

# 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**

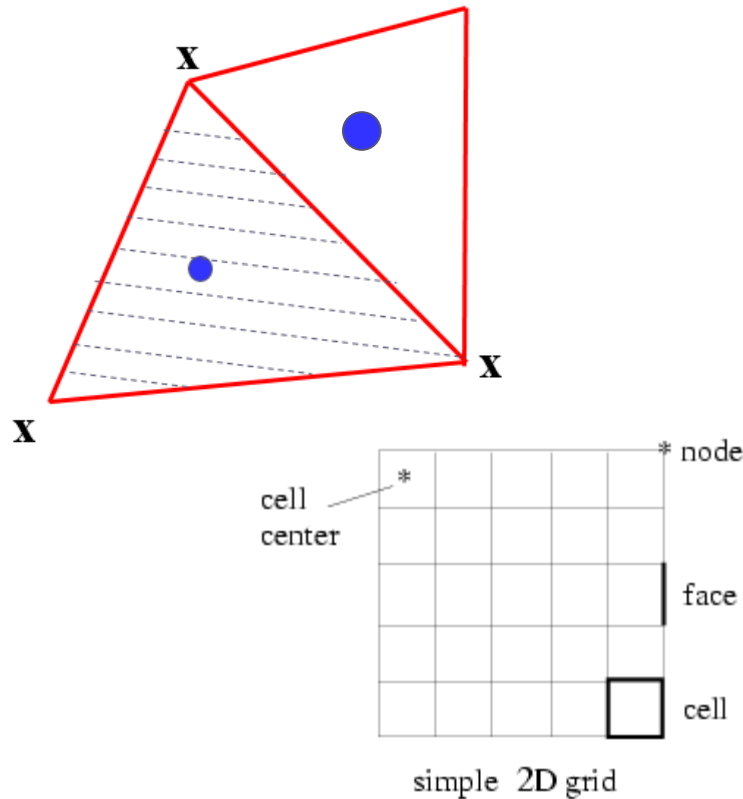
**13.6 多孔介质流动换热问题**



**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
	Node
Interface	Interior face

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

**Gradient calculation,**  
**There are three schemes.**

Gradient

Least Squares Cell Based

Green-Gauss Cell Based

Green-Gauss Node Based

Least Squares Cell Based

$\nabla \phi$

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:

$$\langle \nabla \phi \rangle = \frac{1}{V_C} \int_{V_C} \nabla \phi dV$$

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

How to determine  $\phi_f$  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.

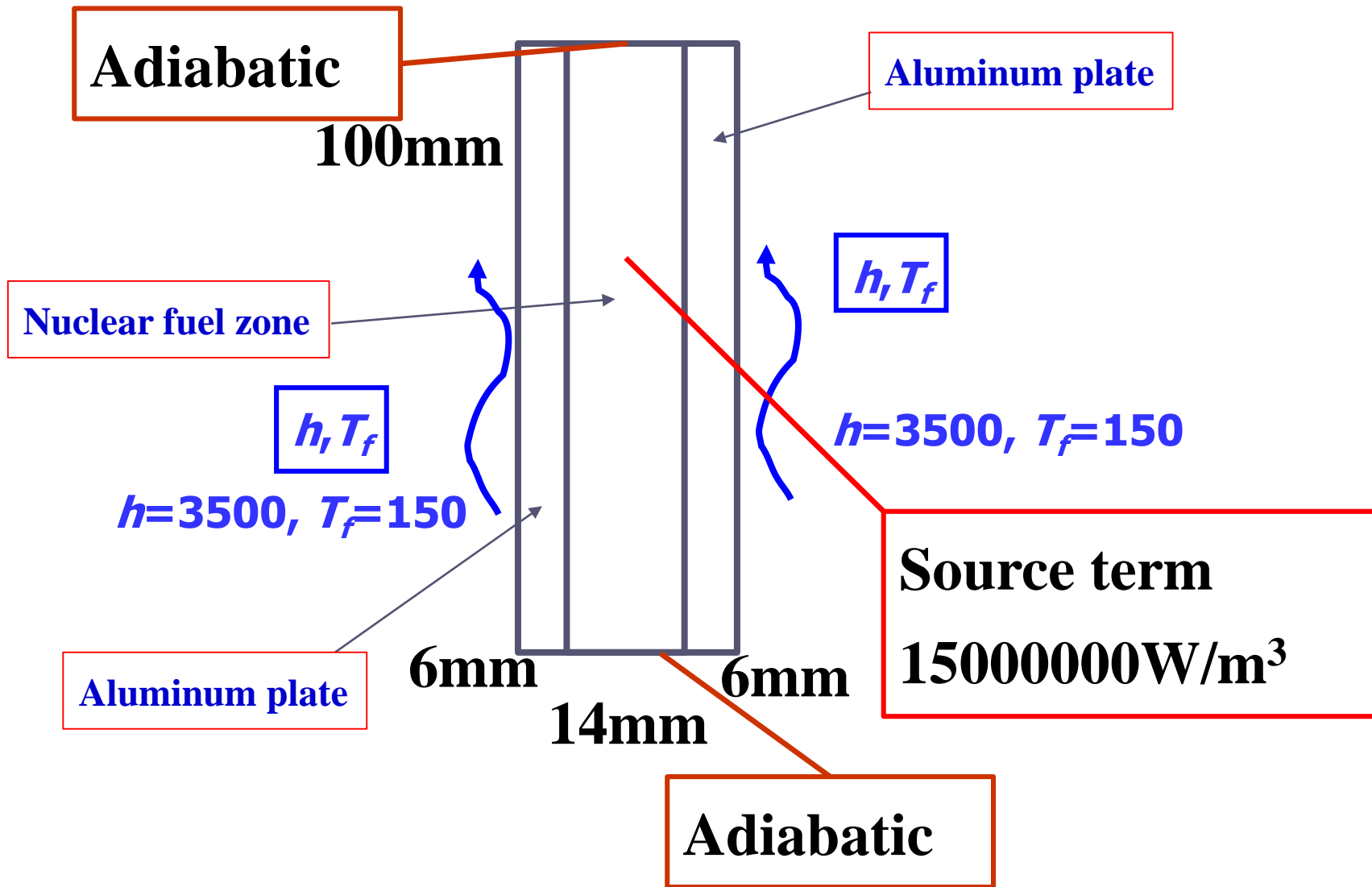
$$\xi = \sum_{i=1}^N \left\{ w_i \left( \phi_{Ci} - \phi_{C0} - \left[ \frac{\partial \phi}{\partial x} \Delta x_i + \frac{\partial \phi}{\partial y} \Delta y_i + \frac{\partial \phi}{\partial z} \Delta z_i \right] \right)^2 \right\}$$



**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.

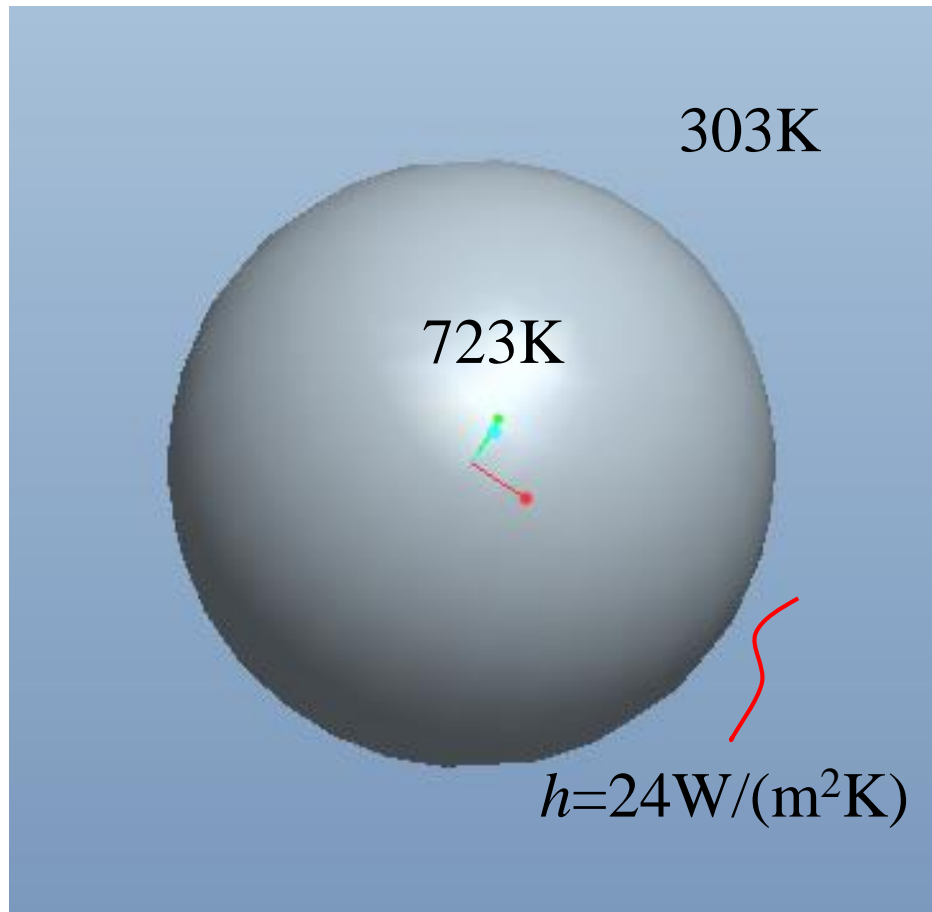
( $D=5\text{cm}$ , density is  $7735\text{kg/m}^3$ , heat capacity is  $480\text{ J}/(\text{kg K})$ , conductivity is  $33\text{W}/(\text{m K})$  ).

- Outside boundary condition : convective BC

fluid temperature: 303K

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

- 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:** 
$$\frac{\partial(\rho C_p T)}{\partial t} = \text{div}(\Gamma_T \text{grad} T)$$

It is a unsteady heat conduction problem with given GAMA.

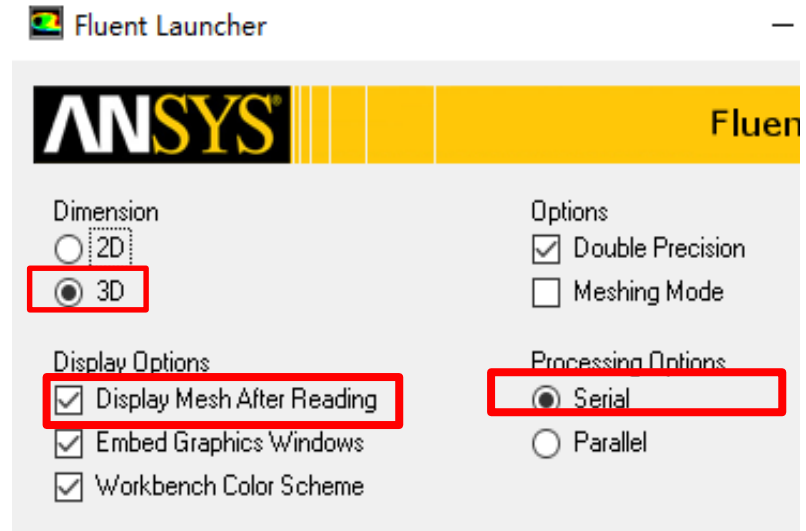
**Remark:** here we write the energy governing equation in the improved form with nominal density  $\rho C_p$ . The improved form is adopted in our general teaching code as well in Fluent.

