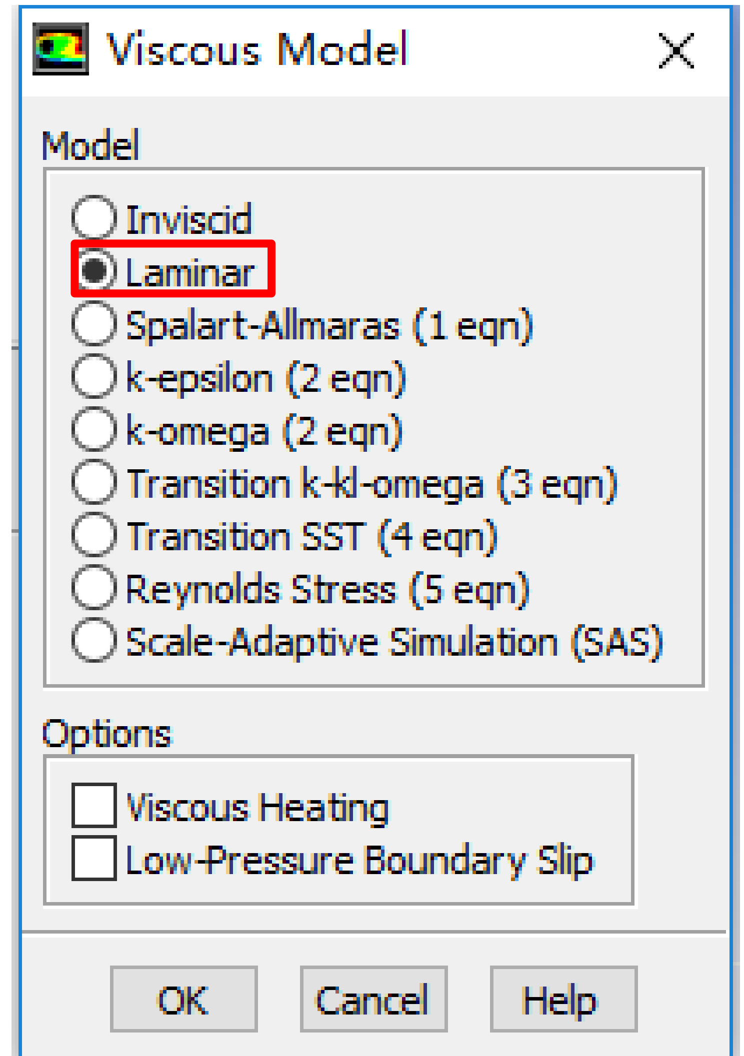


The mesh is generated in ICEM with unit of m. So we do not need to scale the size for this problem.

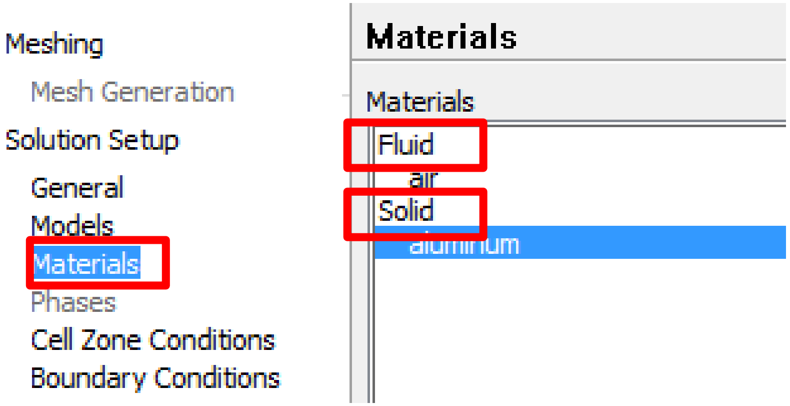
Step 3: Choose the physicochemical model

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

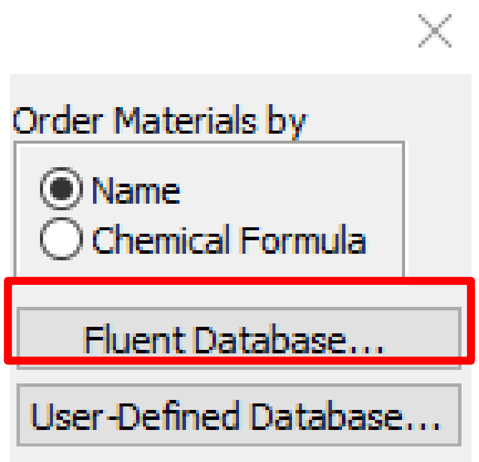




Step 4: Define the materials



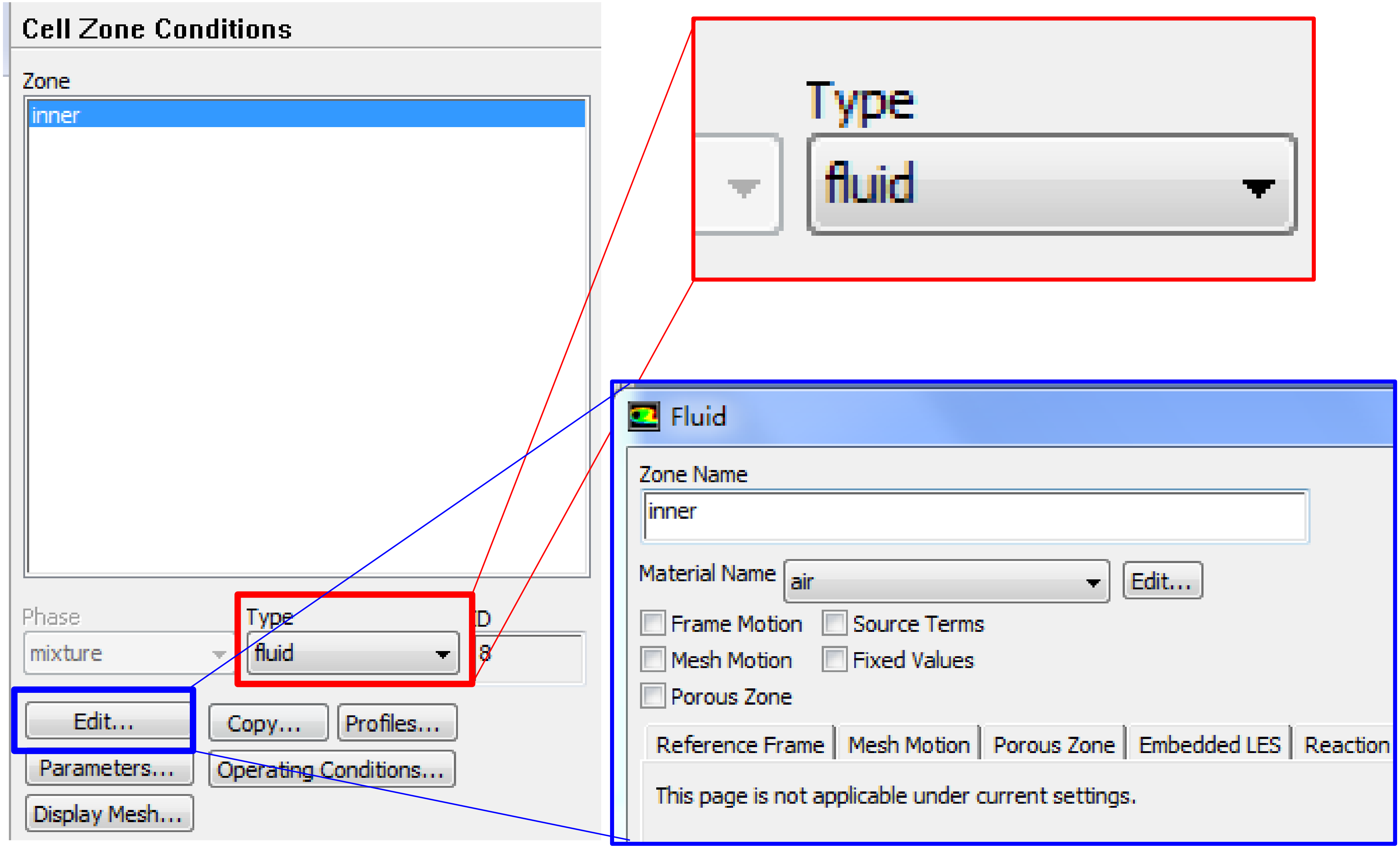
Click "Fluid" or "Solid" or select the "create/edit"



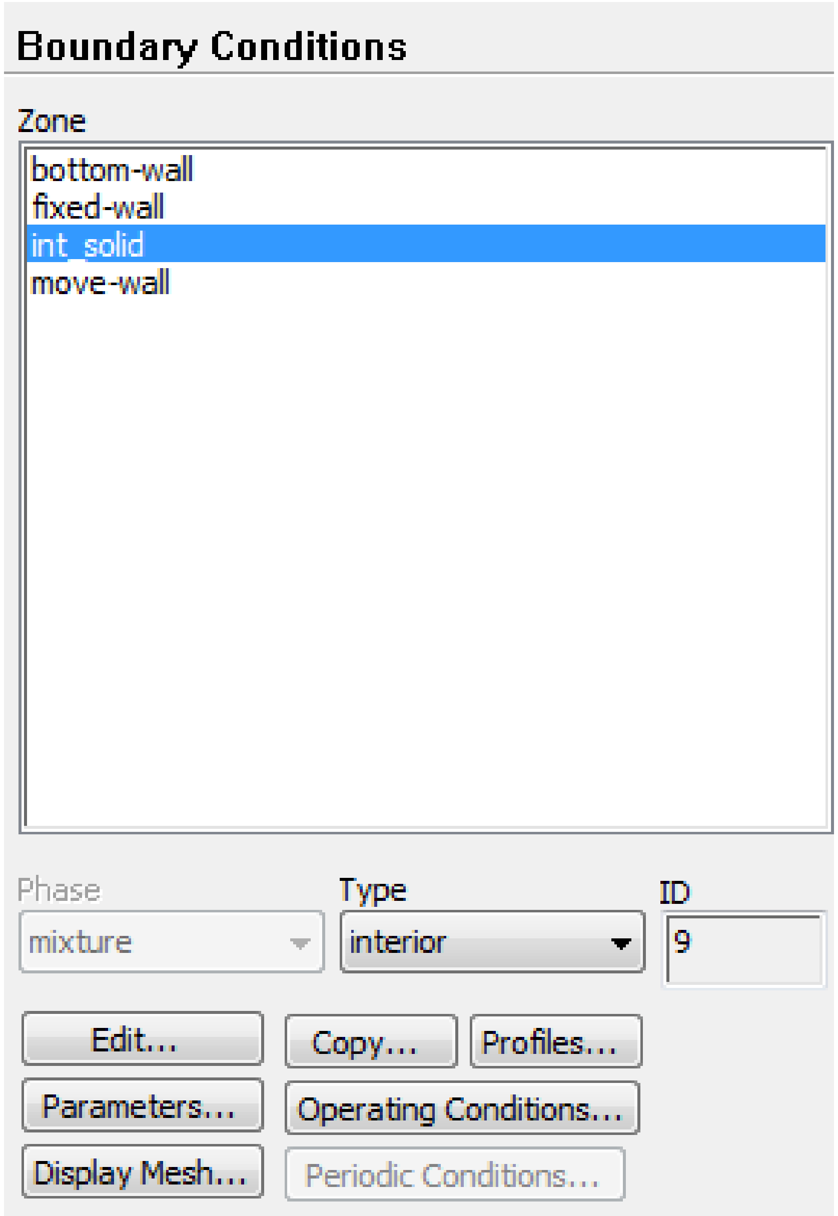
Fluent provide a lot of materials in its database. Usually, You can find the material you need in the database.

However, it will happen that the material you need is not in the database. You can input it manually.