# Start the Fluent software

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

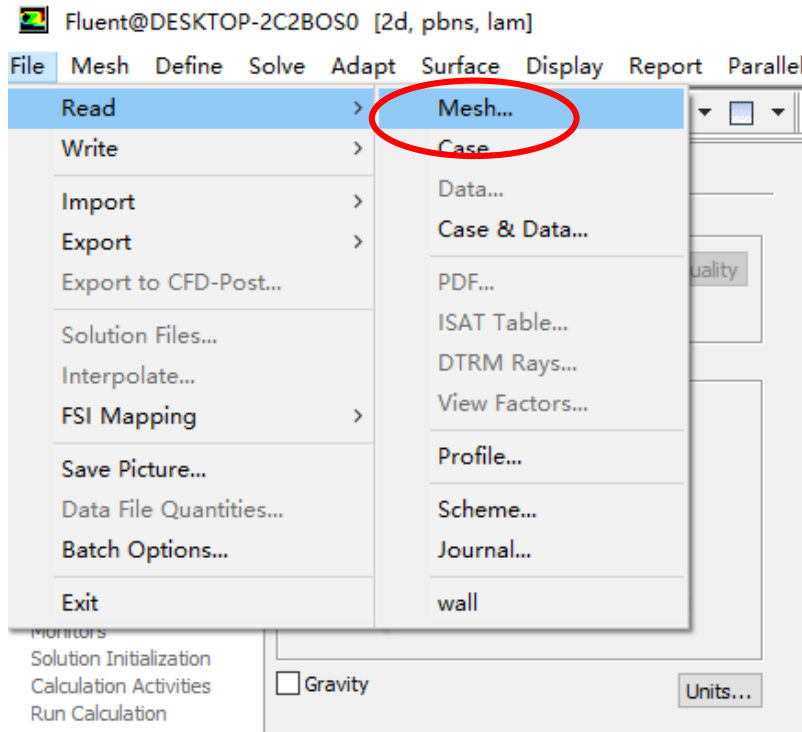
2、 If “display mesh after reading” is selected, after the Fluent is launched, the mesh will automatically shown in the interface.

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\al
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):  $6.519246e-05$

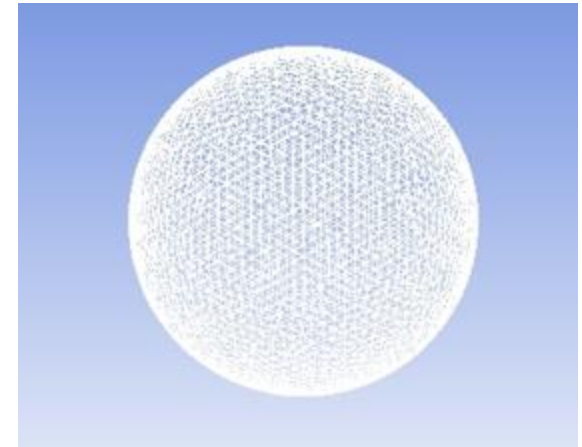
#### Face area statistics:

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):  $5.081937e-03$

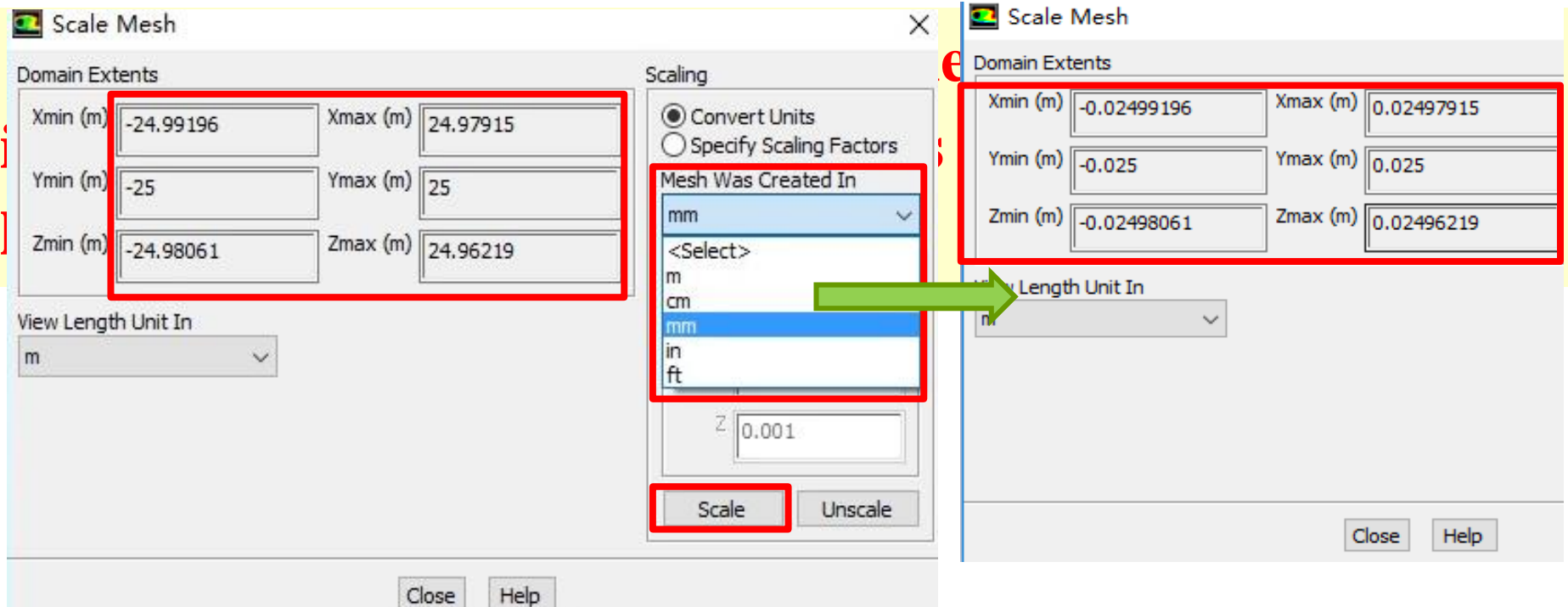
WARNING: Mesh check failed.

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.

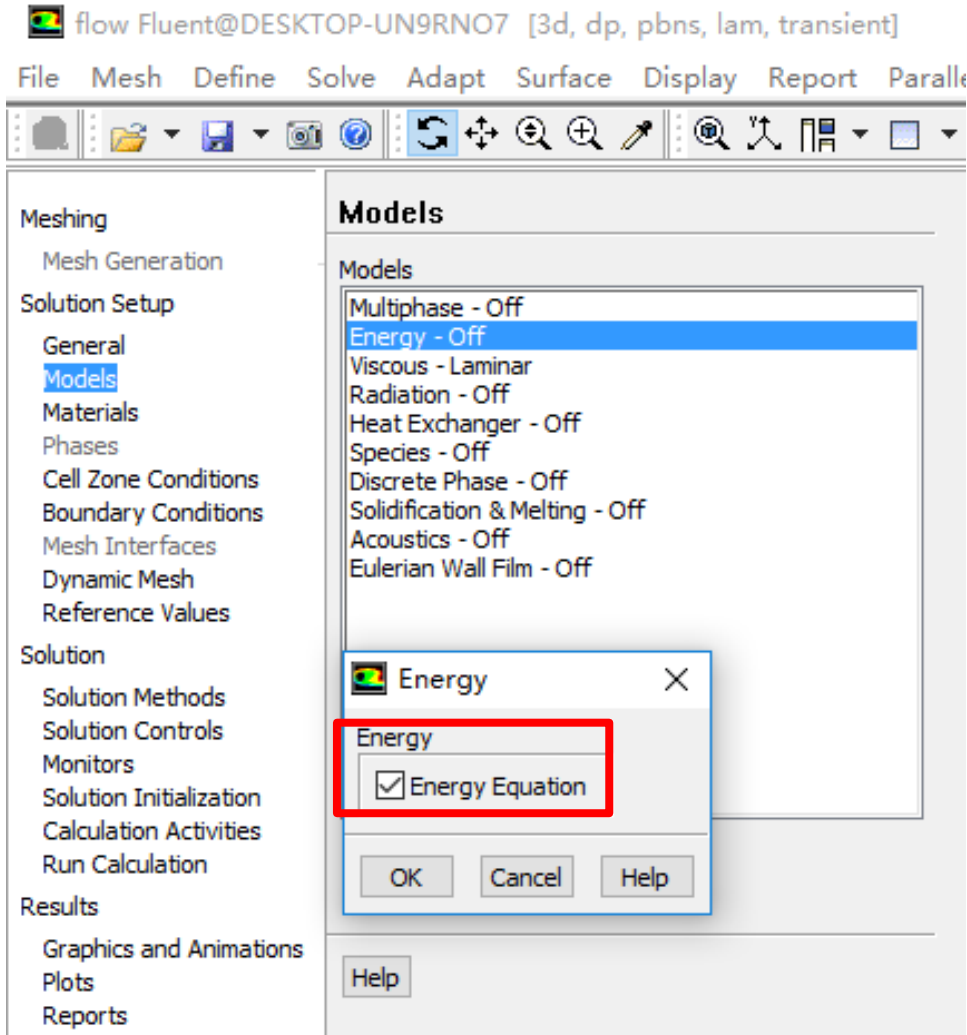




The image shows the 'General' settings panel in ANSYS Fluent. On the left is a navigation tree with categories: Meshing, Solution Setup, Models, Materials, Phases, Cell Zone Conditions, Boundary Conditions, Mesh Interfaces, Dynamic Mesh, Reference Values, Solution, Solution Methods, Solution Controls, Monitors, Solution Initialization, Calculation Activities, and Run Calculation. The 'General' panel is active, showing 'Mesh' options (Scale..., Check, Report Quality, Display...) and 'Solver' options. Under 'Type', 'Pressure-Based' is selected. Under 'Velocity Formulation', 'Absolute' is selected. The 'Time' section is highlighted with a red box, showing 'Steady' and 'Transient' radio buttons, with 'Transient' selected. At the bottom, there is a 'Gravity' checkbox (unchecked) and a 'Units...' button.

**Choose “transient” for a unsteady problem!**

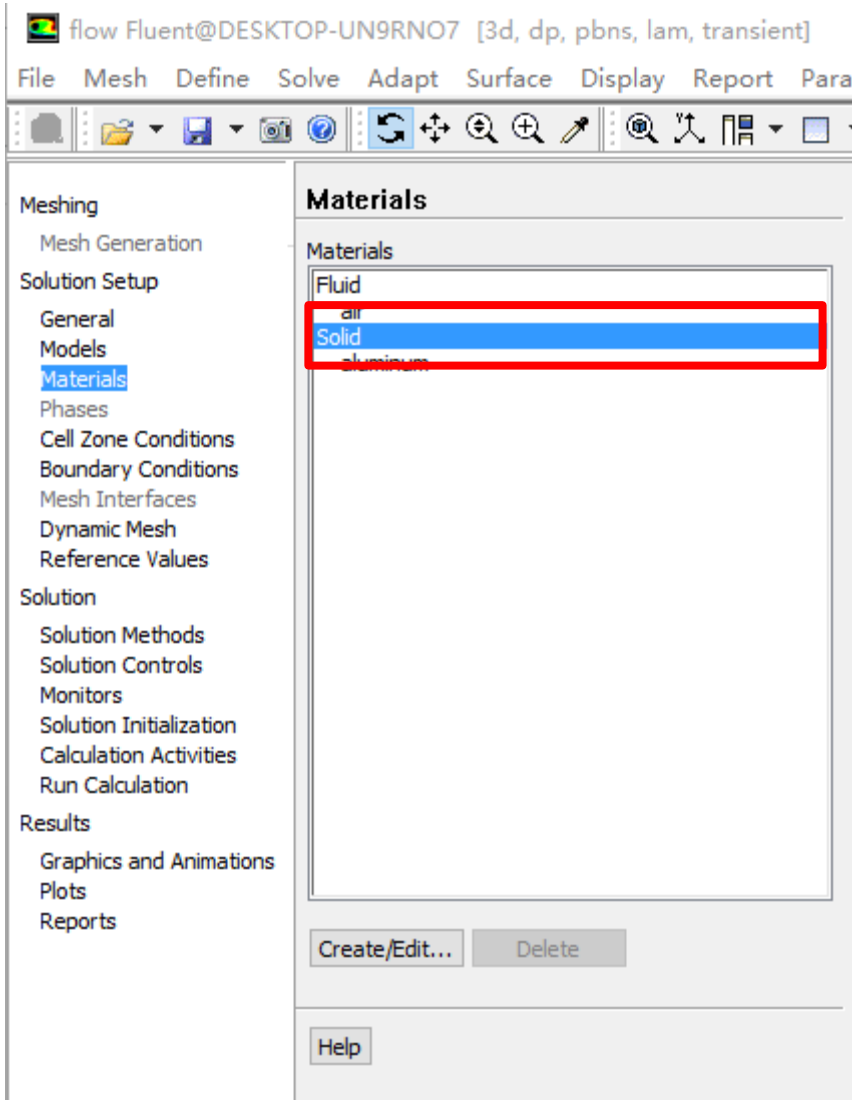
## Step 3: Choose the physicochemical model



$$\frac{\partial(\rho C_p T)}{\partial t} = \text{div}(\Gamma_T \text{grad} T)$$

**The energy equation is activated.**

## Step 4: Define the material properties



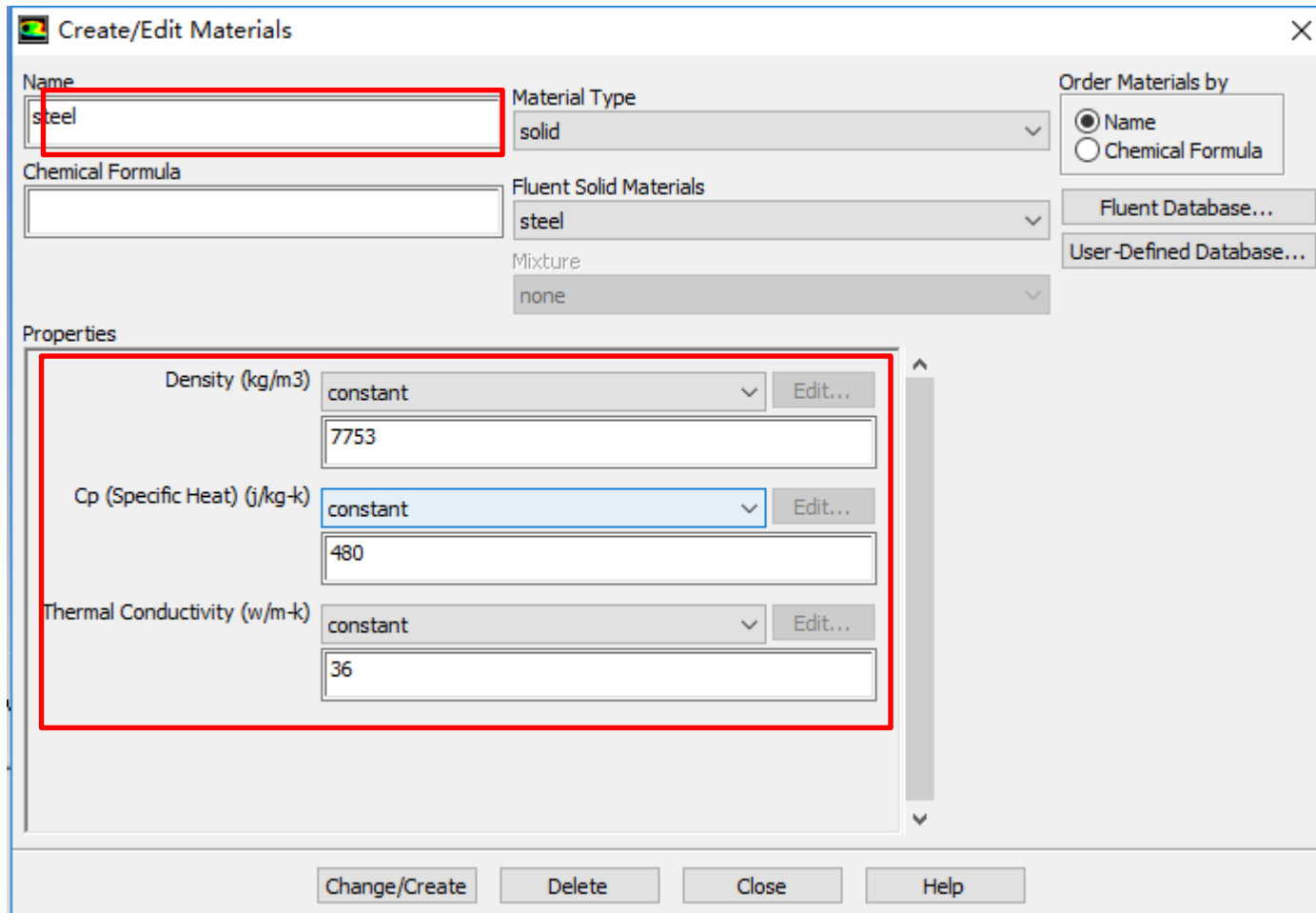
**The default fluid in  
Fluent is air.**

**The default solid in Fluent  
is Aluminum.**

**For Example 2, steel  
material should be added.**

# The properties of steel are manually inputted.

Density is  $7735\text{kg/m}^3$ , heat capacity is  $480\text{ J}/(\text{kg K})$ ,  
conductivity is  $33\text{W}/(\text{m K})$



Create/Edit Materials

Name: steel

Material Type: solid

Order Materials by:  
 Name  
 Chemical Formula

Chemical Formula:

Fluent Solid Materials: steel

Mixture: none

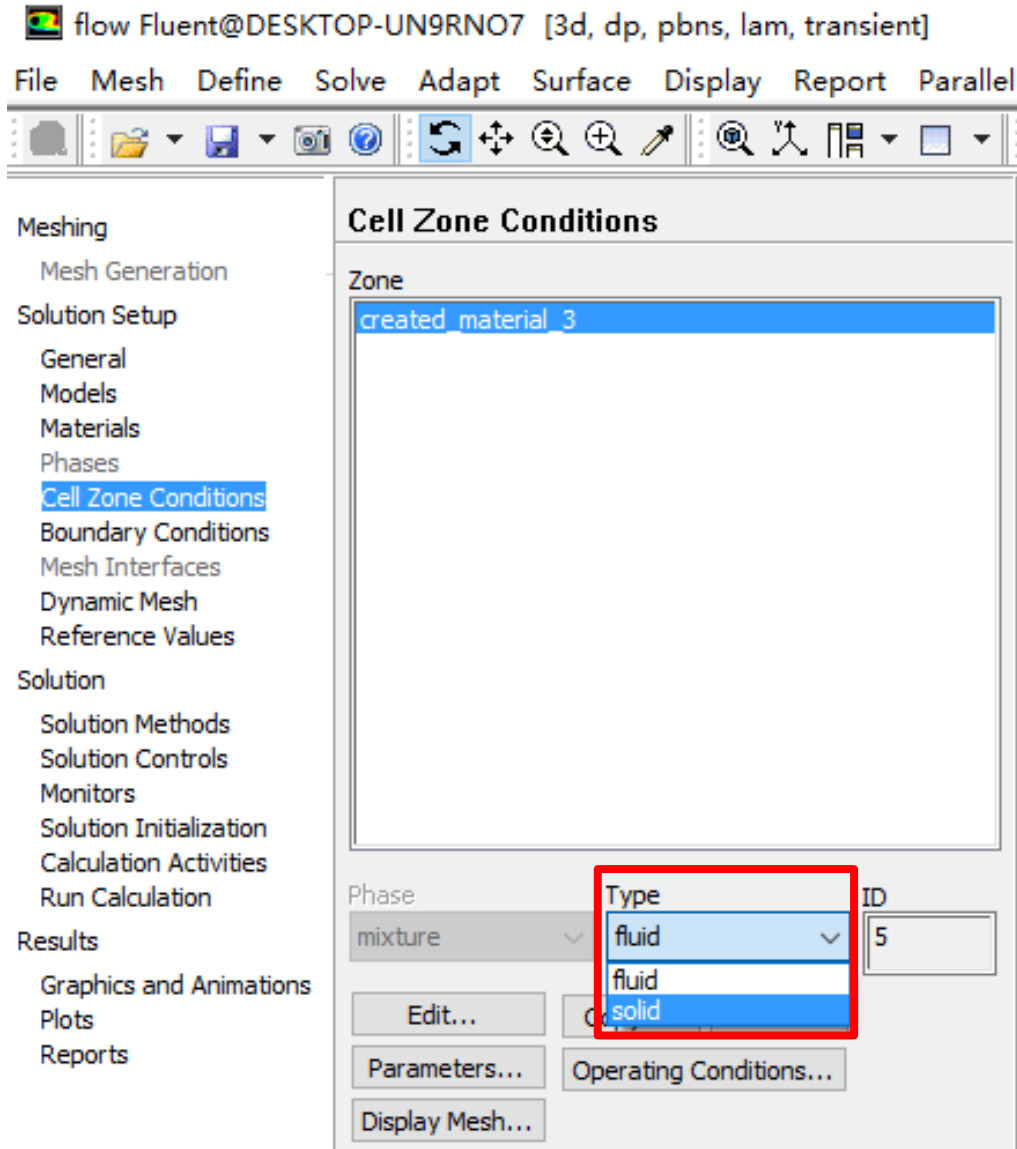
Fluent Database...  
User-Defined Database...

Properties

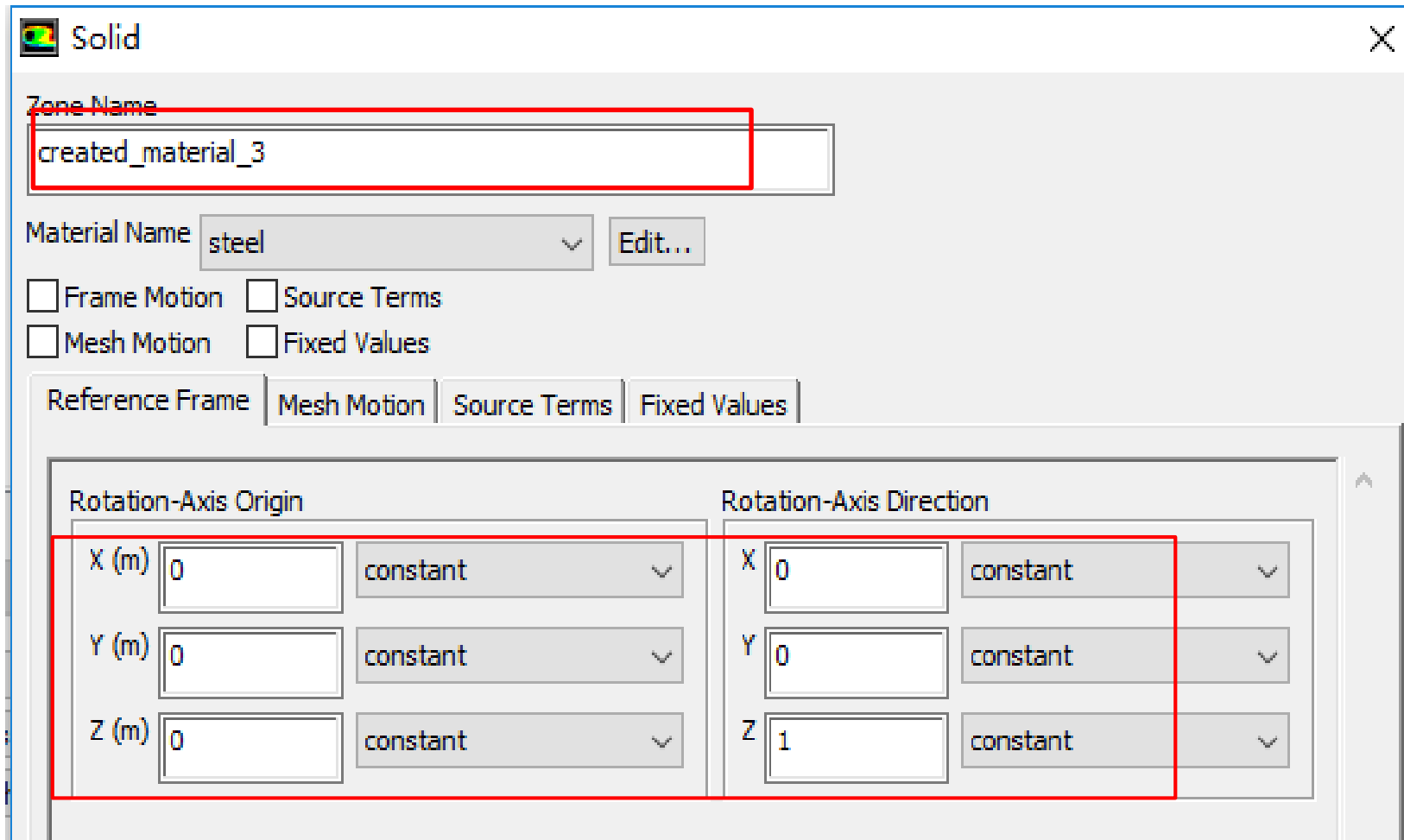
Density (kg/m <sup>3</sup> )	constant	Edit...
	7753	
Cp (Specific Heat) (j/kg-k)	constant	Edit...
	480	
Thermal Conductivity (w/m-k)	constant	Edit...
	36	