5st step: Define the cell-zone condition



6st step: Define the Boundary conditions

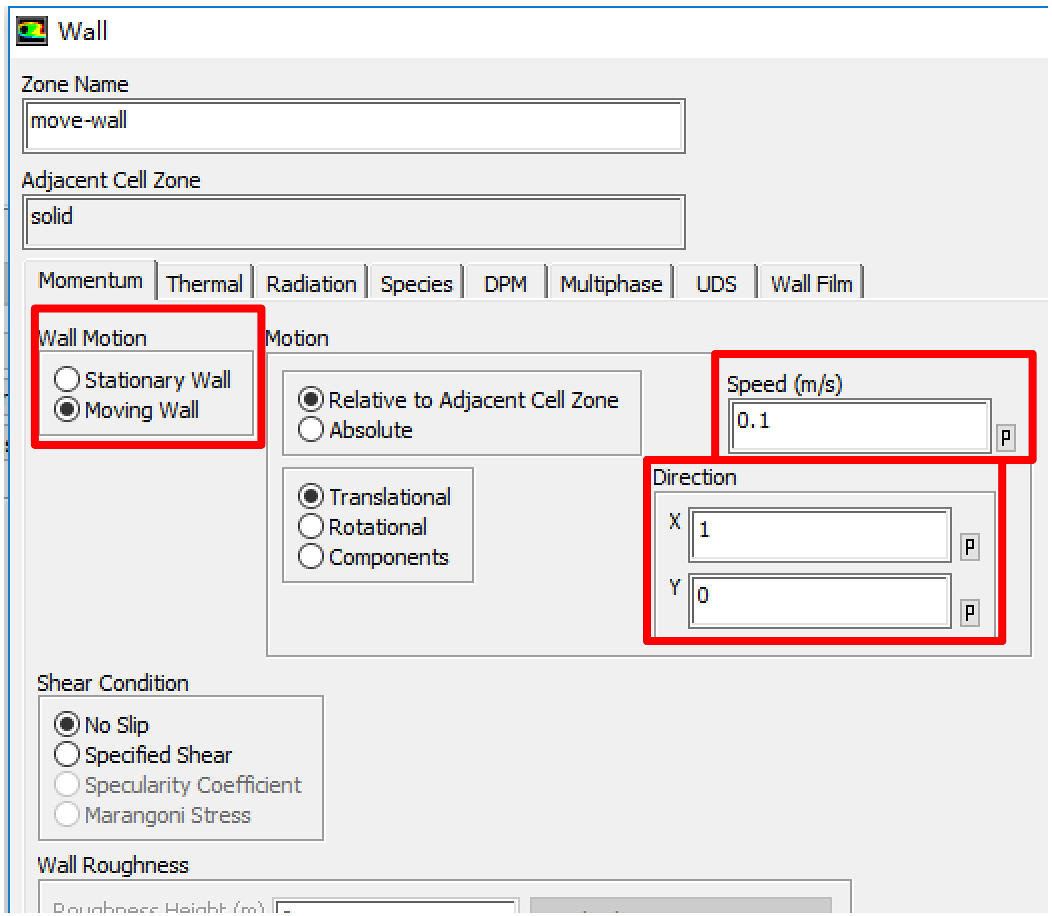


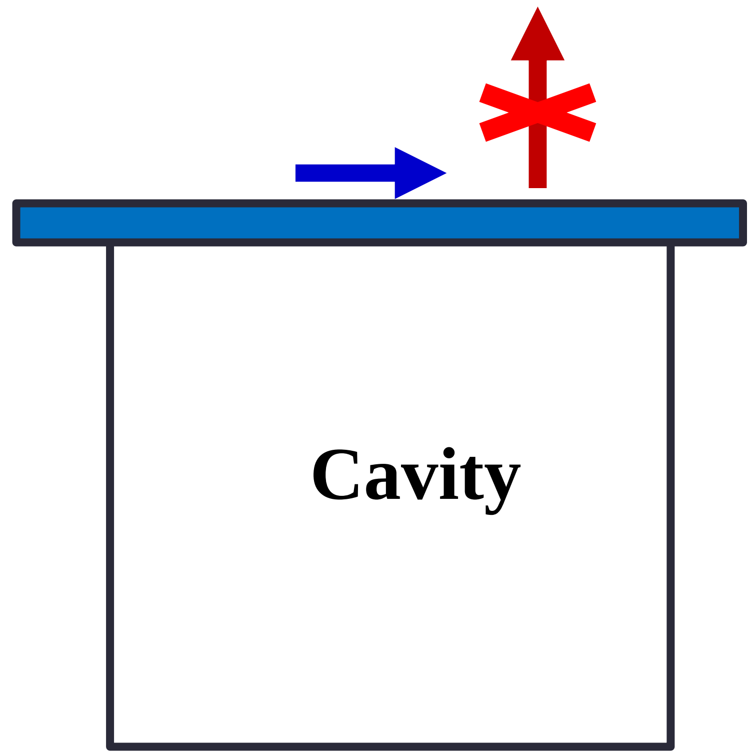
The bottom wall is not moving and its temperature is 80°C. The left and right wall is adiabatic.

All these boundary conditions are easy to set in Fluent.

The top wall is moving. We will discuss it in detail.

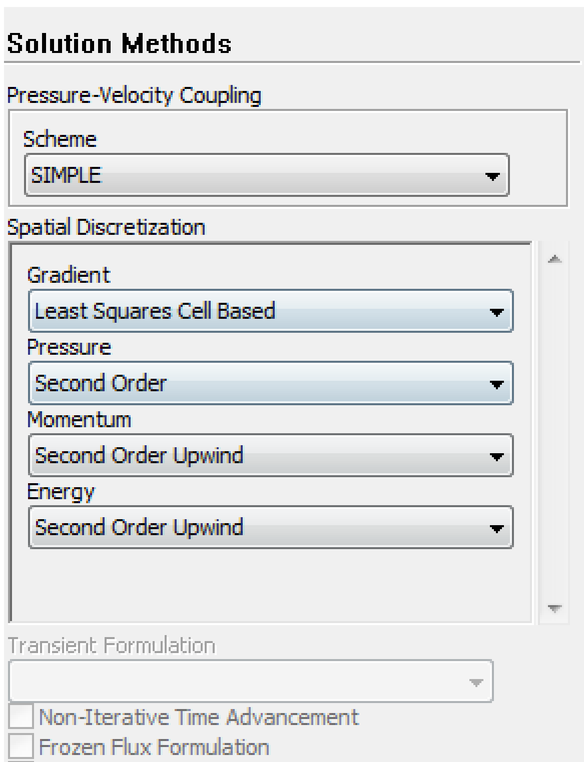
"Moving wall" is used to include tangential (切向) motion of the wall. This function cannot be used to include the normal (法向) motion of a wall.





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.