Change/Create Delete Close Help

# Step 5: Define zone condition



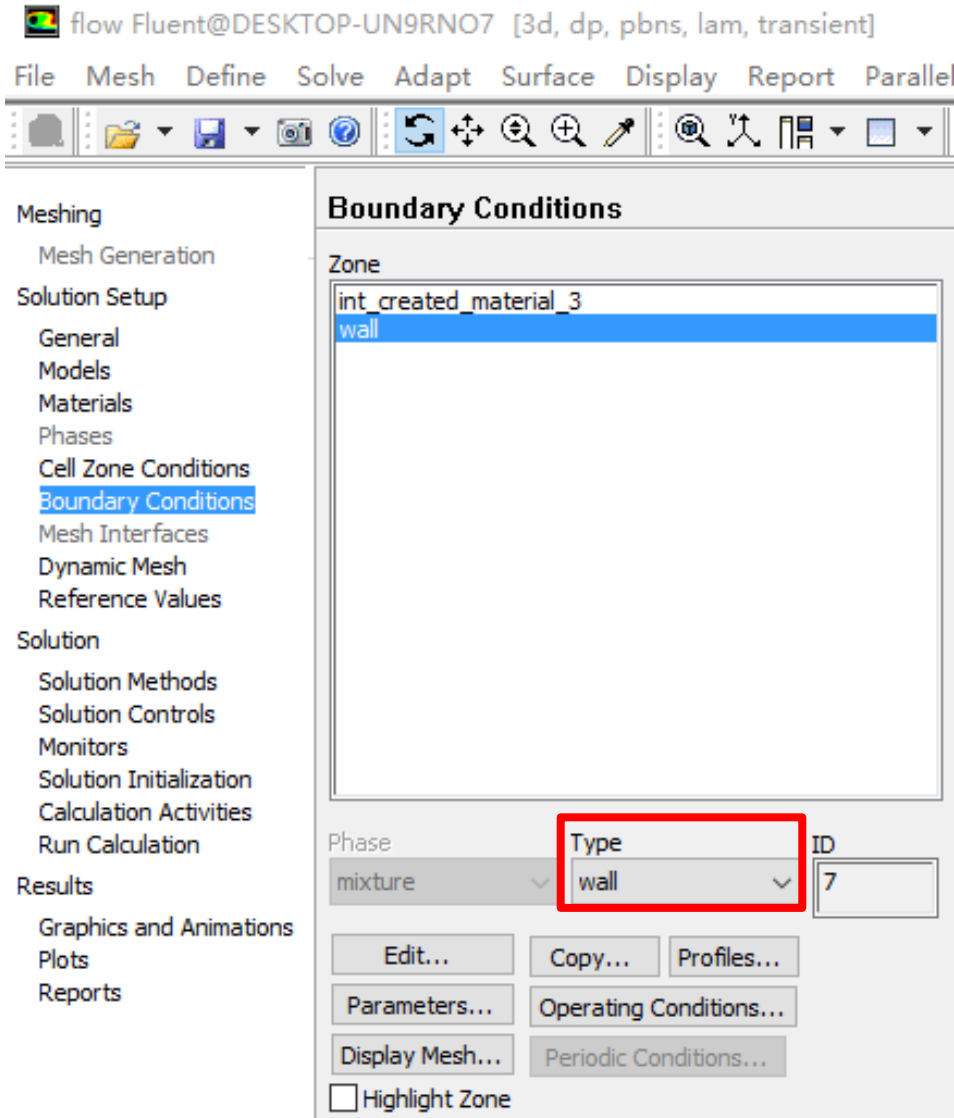
**In this step, we define the cell zone conditions. , the cell zone is a ball made of steel, so you should choose the type “solid”.**



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



## Step 6: Define the boundary condition



flow Fluent@DESKTOP-UN9RNO7 [3d, dp, pbns, lam, transient]

File Mesh Define Solve Adapt Surface Display Report Parallel

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

**Boundary Conditions**

Zone	Phase	Type	ID
int_created_material_3	mixture	wall	7

Buttons: Edit..., Copy..., Profiles..., Parameters..., Operating Conditions..., Display Mesh..., Periodic Conditions..., Highlight Zone

Now, you need to define the “Boundary conditions”  
 Firstly, Ensure the “type” is “wall”.

Then click the “edit” to edit the BC.

Wall
✕

Zone Name

Adjacent Cell Zone

Momentum
Thermal
Radiation
Species
DPM
Multiphase
UDS
Wall Film

Thermal Conditions

Heat Flux
  Temperature
  Convection
  Radiation
  Mixed
  via System Coupling

Material Name  

Edit...

Heat Transfer Coefficient (w/m<sup>2</sup>-k)  constant

Free Stream Temperature (k)  constant

Wall Thickness (m)  P

Heat Generation Rate (w/m<sup>3</sup>)  constant

Shell Conduction Define...

**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 screenshot displays the ANSYS Fluent software interface. The top menu bar includes File, Mesh, Define, Solve, Adapt, Surface, Display, Report, and Parallel. The main workspace shows the 'Solution Methods' panel on the left and the 'Solution Controls' panel on the right.

**Solution Methods Panel:**

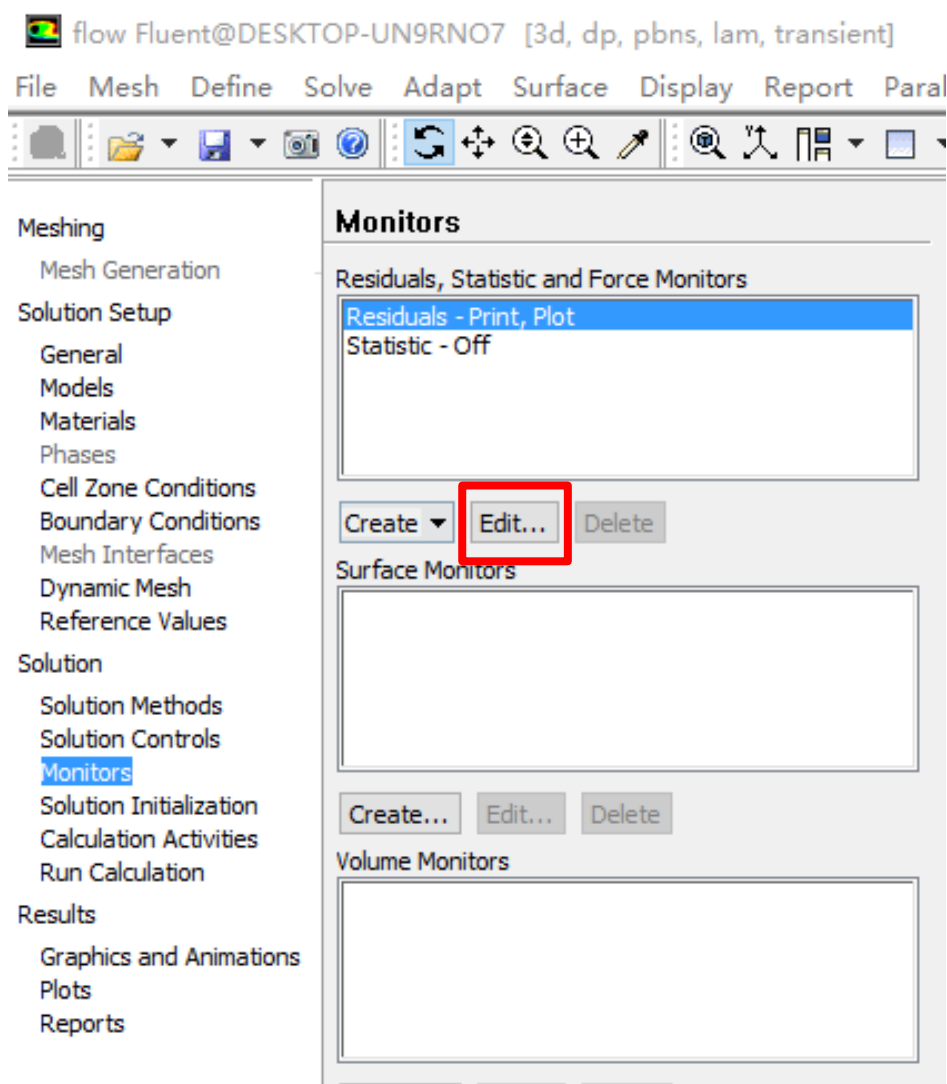
- Pressure-Velocity Coupling: Scheme: SIMPLE
- Spatial Discretization:
  - Gradient: Least Squares Cell Based
  - Pressure: Second Order
  - Momentum: Second Order Upwind
  - Energy: Second Order Upwind
- Transient Formulation: First Order Implicit
  - Non-Iterative Time Advancement
  - Frozen Flux Formulation
  - High Order Term Relaxation

**Solution Controls Panel:**

- Under-Relaxation Factors:
  - Pressure: 0.3
  - Density: 1
  - Body Forces: 1
  - Momentum: 0.7
  - Energy: 1
- Buttons: Default, Equations..., Limits..., Advanced..., Help

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

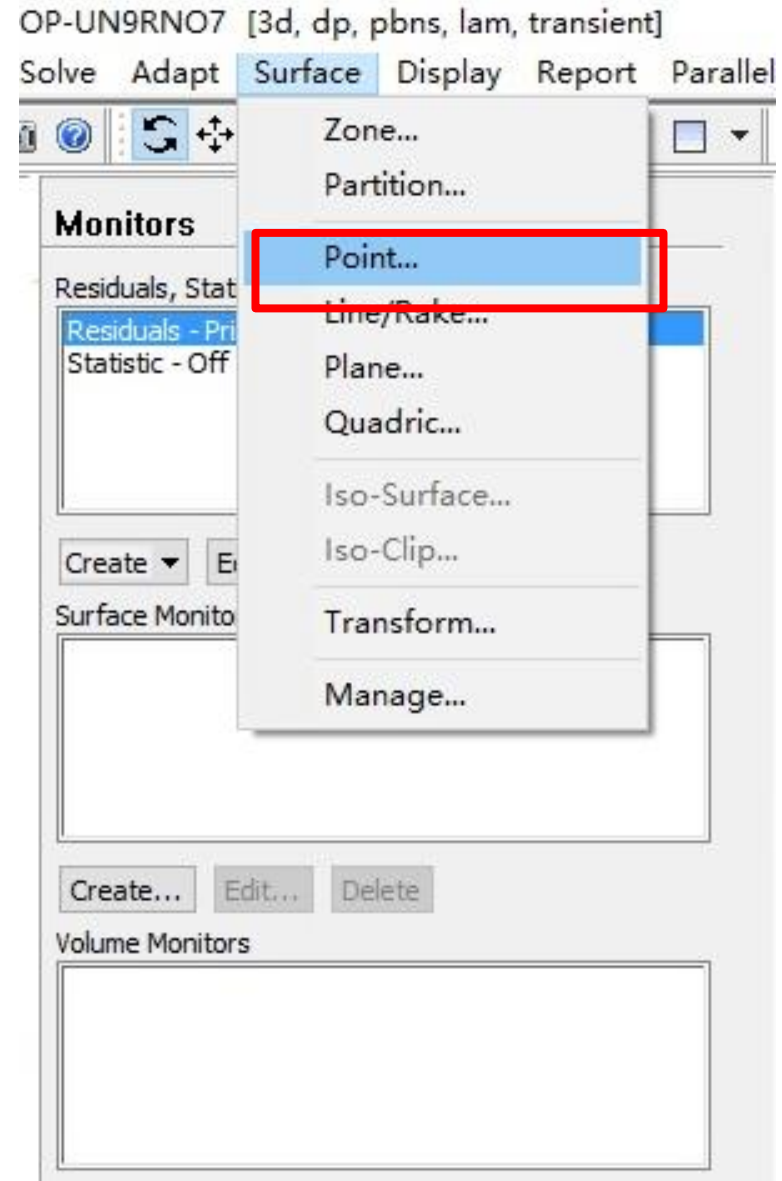
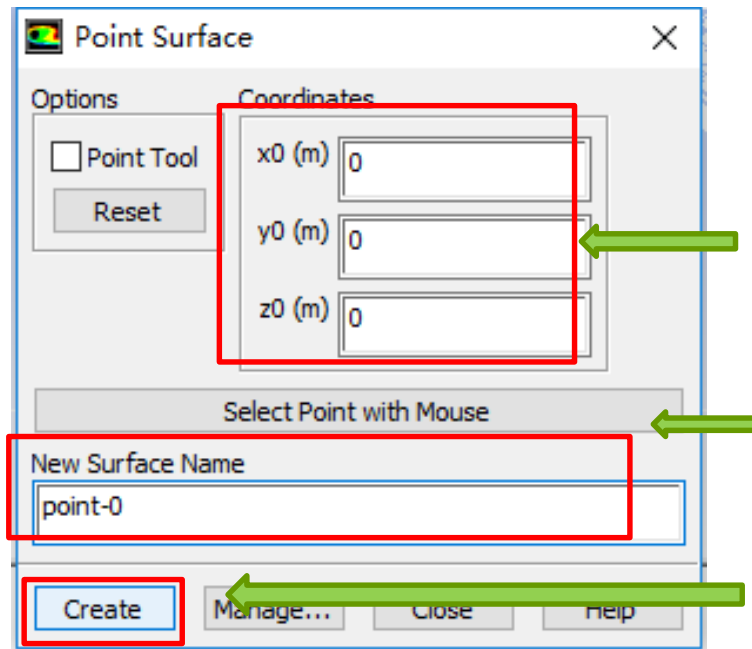
# Step 7: Solution setup: monitors