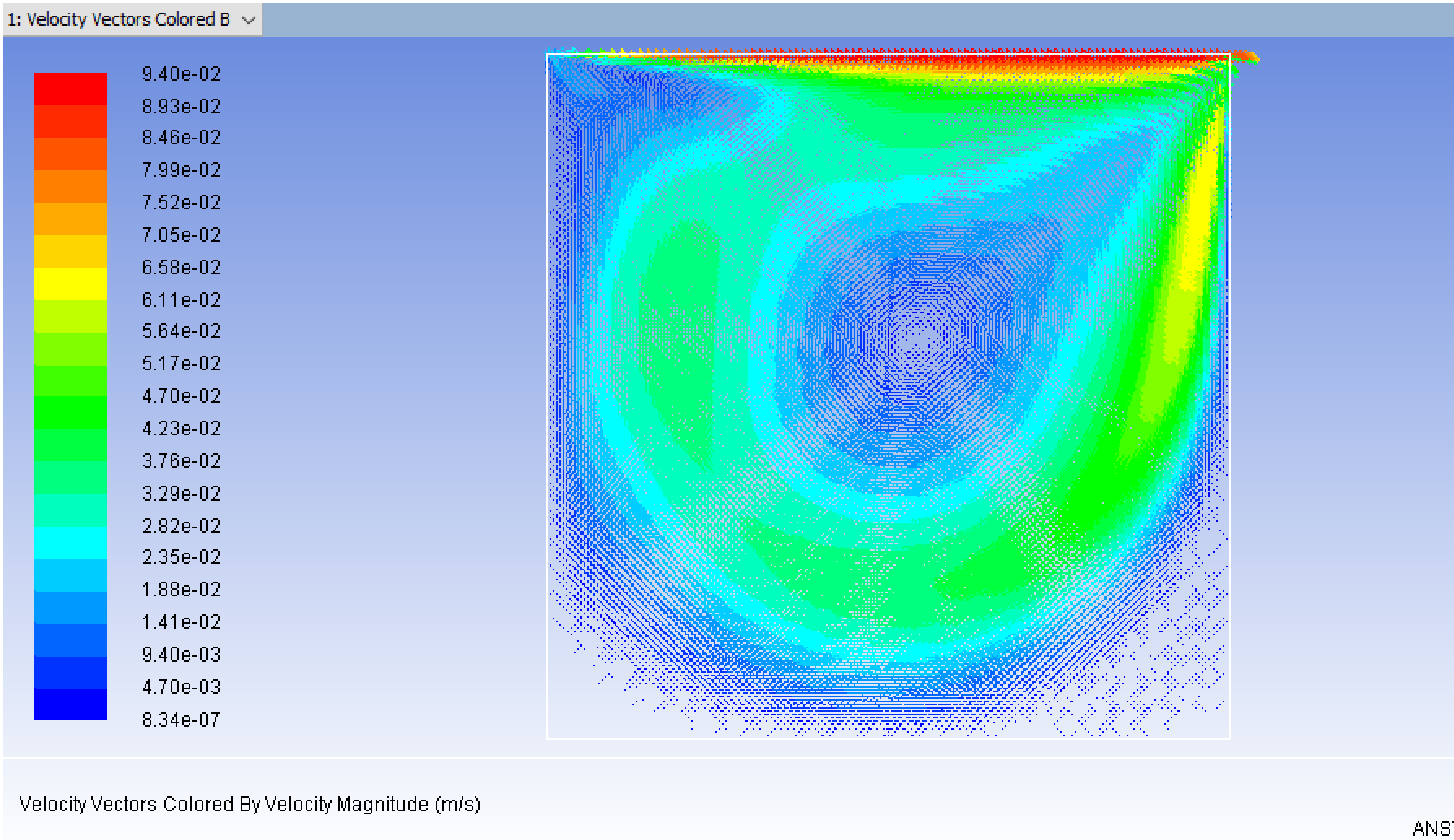
8st step: Initialization

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

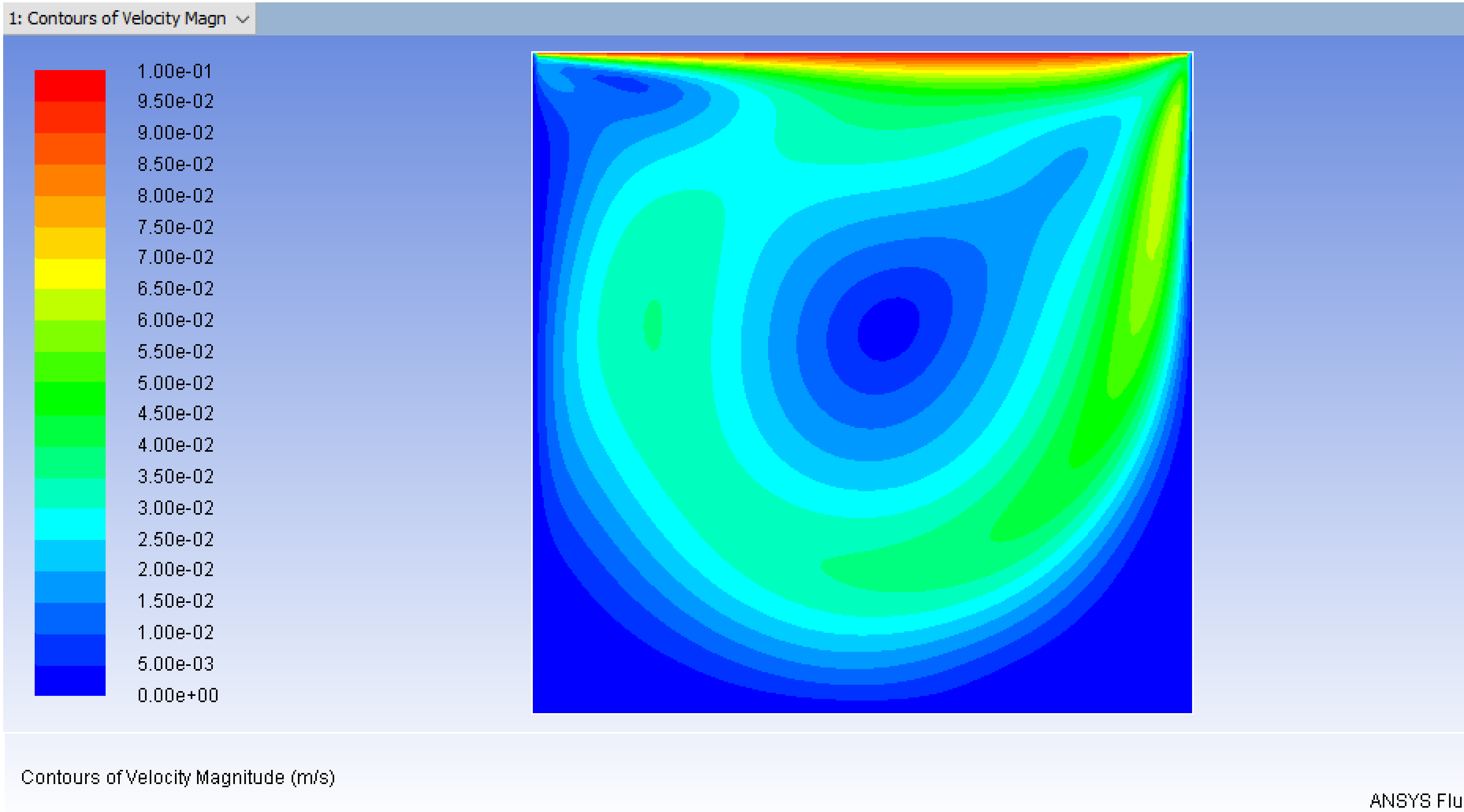
Step 9: Run the simulation

Step 10: Post-processing results

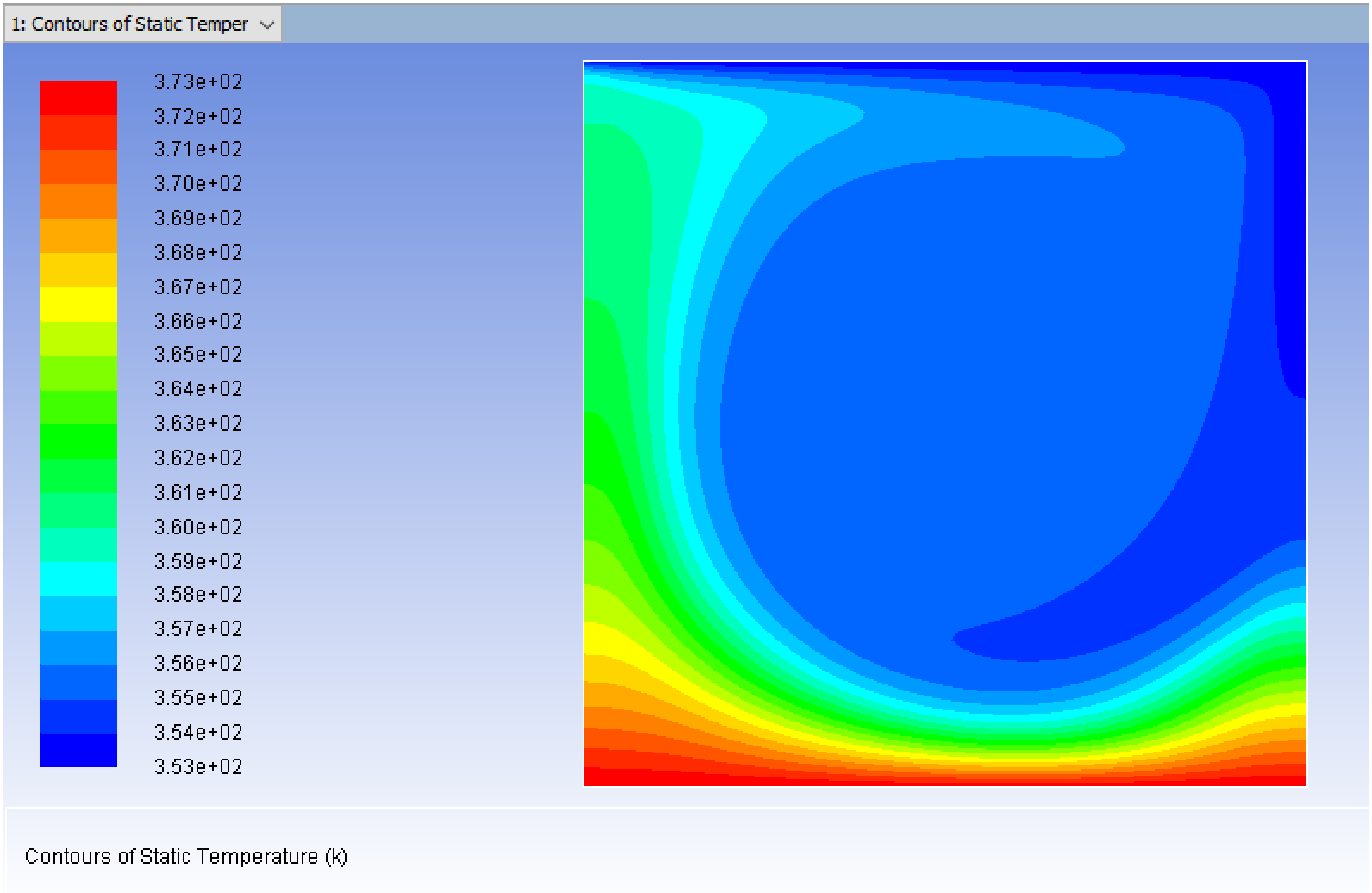
Velocity Vector



Velocity magnitude



Temperature



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.-25

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

辅讲 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25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

13.2 非稳态圆球冷却问题

导热问题

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 多孔介质流动换热问题

多孔介质流动

13.7 气膜冷却流动换热换热问题

湍流

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

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℃ flows into the inlet of a micro channel with rectangular ribs (MC-RR) with velocity u=0.1m/s. The side walls of MC-RR are 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 and natural convective heat transfer from the micro channel heat sink.

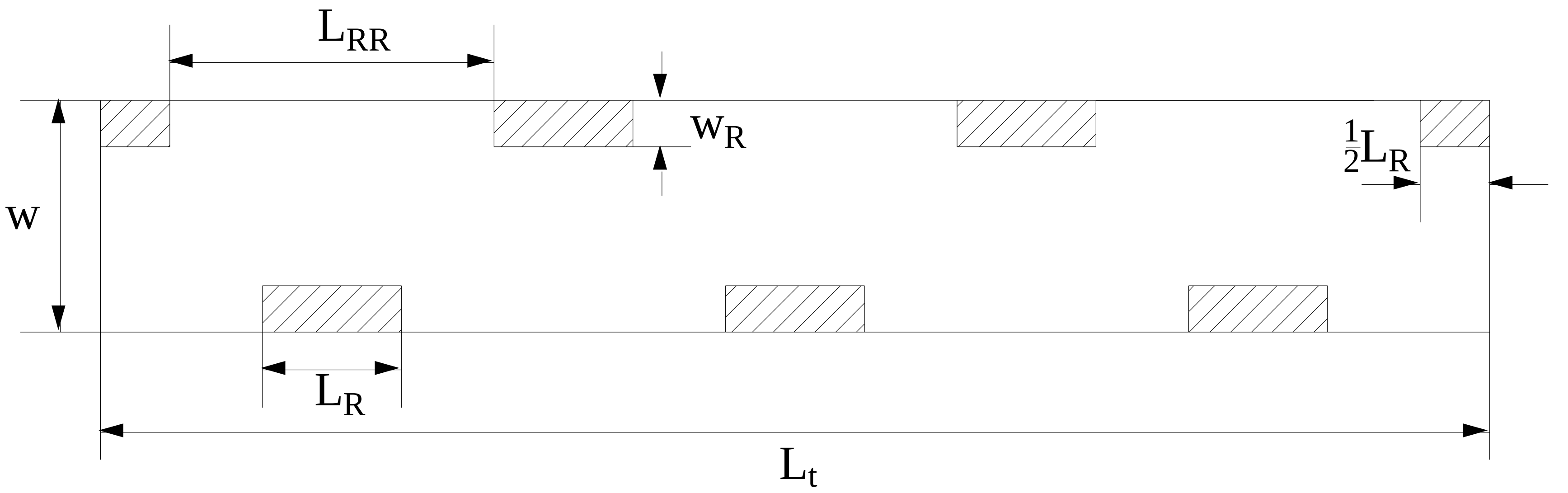


Fig.1 Computational domain

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 and velocity distribution in the domain.**

Governing equations:

Continuity equation:

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

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:

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

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:

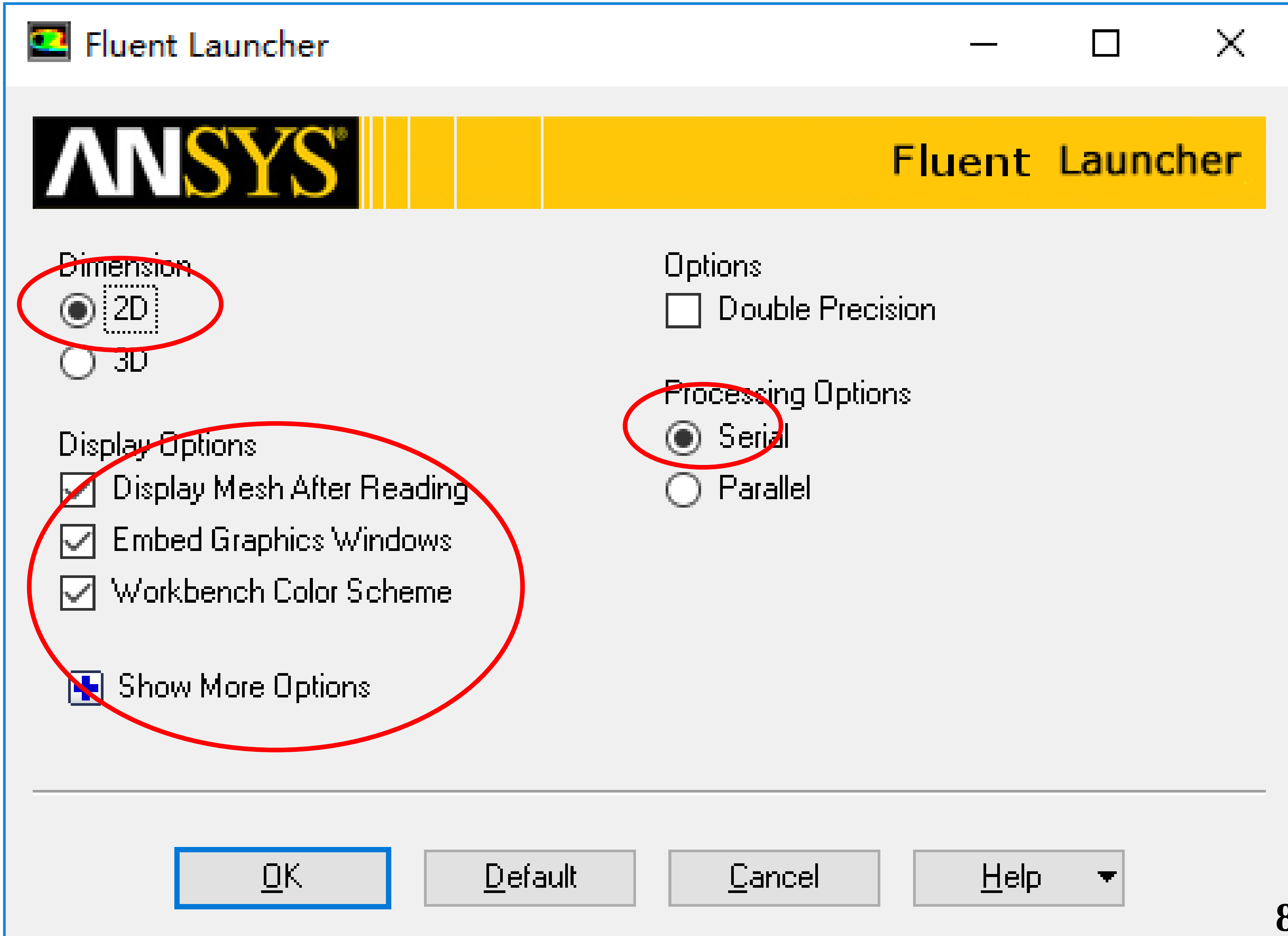
$$0=k_s\left(\frac{\partial^2 T_s}{\partial x^2}+\frac{\partial^2 T_s}{\partial y^2}\right)$$

where T_s is solid temperature and k_s is solid thermal conductivity

Boundary condition:

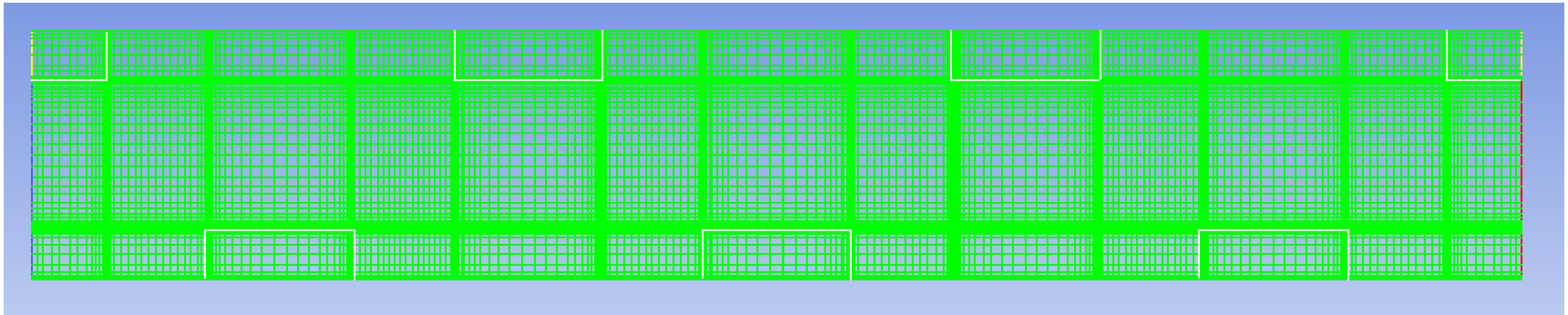
1.	channel inlet $x=0$ $u=u_f$ For fluid $T_f=T_{in}=293.15\text{K}$
2.	Channel outlet $x=3\text{mm}$ $P_f=P_{out}=1\text{atm}$
3.	fluid/solid surface $u=v=0$ $-k_s(\frac{\partial T_s}{\partial n})=-k_f(\frac{\partial T_f}{\partial n})$ $T_f=T_s$ where (n) is the coordinate normal to the wall
4.	At side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

Start the Fluent software



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） “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

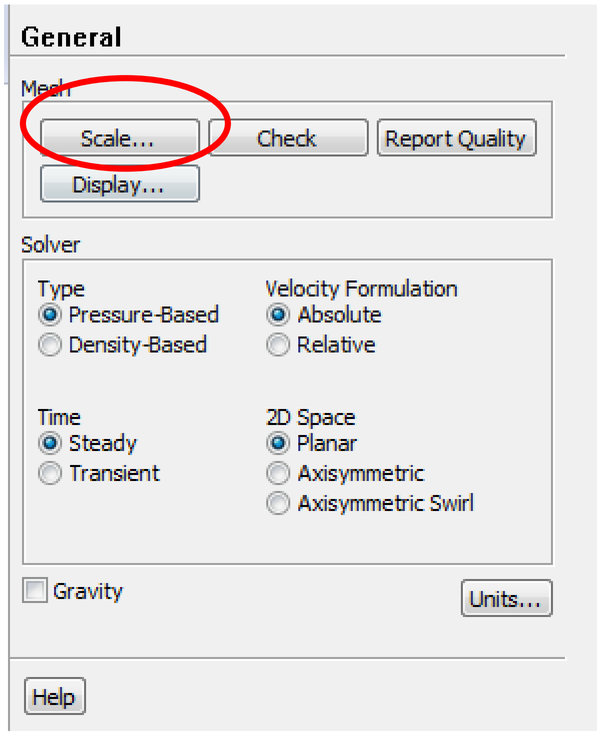
Mesh→Check

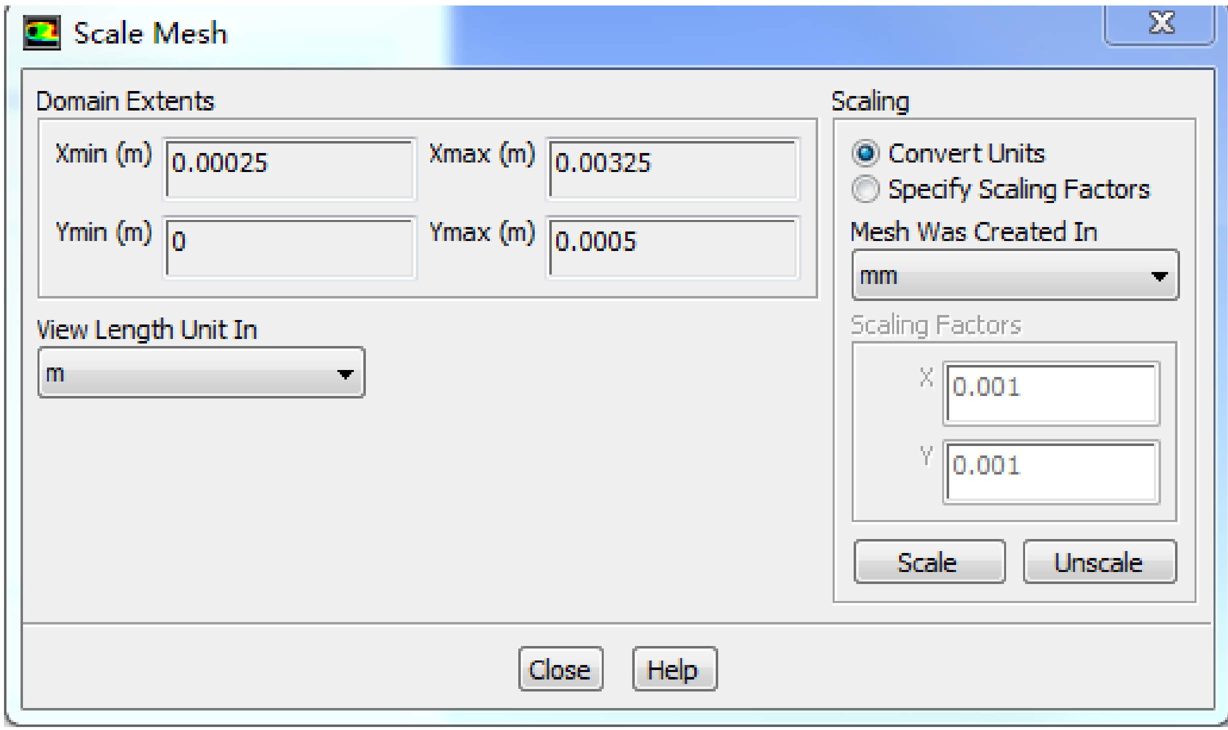
- Check the quality and topological information of the mesh

```
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

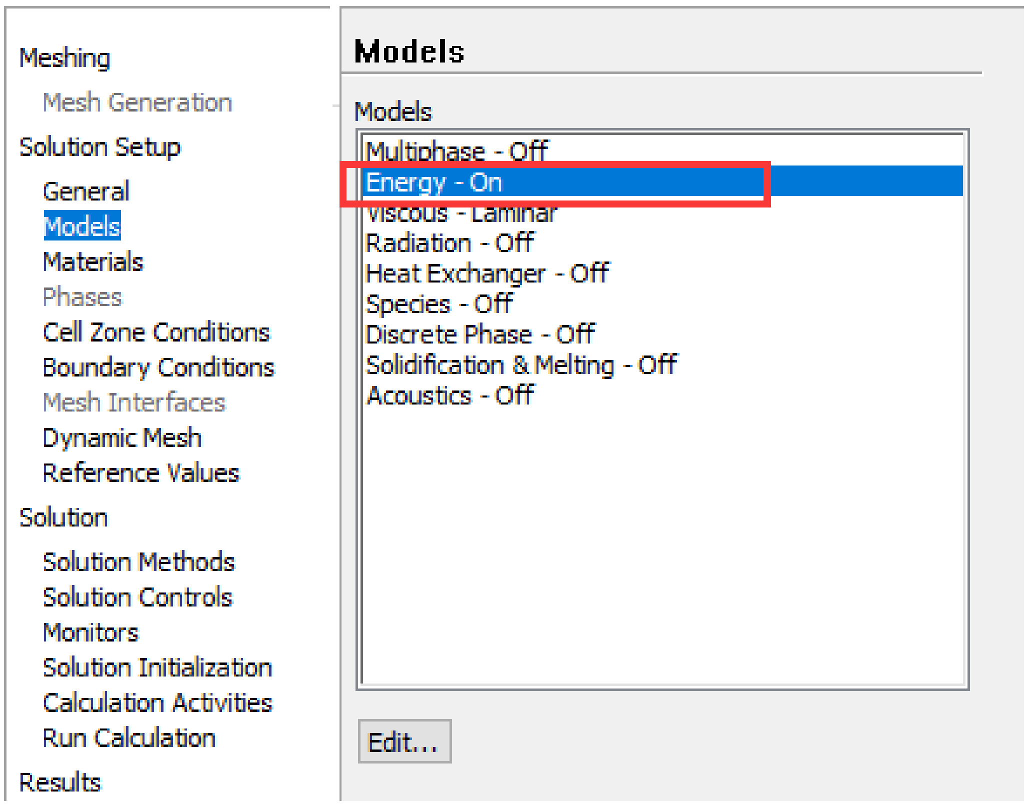




The mesh is generated in Fluent using unit of mm. Fluent import it as unit of m. Thus, "Convert units" is used.

Step 3: Choose the physicochemical model

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

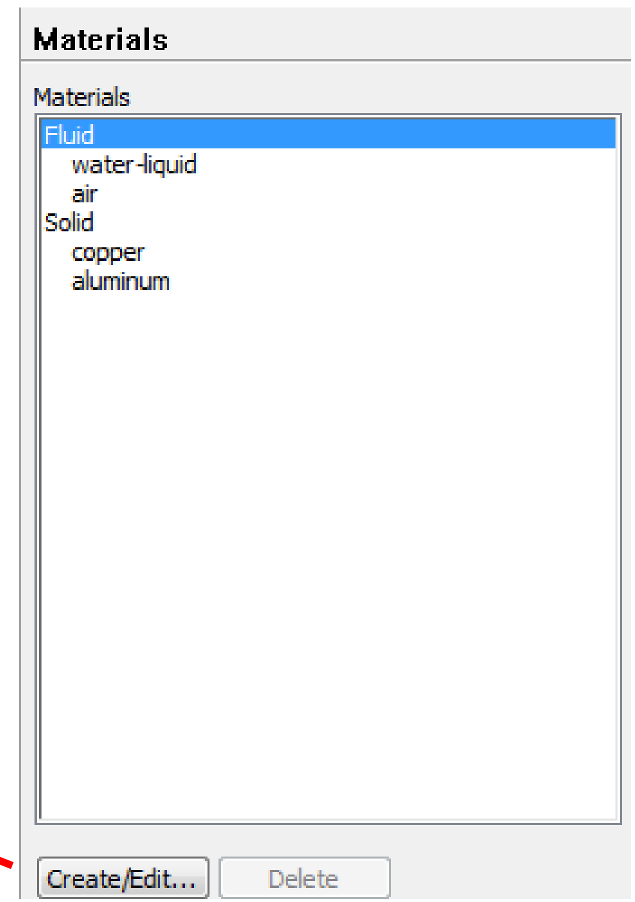