**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.

Plane Surface

Options

- Aligned with Surface
- Aligned with View Plane
- Point and Normal
- Bounded
- Sample Points
- Plane Tool

Sample Density

Edge 1: 1

Edge 2: 1

Select Points

Reset Points

Surfaces

int\_created\_material\_3  
point-0  
wall  
z-0

Points

x0 (m)	x1 (m)	x2 (m)
0	0.001	0.005
y0 (m)	y1 (m)	y2 (m)
0	0.001	0.005
z0 (m)	z1 (m)	z2 (m)
0	0	0

Normal

ix (m): 1

iy (m): 0

iz (m): 0

New Surface Name

z-0

Create    Manage...    Close    Help

OP-UN9RNO7 [3d, dp, pbns, lam, transient]

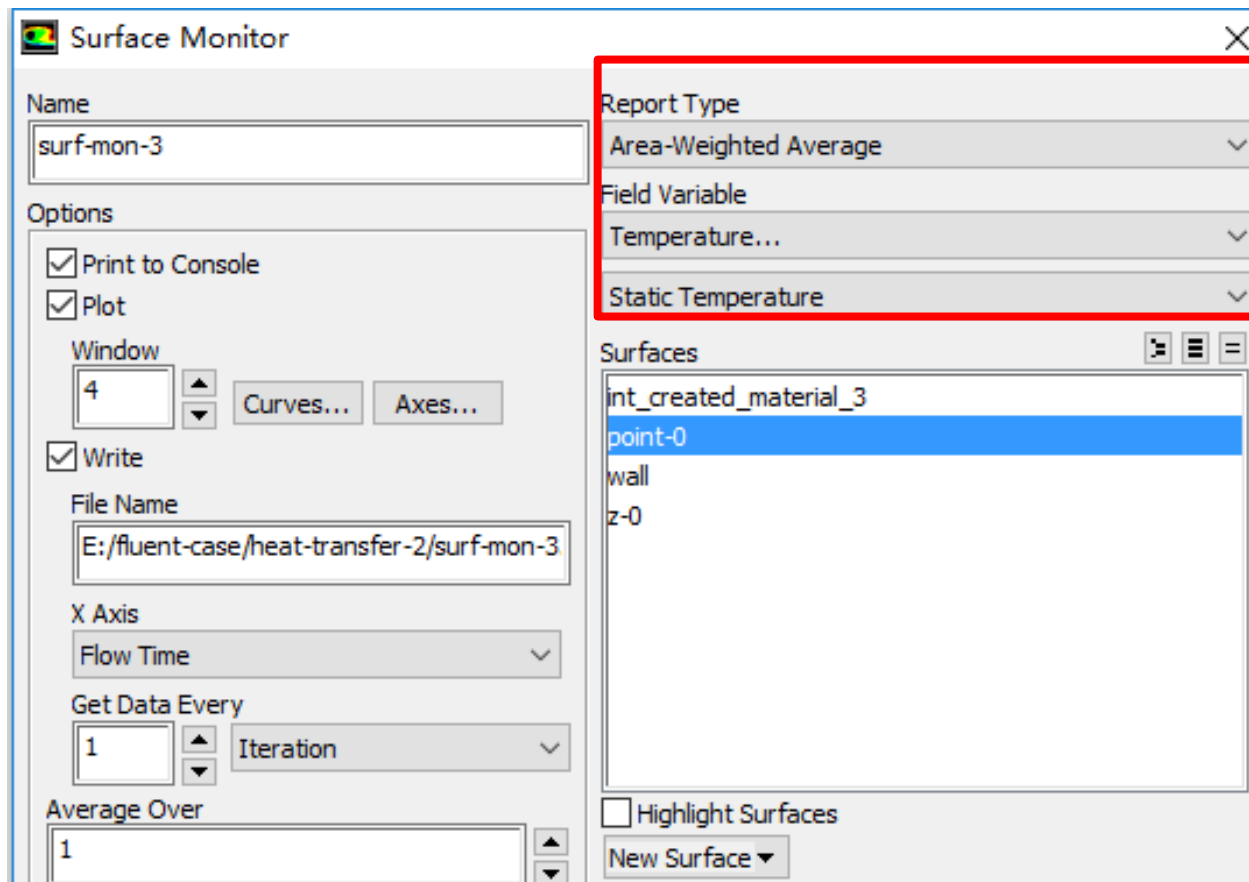
ve    Adapt    **Surface**    Display    Report    Paralle

Cell Zone C

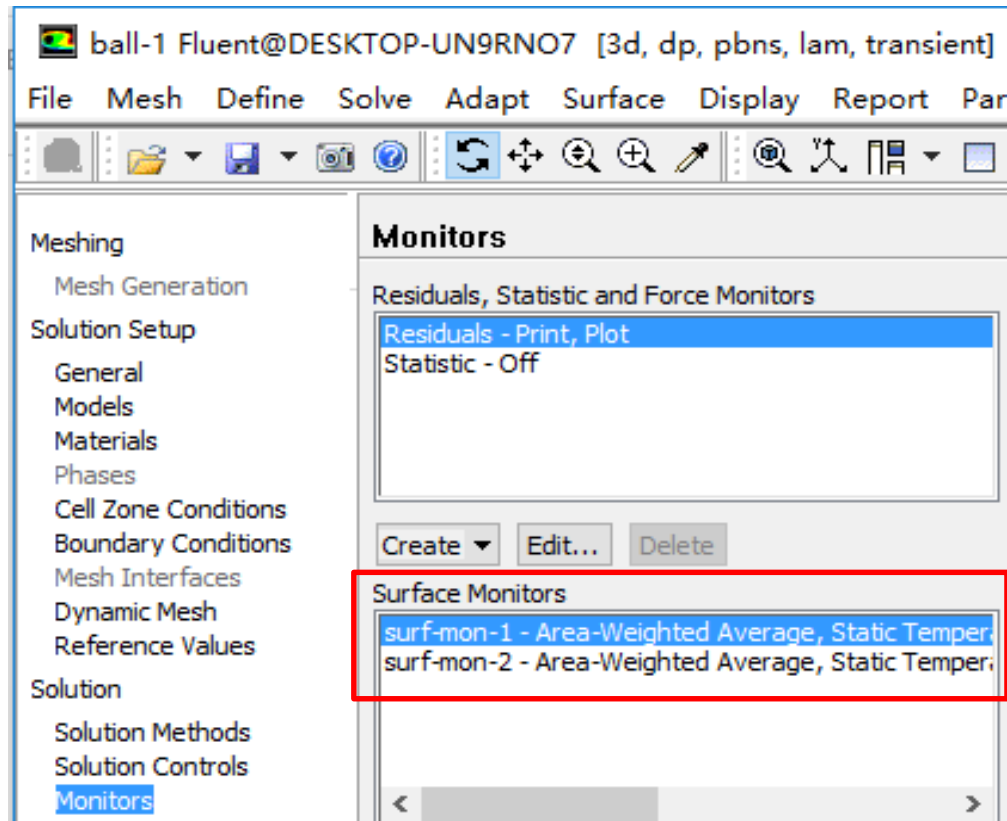
Zone

created mate

- Zone...
- Partition...
- Point...
- Line/Rake
- Plane...**
- Quadric...
- Iso-Surface...
- Iso-Clip...
- Transform...
- Manage...



**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.**

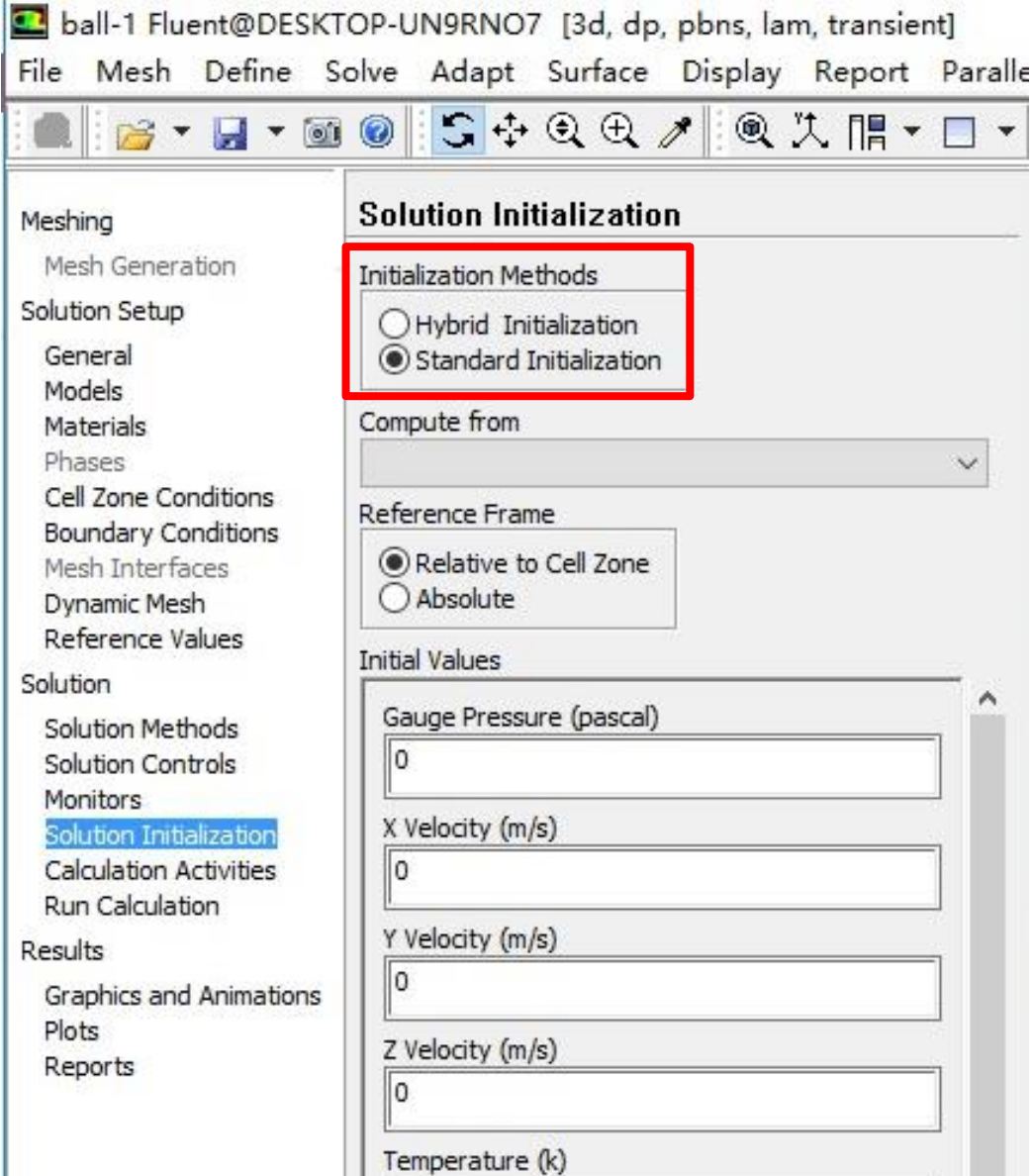


**Similarly, you can create a monitor to monitor the average temperature on the surface “z-0”.**

**In the “Surface Monitors”, you can see two monitors created.**



# Step 8: Initialization



ball-1 Fluent@DESKTOP-UN9RNO7 [3d, dp, pbns, lam, transient]

File Mesh Define Solve Adapt Surface Display Report Paralle

**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)

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

Here, the Standard initialization is adopted.

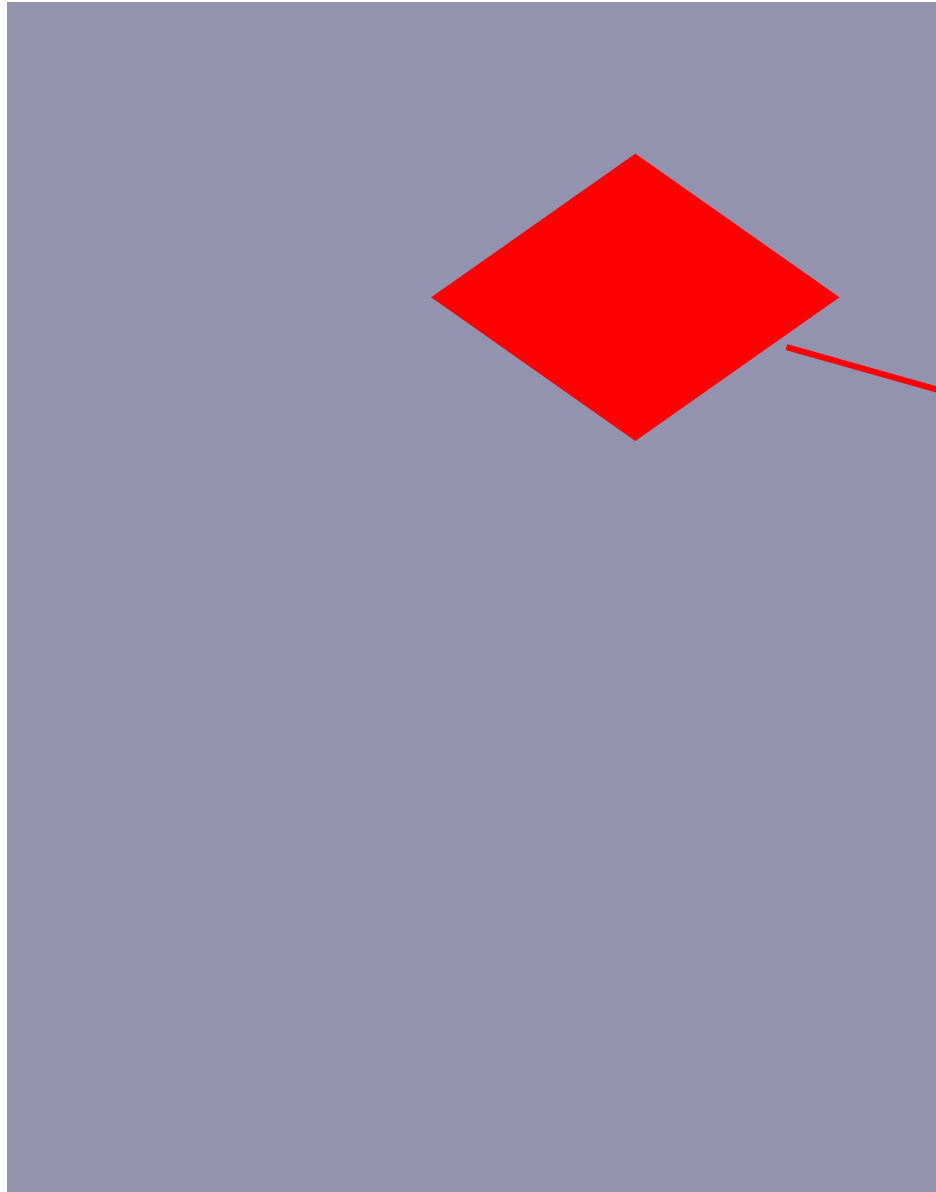
## **Patching (修补) Values in Selected Cells**

**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.**



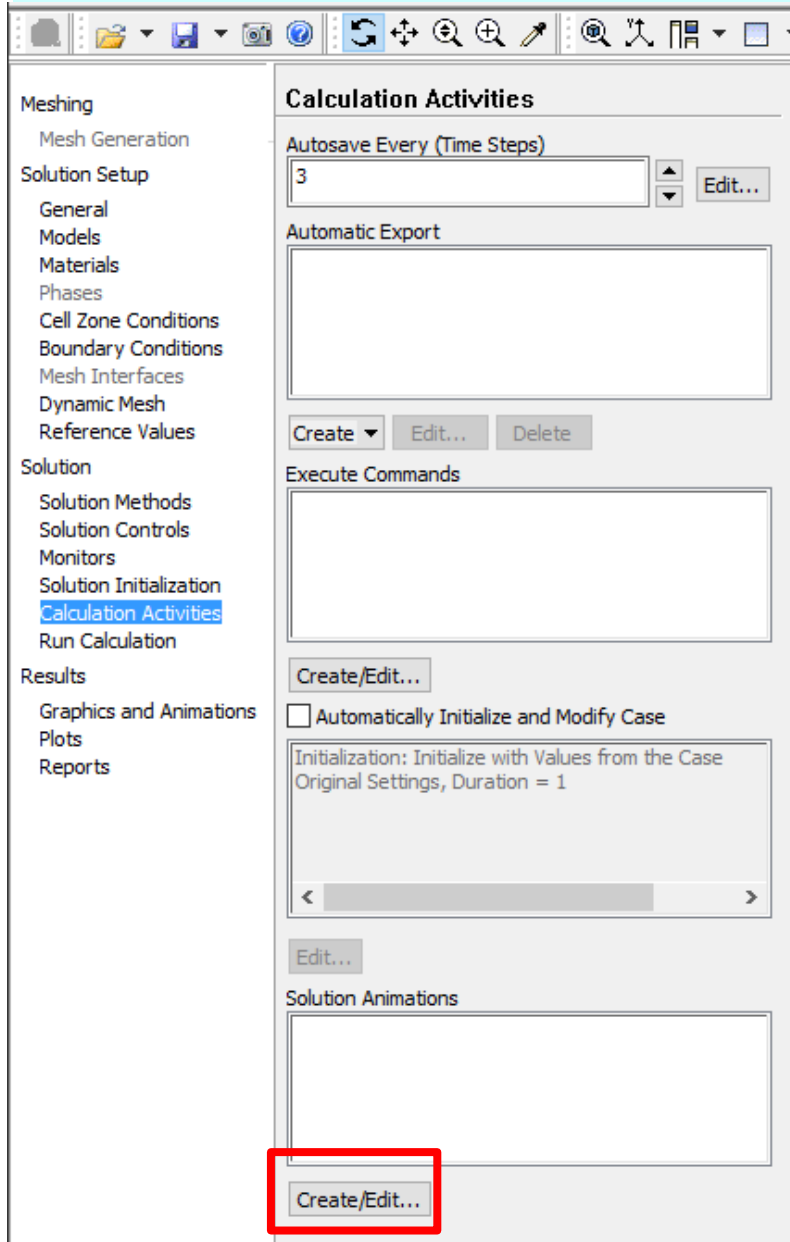
**Domain**

**Sub-region need to Patch**

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



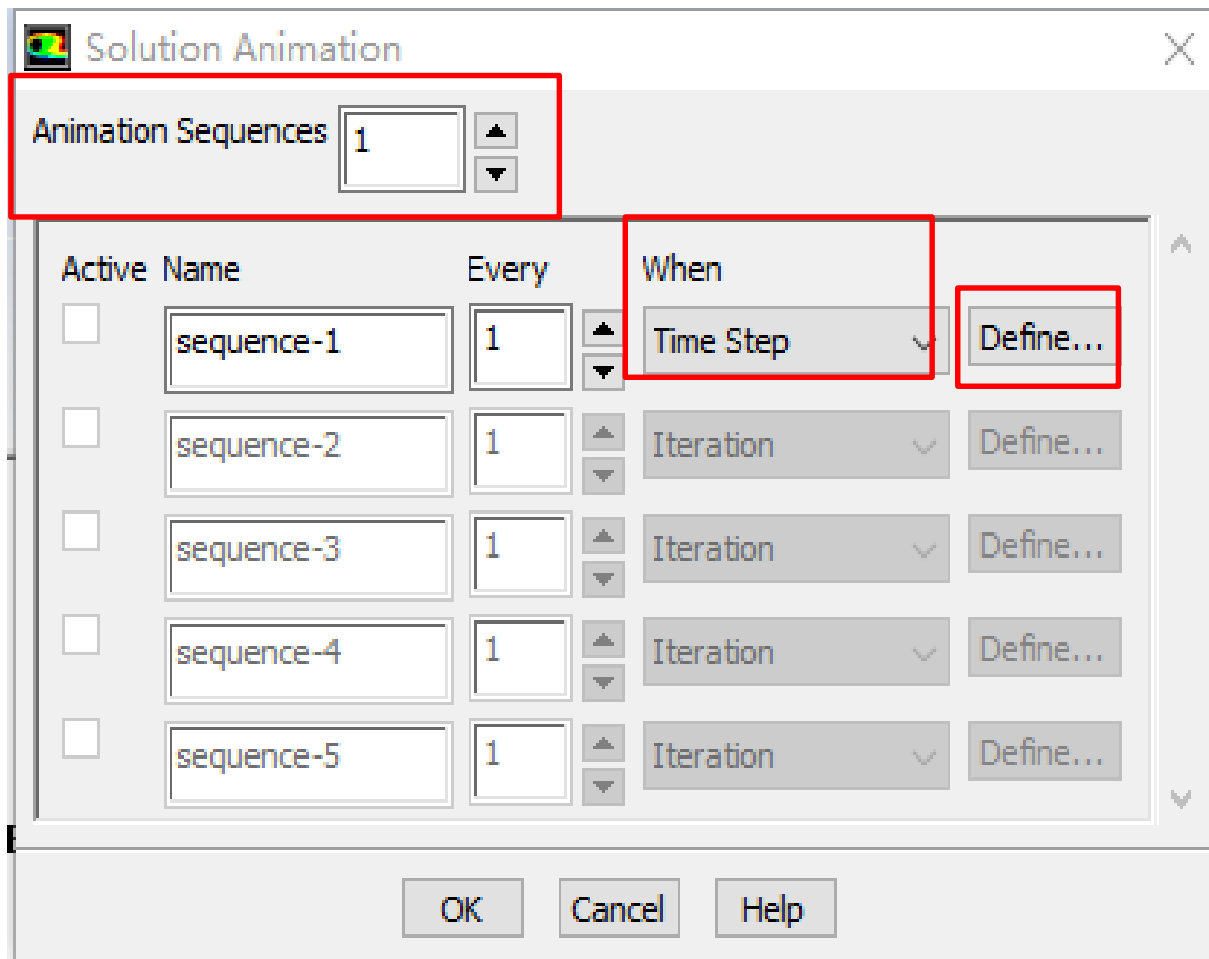
# 9st step: set animations



The screenshot shows the ANSYS Fluent software interface. The left sidebar contains a tree view with categories: Meshing, Solution Setup, Solution, and Results. Under Solution Setup, 'Calculation Activities' is selected. The main window displays the 'Calculation Activities' dialog box. It has several sections: 'Autosave Every (Time Steps)' with a value of 3 and an 'Edit...' button; 'Automatic Export' with an empty text box and 'Create', 'Edit...', and 'Delete' buttons; 'Execute Commands' with an empty text box and a 'Create/Edit...' button; a checkbox for 'Automatically Initialize and Modify Case' which is unchecked; a text box containing 'Initialization: Initialize with Values from the Case Original Settings, Duration = 1' with a scroll bar; an 'Edit...' button; and 'Solution Animations' with an empty text box. A red box highlights the 'Create/Edit...' button at the bottom of the dialog.