Example 4 is about single-phase laminar flow and heat transfer, so Energy model should be activated.

Step 4: Define the material properties

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

Solution Setup→Materials



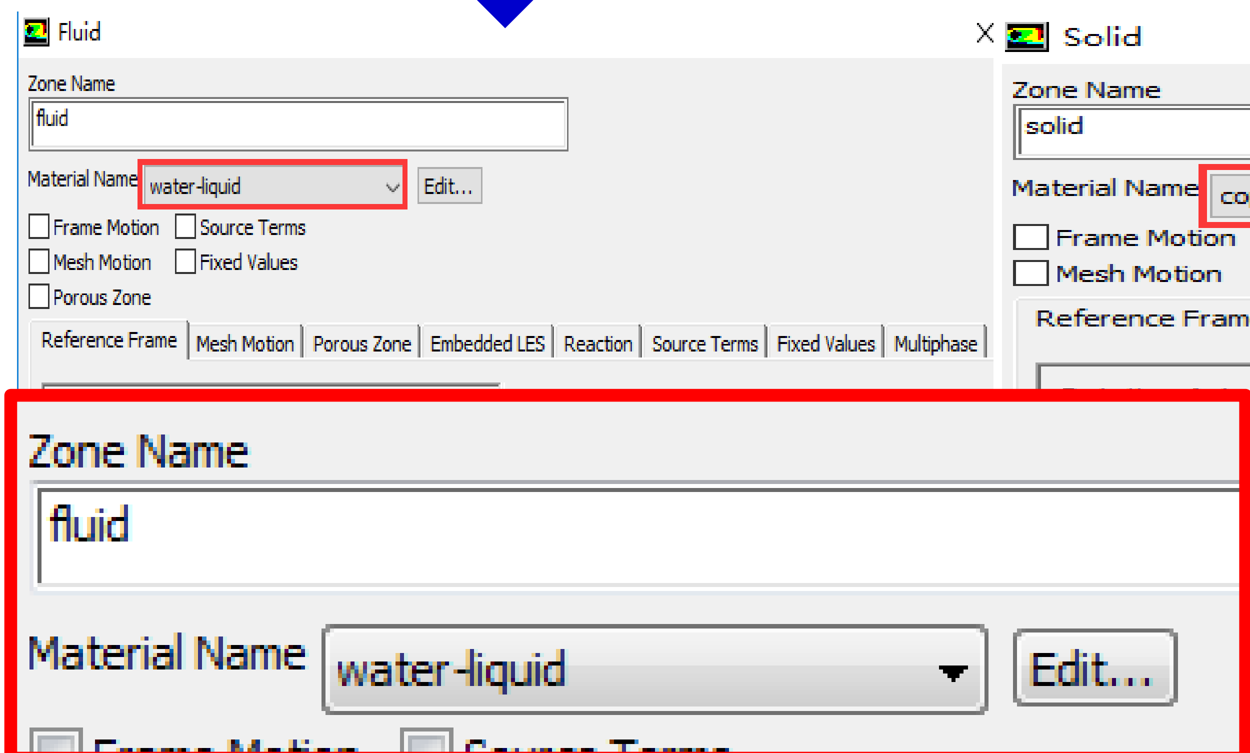
In Fluent, the default fluid is air and the default solid is Al.

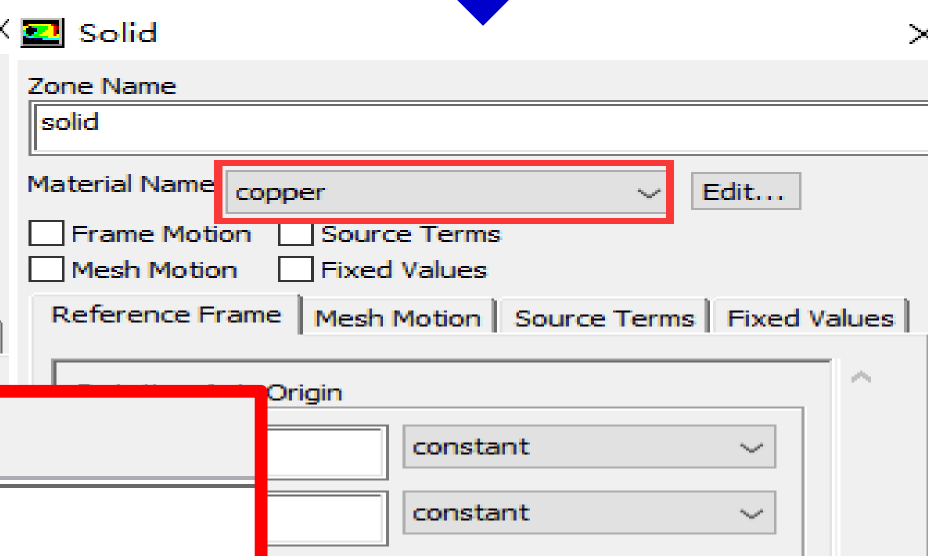
Click the Create/Edit button to add Water and Copper.

Step 5: Define zone condition

Solution Setup→Cell Zone Condition

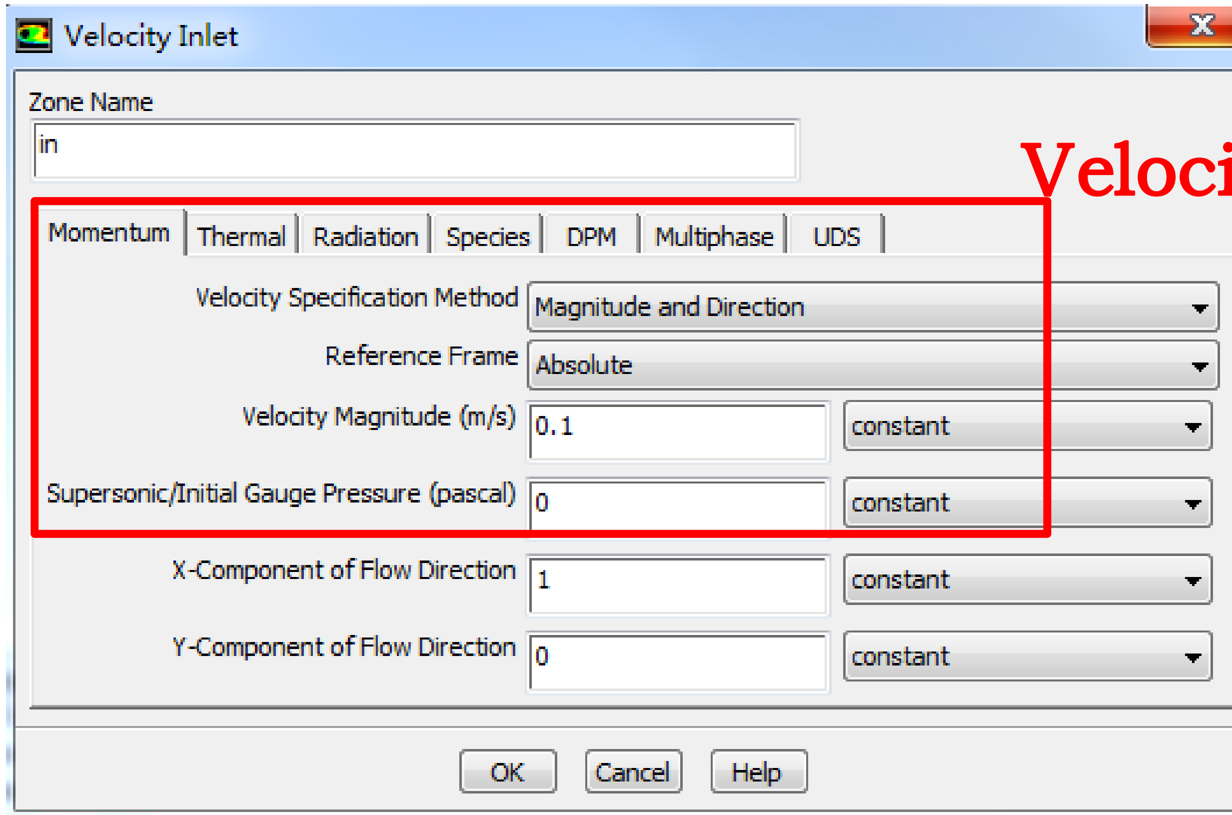
Choose water for Fluid zone **Choose copper for Solid zone**



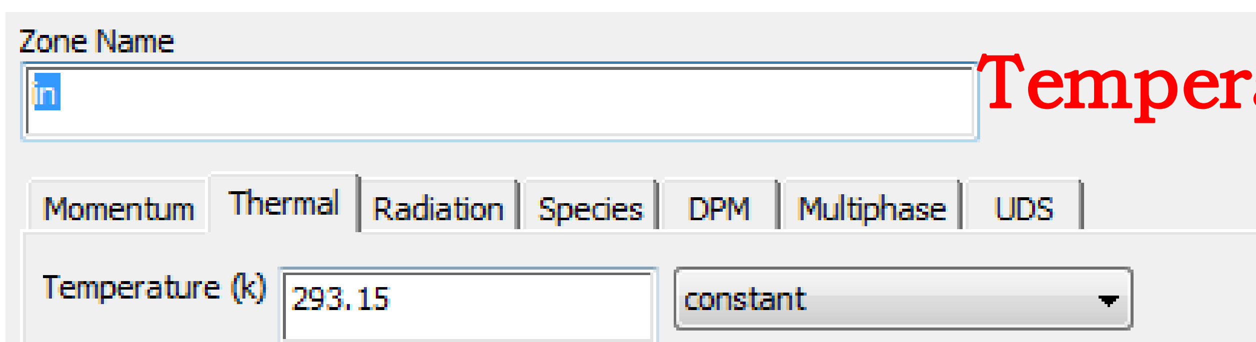


Step 6: Define the boundary condition

Inlet