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

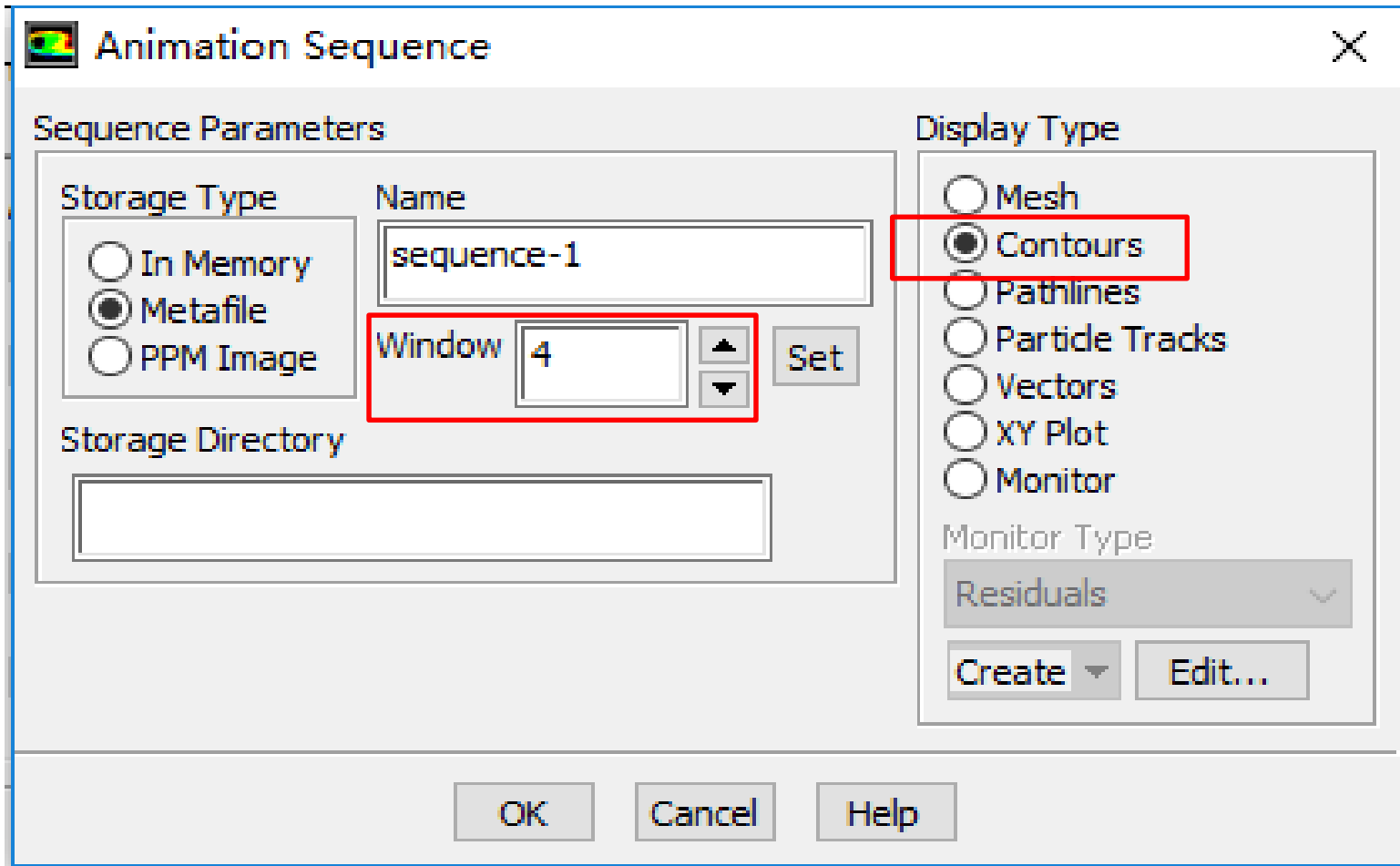
**In the “Calculation Activities” dialog, click “Change/Create” in “Solution Animations”.**



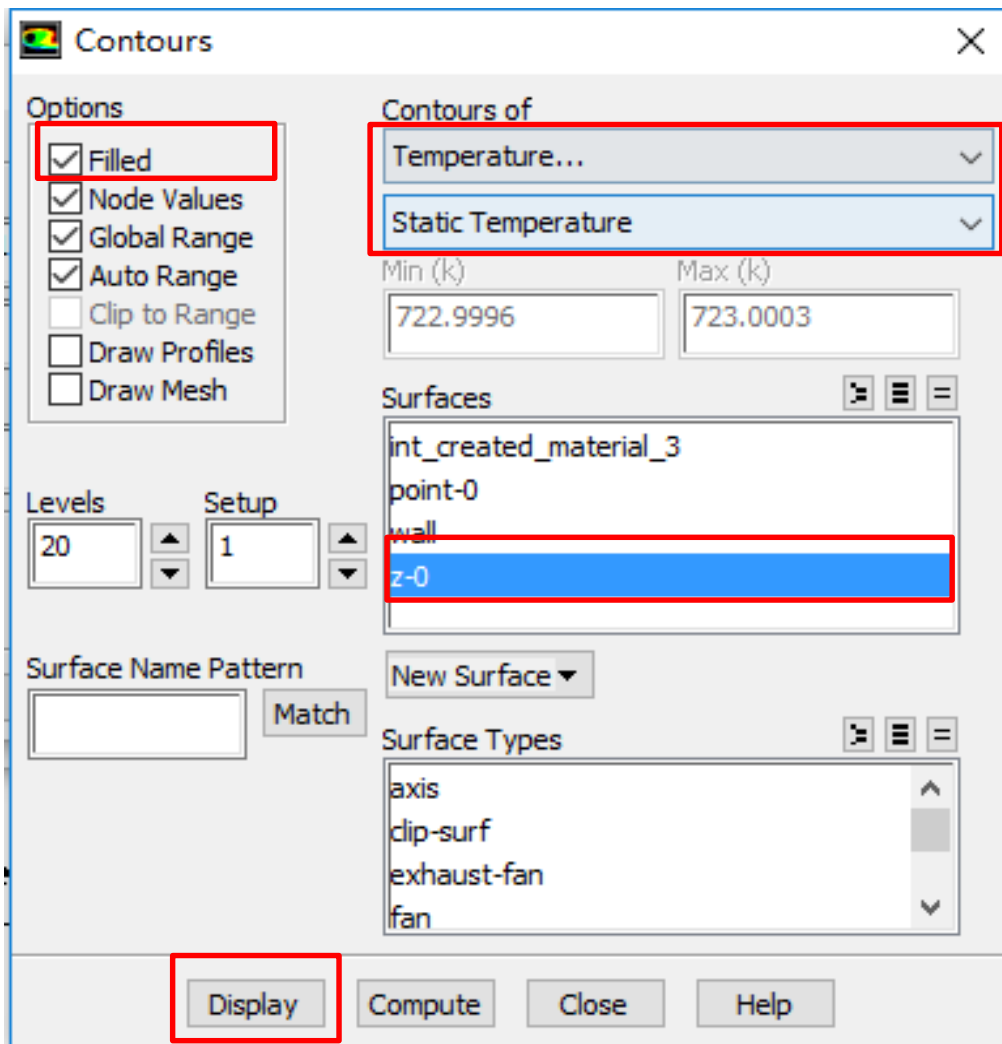
**Set the “Animation Sequences” as 1.**

**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.**



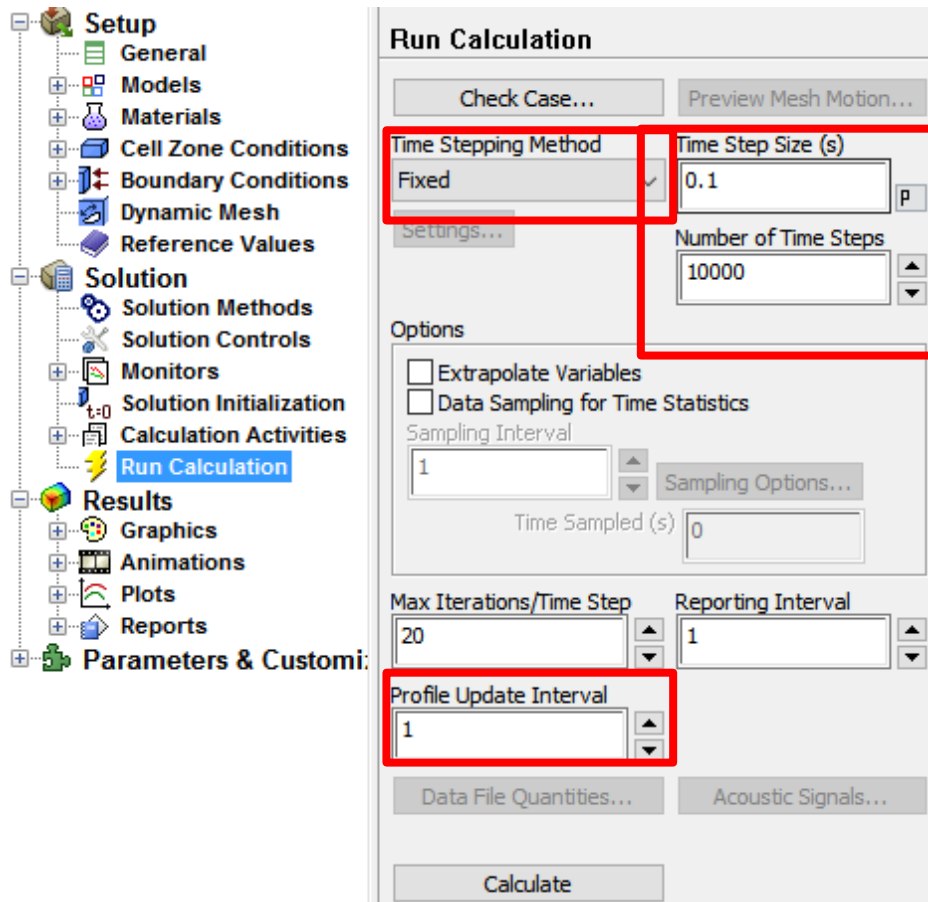
**In “Contours” dialog, we choose “Temperature”, select “Filled”, and choose the surface: z-0.**

**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.



The screenshot shows the 'Run Calculation' dialog box in ANSYS Fluent. The 'Time Stepping Method' is set to 'Fixed'. The 'Time Step Size (s)' is set to 0.1. The 'Number of Time Steps' is set to 10000. The 'Profile Update Interval' is set to 1. The 'Max Iterations/Time Step' is set to 20, and the 'Reporting Interval' is set to 1. The 'Calculate' button is visible at the bottom.

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

## Time stepping method

### Run Calculation

Check Case...

### Time Stepping Method

Fixed



Settings...

## Time step size

### Time Step Size (s)

0.1



### Number of Time Steps

10000



## Iteration per time step

### Max Iterations/Time Step

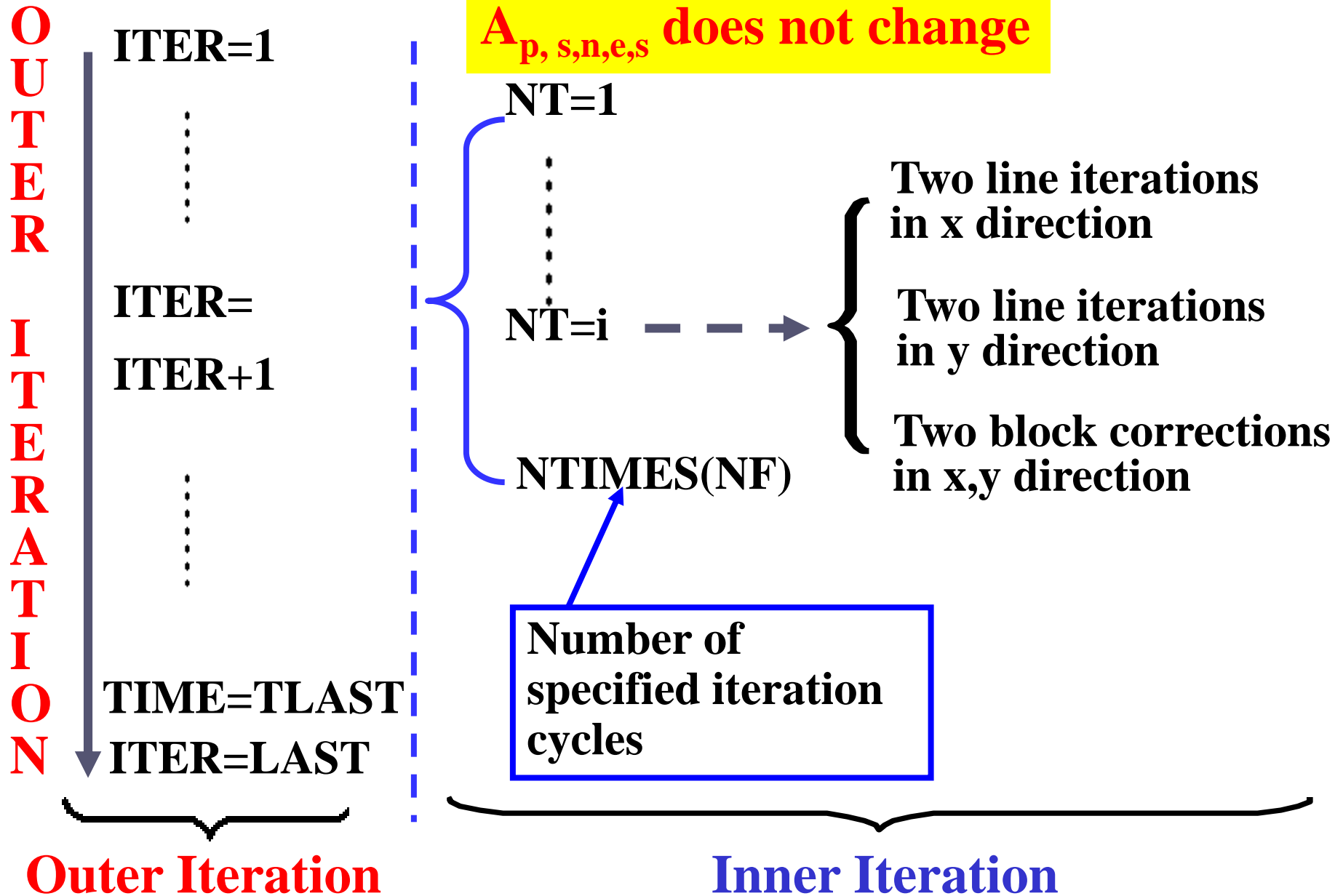
20



### Reporting Interval

1





## 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 adopted in Fluent. Therefore, the value of  $\Delta t$  will not affect the stability. However, it will affect the accuracy.