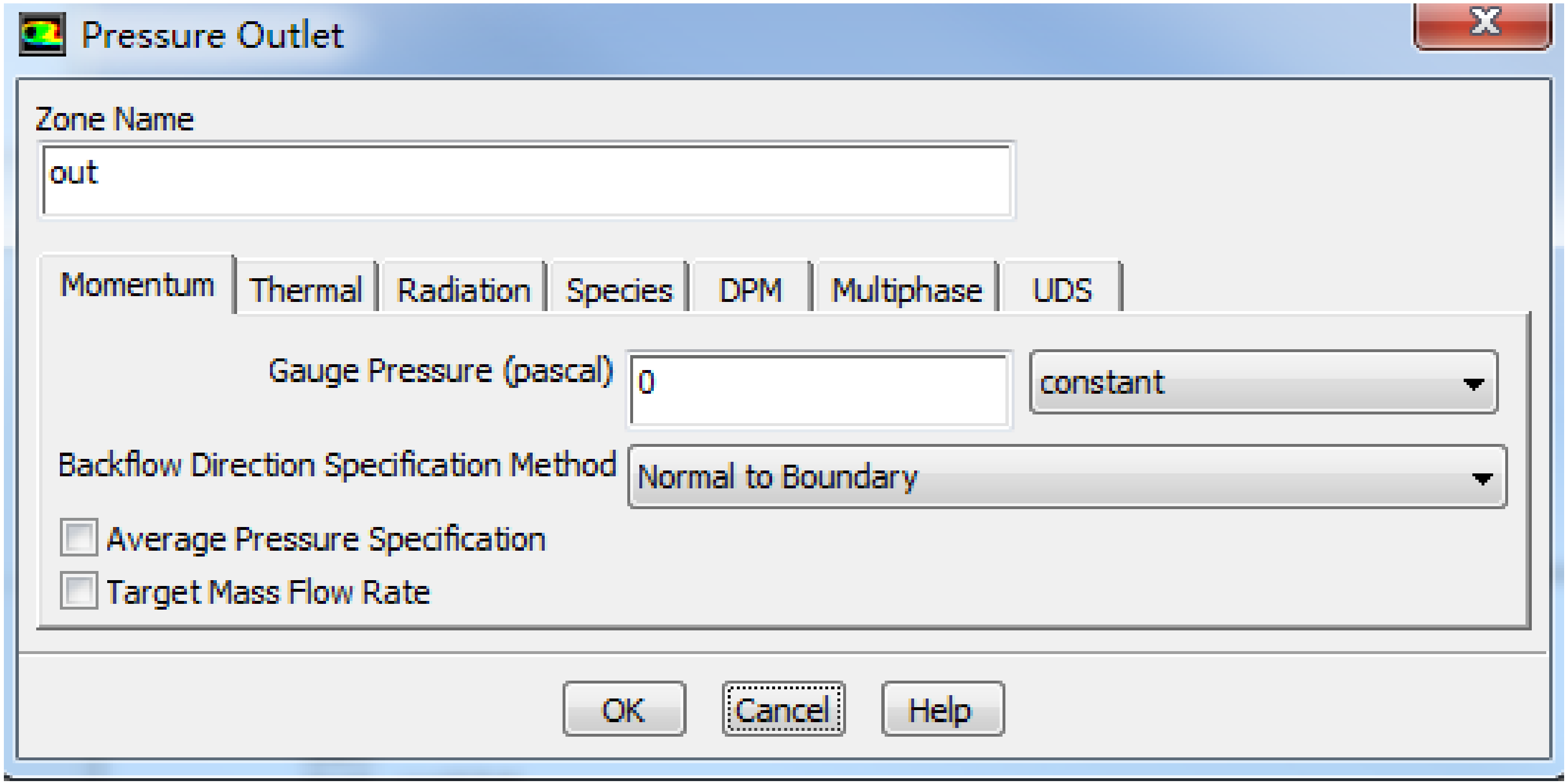


Velocity



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 $0.5\rho U^2$

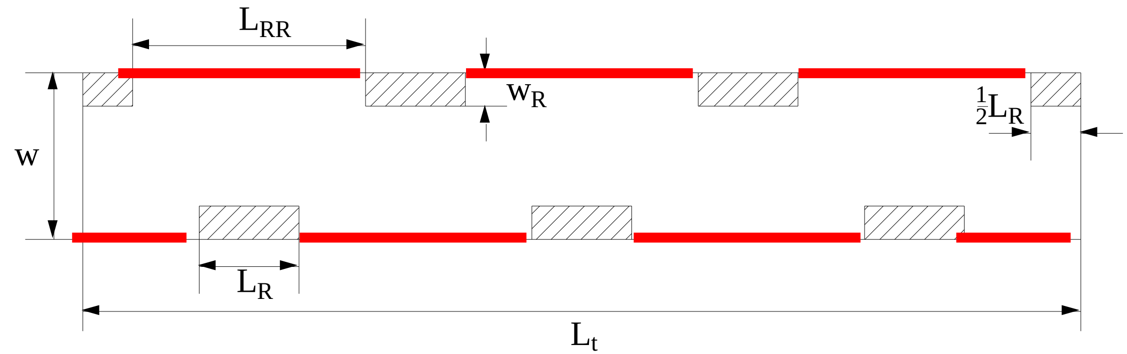
Is related to the velocity.

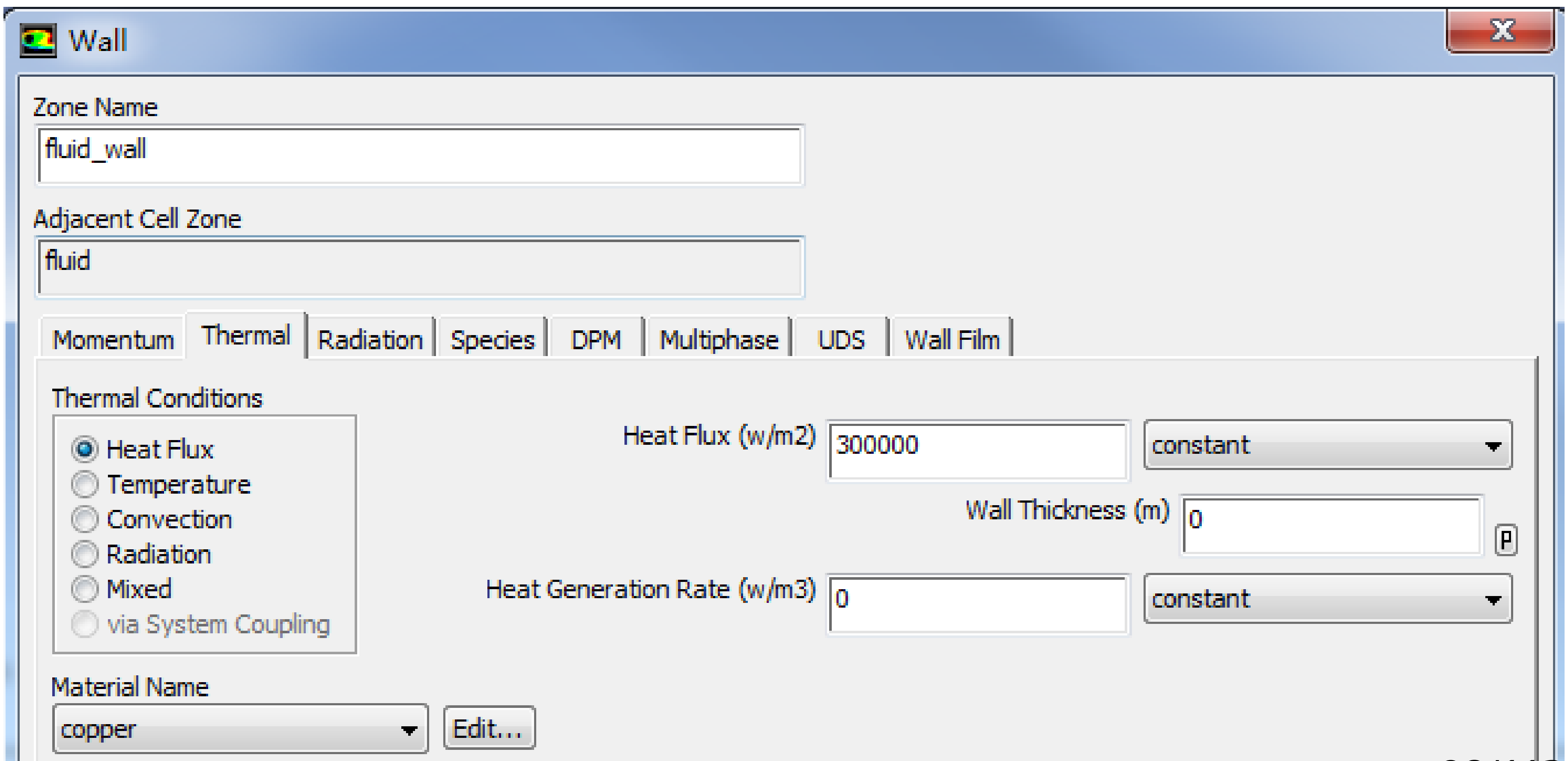
Total pressure (总压):

= Static pressure + dynamic pressure

Wall

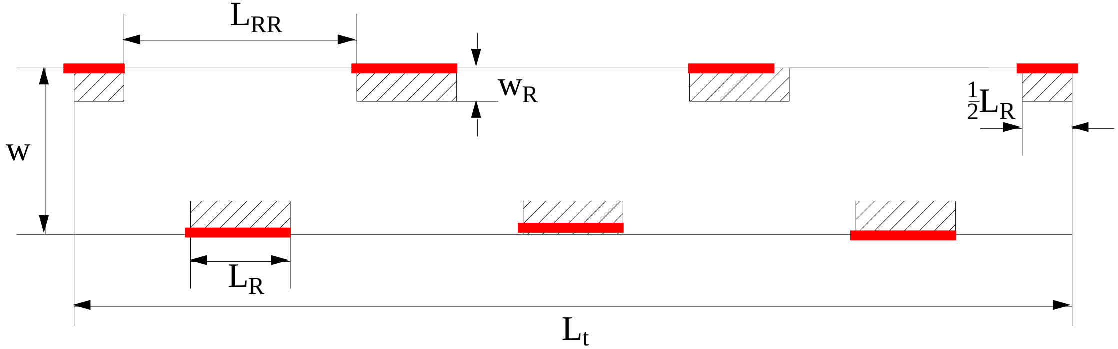
$$q = 30\mathrm{W/cm^2}$$

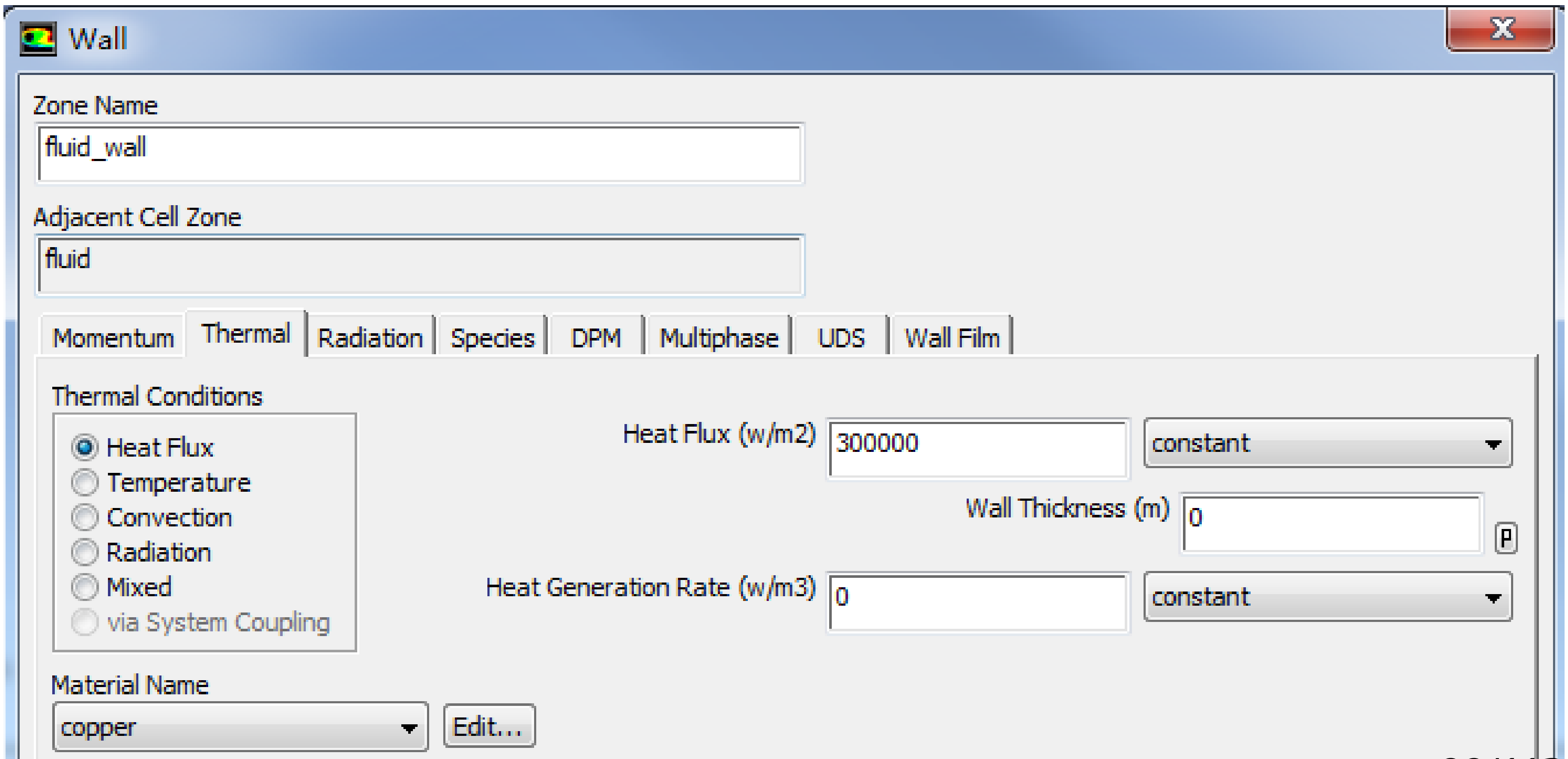




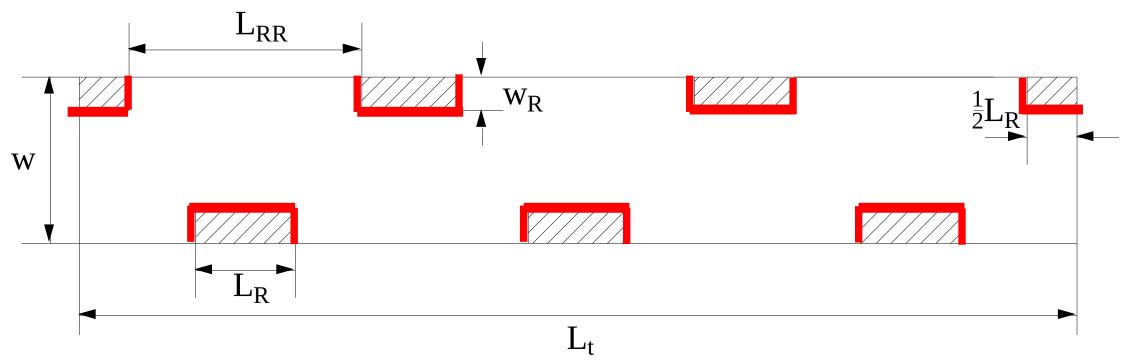
Wall

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





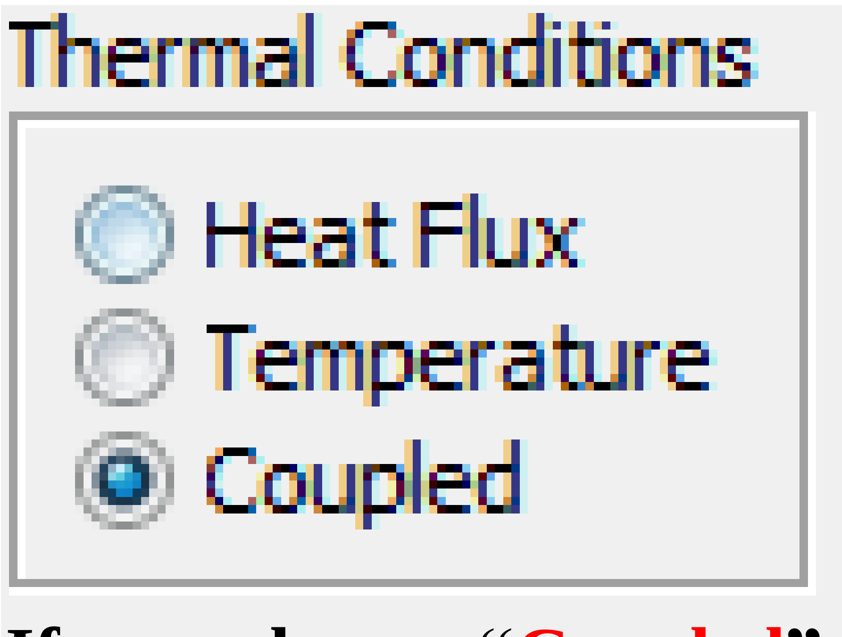
Fluid-solid interface



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.

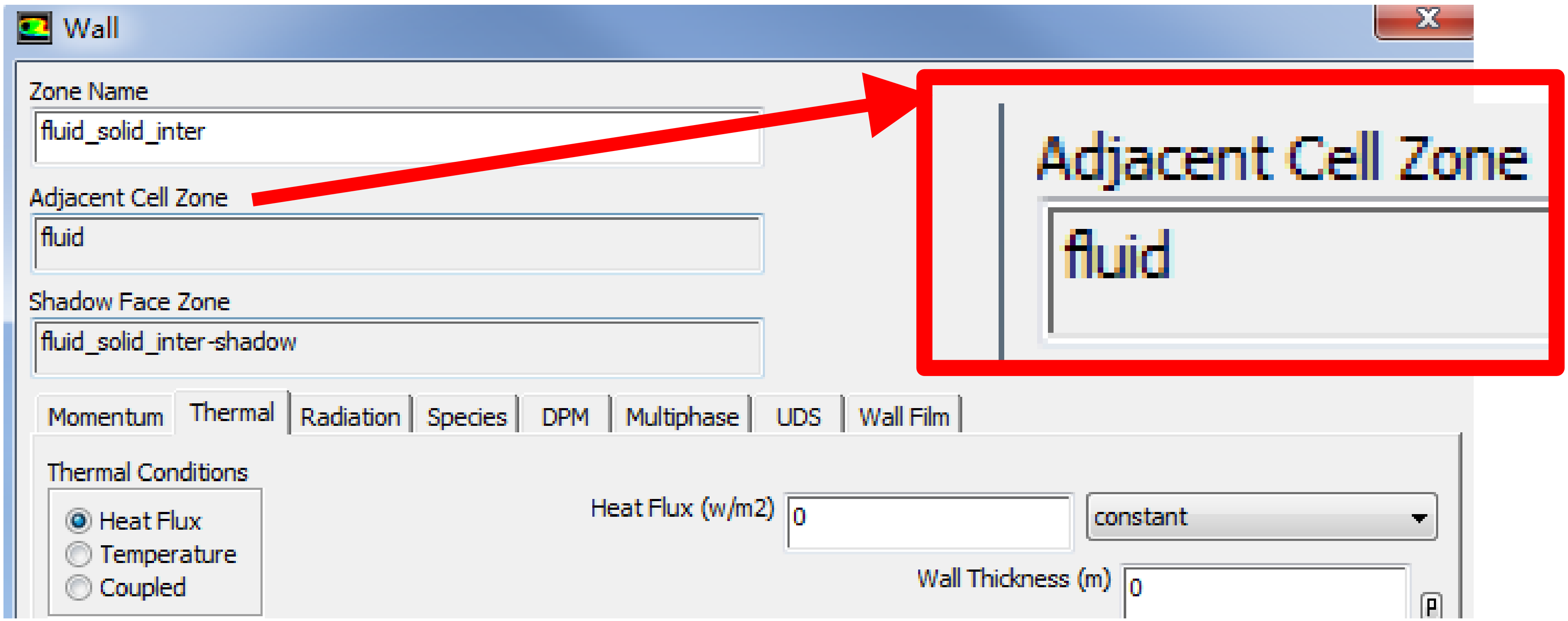
There are three options for the temperature boundary conditions of such "two-sided-wall".



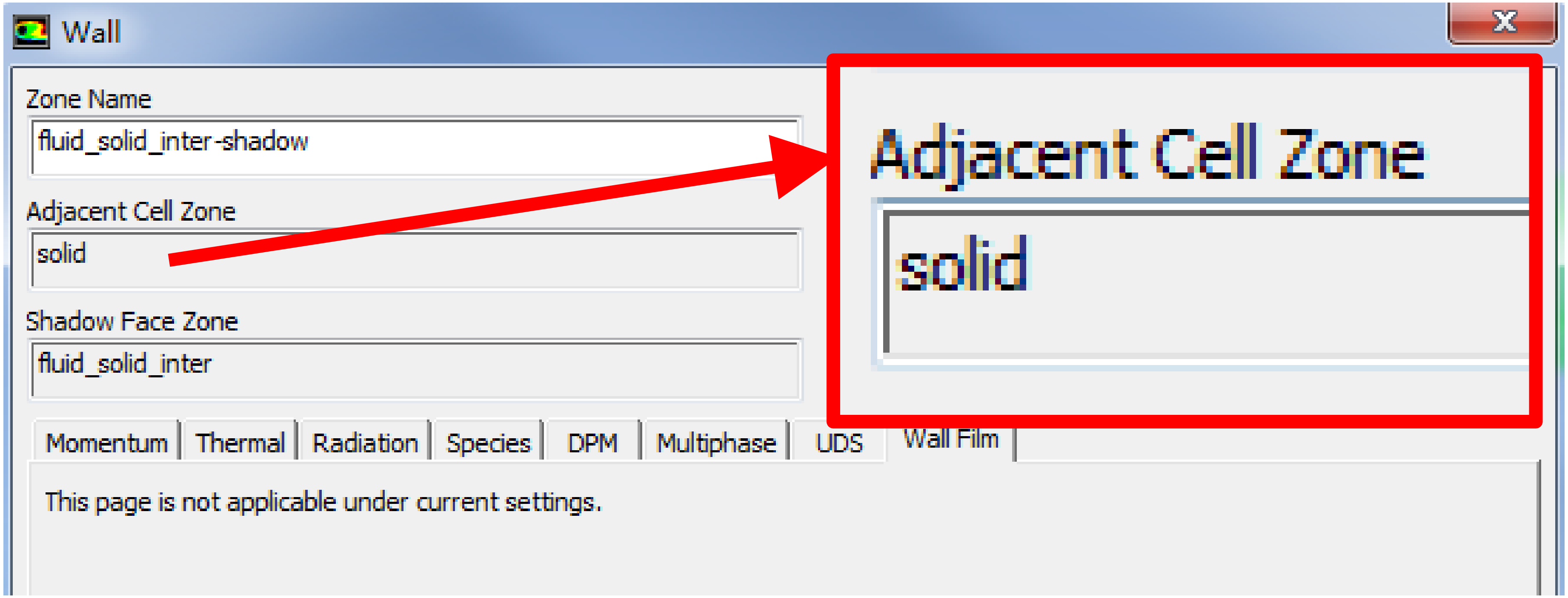
- Heat flux
- Temperature
- Coupled

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!

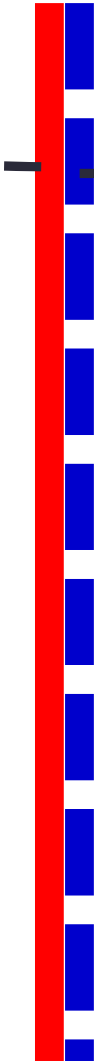


The adjacent cell zone of this shadow wall is solid!

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

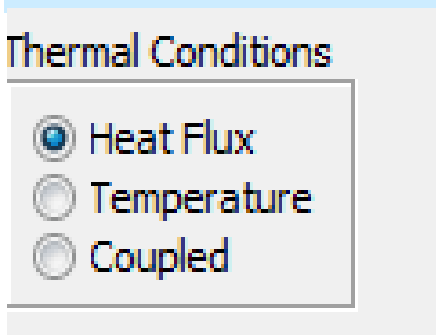
The original two side wall

Adjacent to fluid

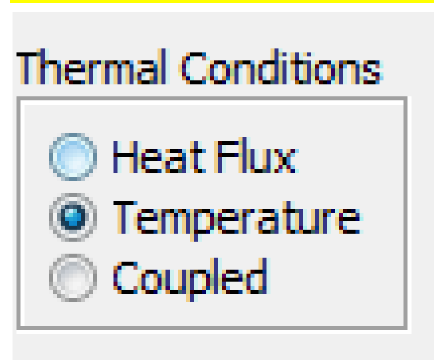


Adjacent to solid

Specify BC for the fluid side



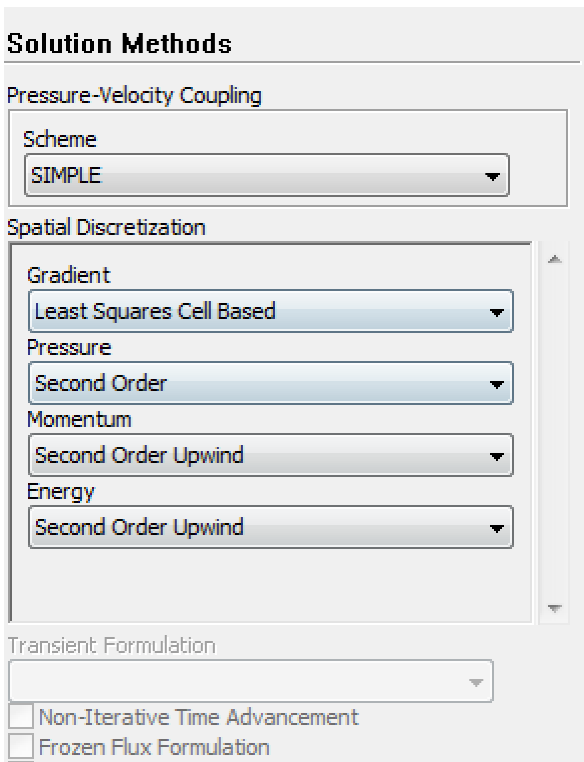
Specify BC for the solid side



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.

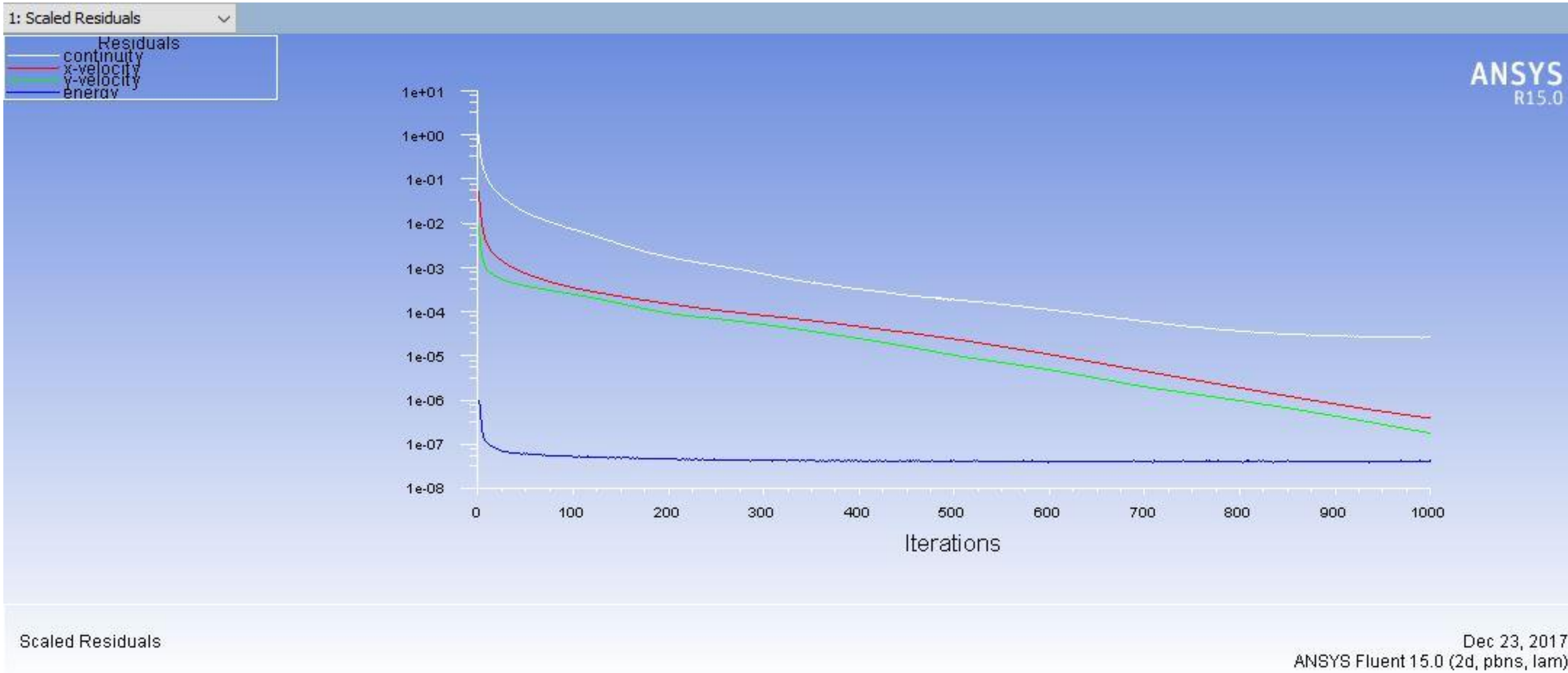
8st step: Initialization

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

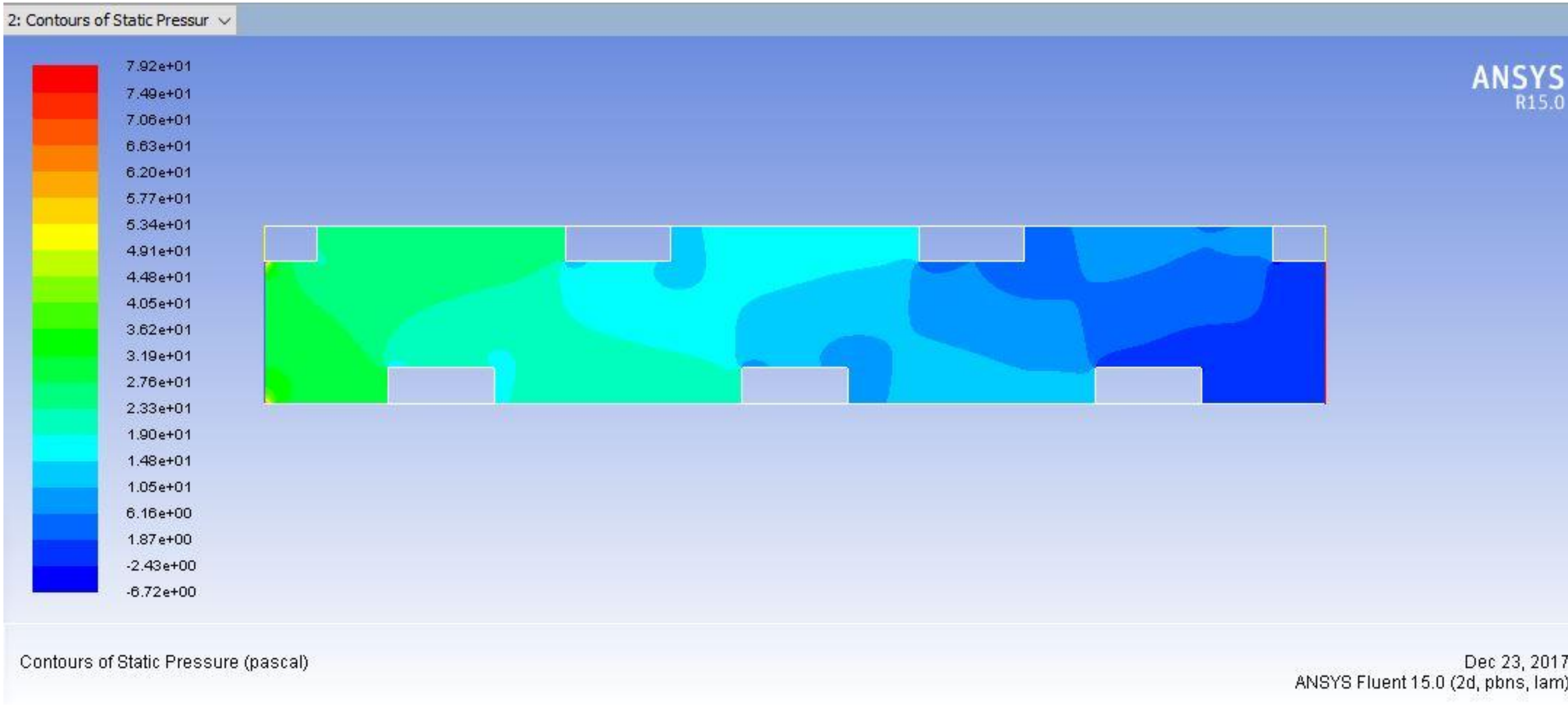
Step 9: Run the simulation

Step 10: Post-processing results

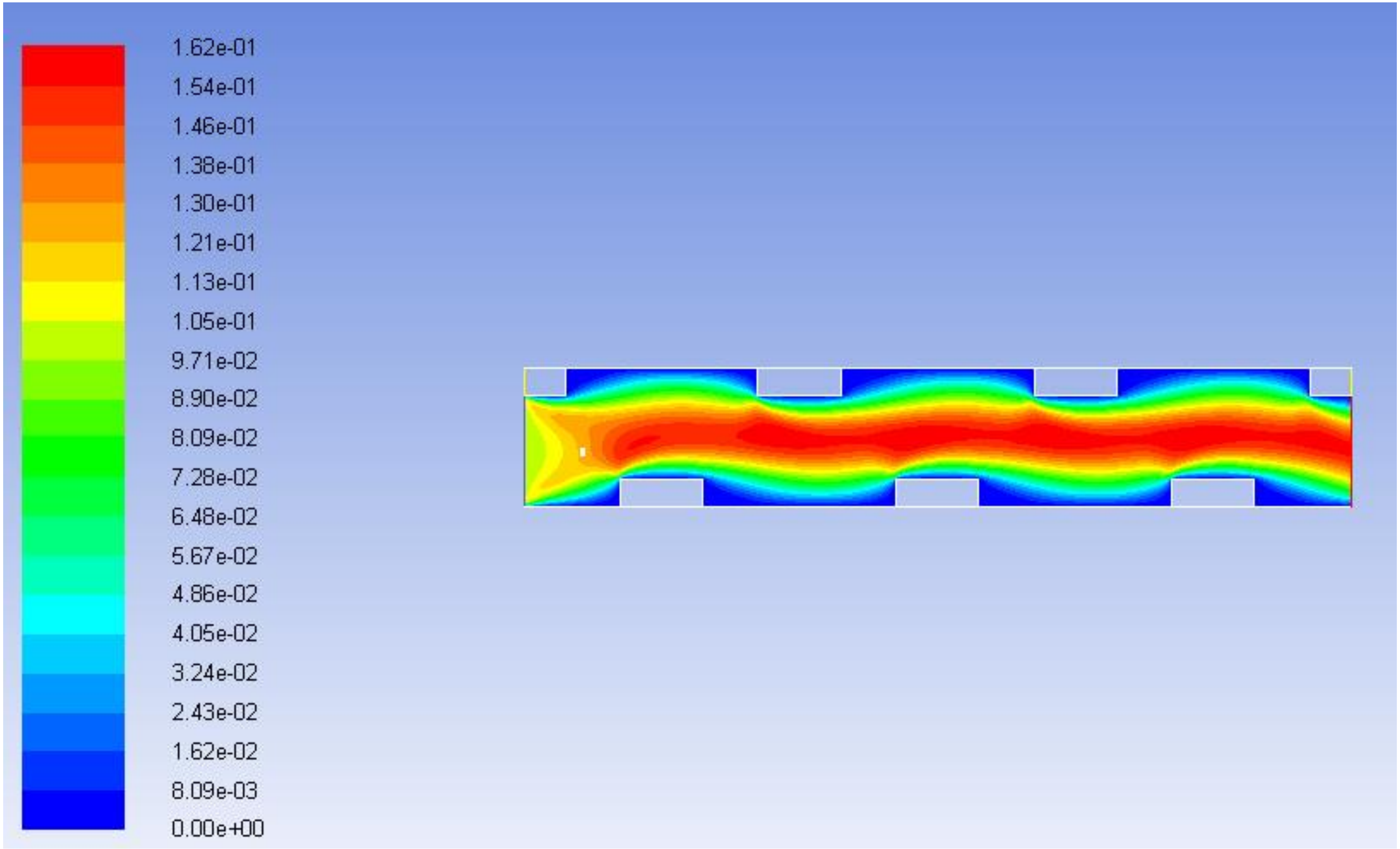
Residuals


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)

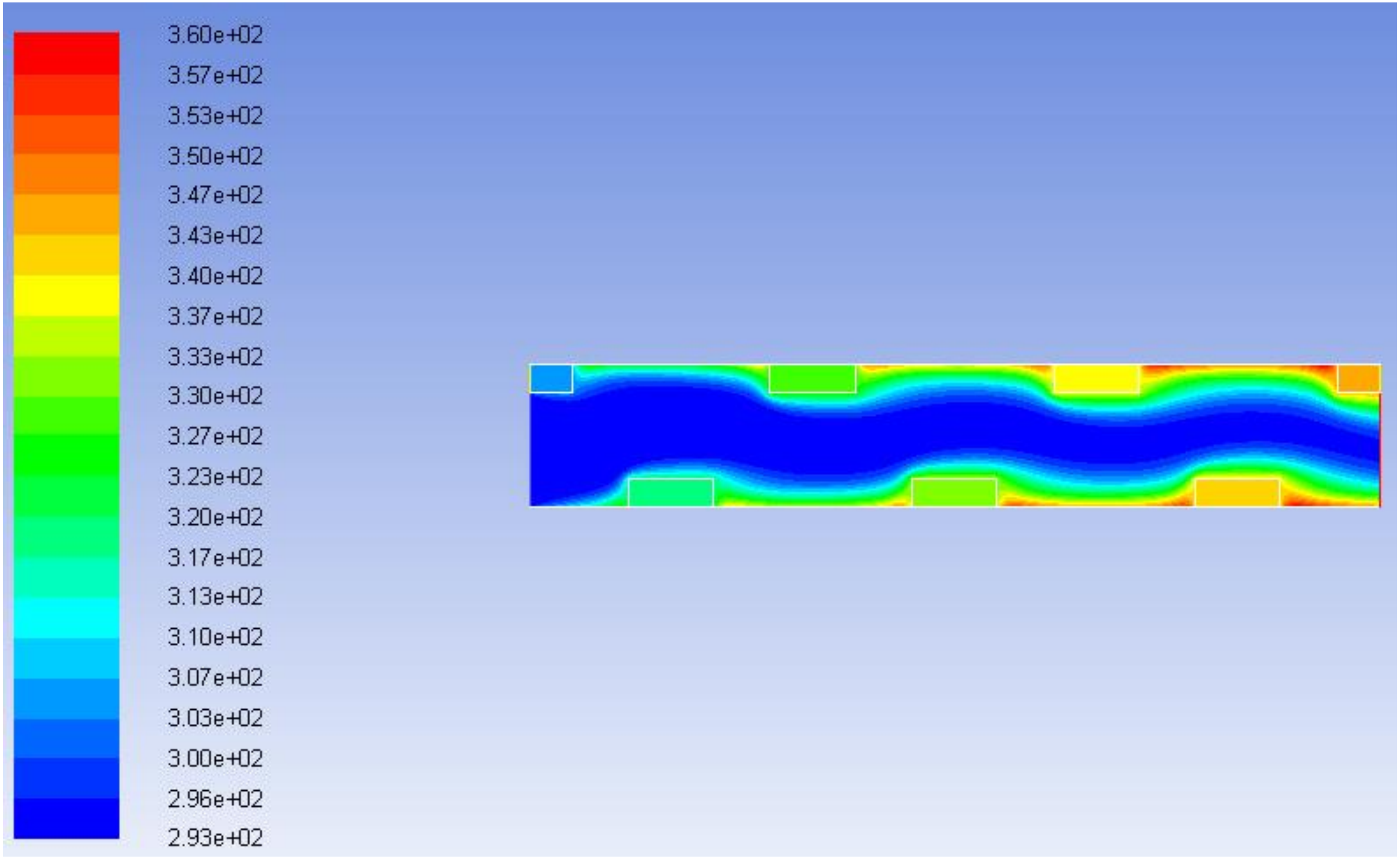
Contours of static pressure (Pa)



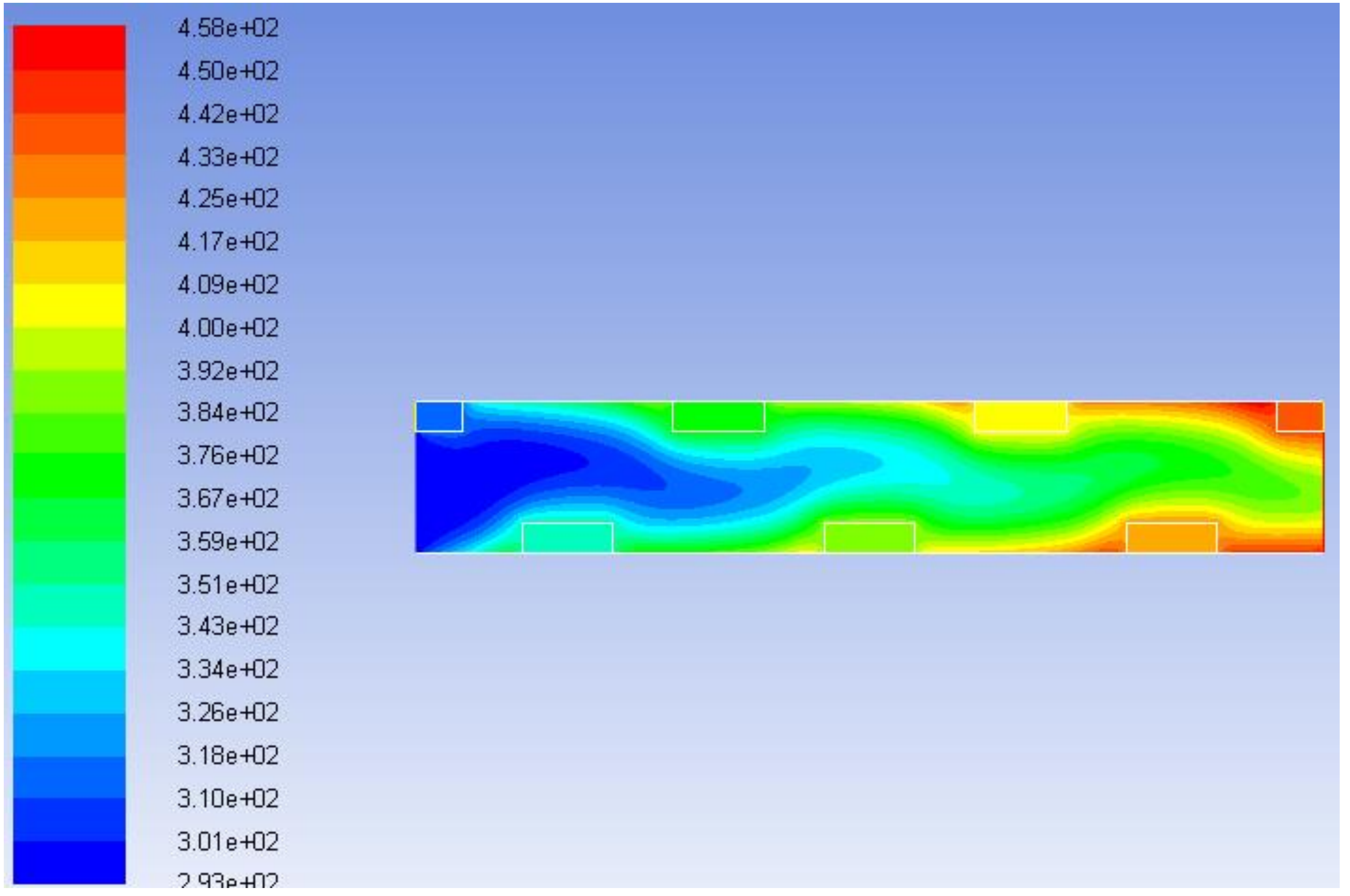
Velocity magnitude



Temperature (K)



Temperature (K) of velocity as 0.01



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

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....