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

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

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

## 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) :

$$\frac{\sum |a_{nb}|}{|a_p|} \leq 1 \quad \left\{ \begin{array}{l} \leq 1 \text{ for all equations (a)} \\ < 1 \text{ at least for one equations (b)} \end{array} \right.$$

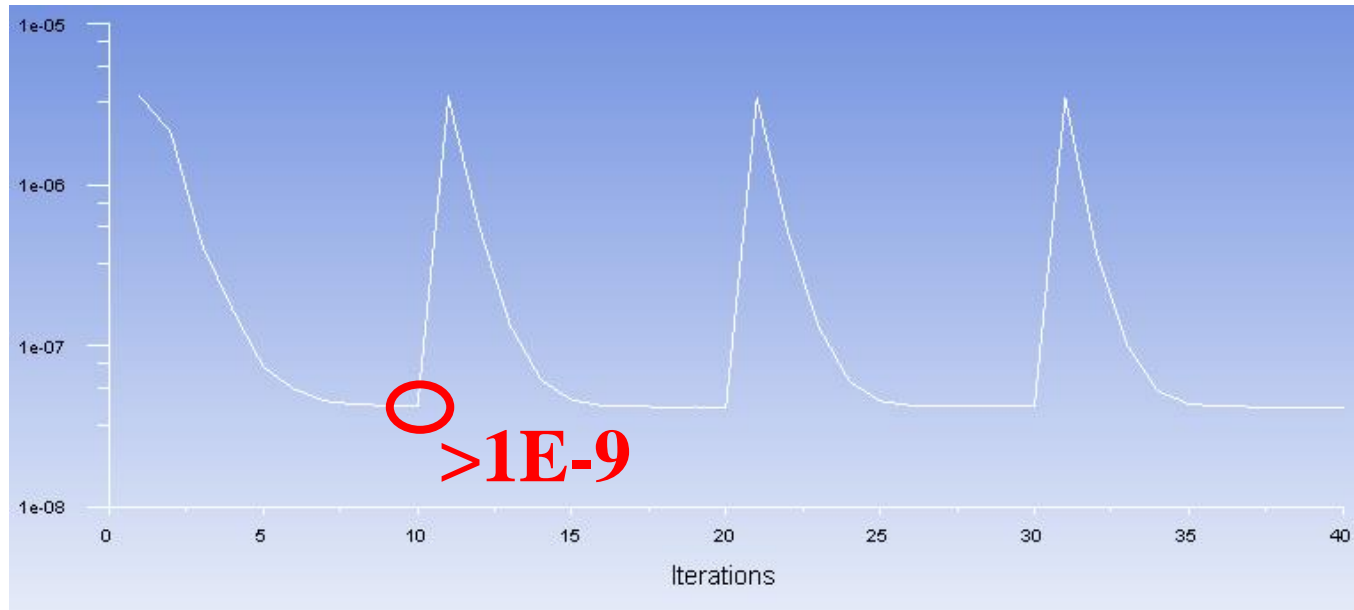
$$a_p = a_E + a_W + a_N + a_S + a_p^0 - S_p \Delta V$$

$$b = S_C \Delta V + a_{PP}^0 \phi_P^0 \quad a_p^0 = \frac{\rho_P \Delta V}{\Delta t}$$

However,  $\Delta t$  will affect the accuracy of the simulation results.

The following way is recommended by Fluent to set  $\Delta t$ :

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,  $\Delta t$  is too large.
3. If Fluent needs only a few iteration steps,  $\Delta t$  is too small.



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

Usually,  $\Delta t$  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  $\Delta t$  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

ball Fluent@DESKTOP-UN9RNO7 [3d, pbns, lam, transient]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

**Setup**

- General
- Models
- Materials
- Cell Zone Conditions
- Boundary Conditions
- Dynamic Mesh
- Reference Values
- Solution**
- Solution Methods
- Solution Controls
- Monitors
- Solution Initialization
- Calculation Activities
- Run Calculation**
- Results
- Graphics
- Animations
- Plots
- Reports
- Parameters & Customi...

**Run Calculation**

Check Case... Preview Mesh Motion...

Time Stepping Method: Fixed Time Step Size (s): 0.1

Number of Time Steps: 10000

Options

Extrapolate Variables

Data Sampling for Time Statistics

Sampling Interval: 1

Time Sampled (s): 0

Max Iterations/Time Step: 20 Reporting Interval: 1

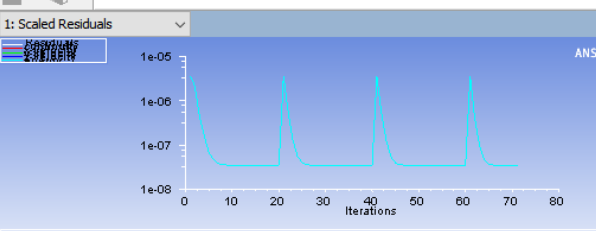
Working

Calculating the solution...

Cancel

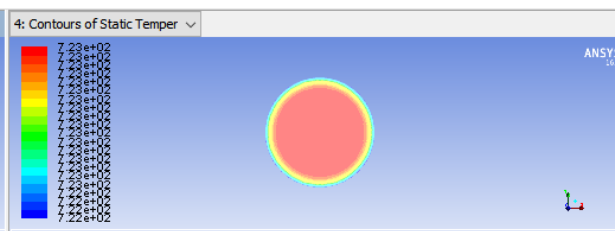
Help

1: Scaled Residuals



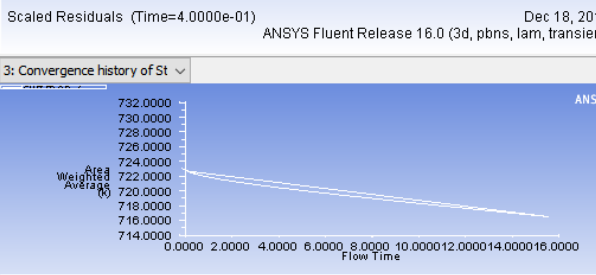
Scaled Residuals (Time=4.0000e-01) Dec 18, 2017  
ANSYS Fluent Release 16.0 (3d, pbns, lam, transient)

4: Contours of Static Temper



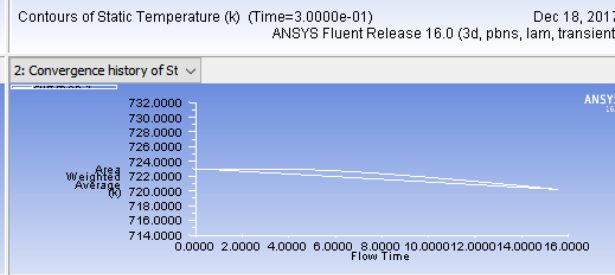
Contours of Static Temperature (K) (Time=3.0000e-01) Dec 18, 2017  
ANSYS Fluent Release 16.0 (3d, pbns, lam, transient)

3: Convergence history of St



Convergence history of Static Temperature on wall (Time=1.0000e-01) Dec 18, 2017  
ANSYS Fluent Release 16.0 (3d, pbns, lam, transient)

2: Convergence history of St



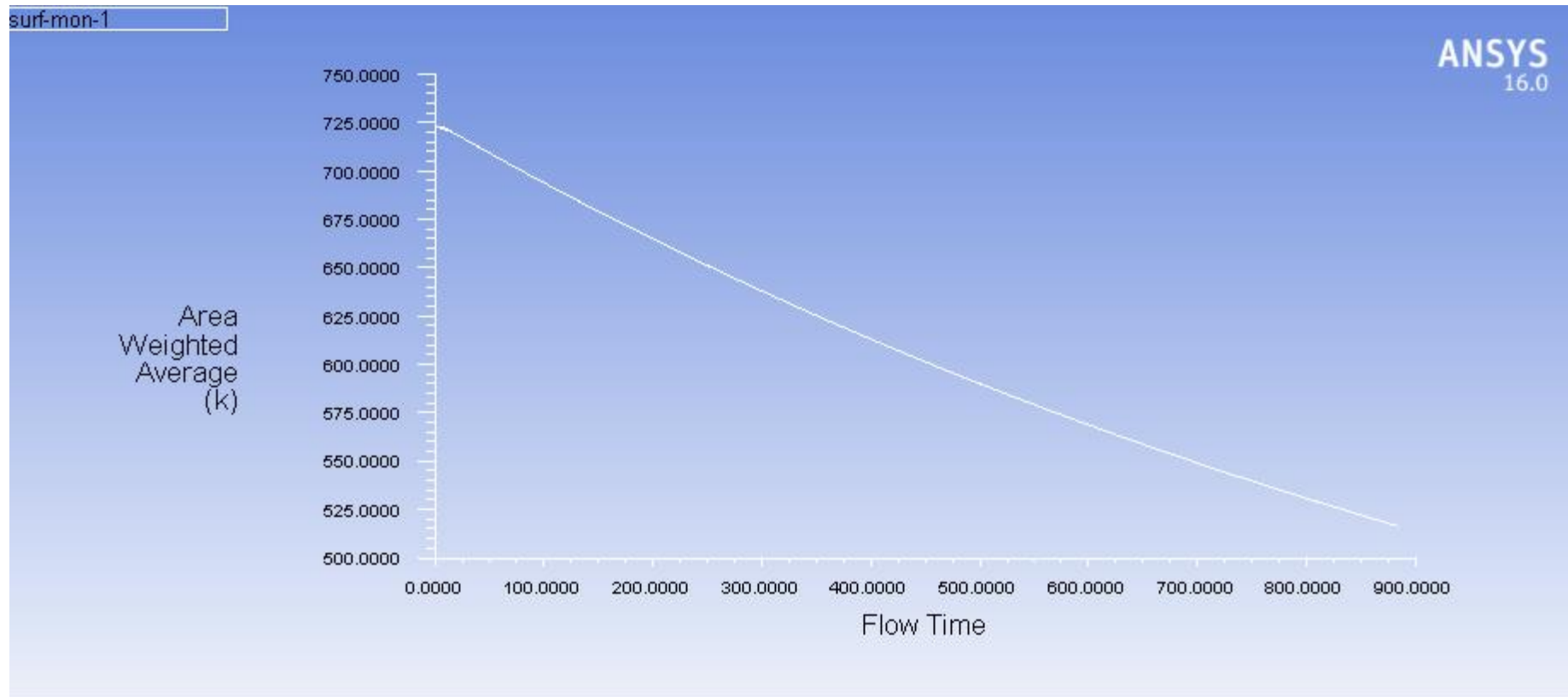
Convergence history of Static Temperature on point-0 (Time=1.0000e-01) Dec 18, 2017  
ANSYS Fluent Release 16.0 (3d, pbns, lam, transient)

```

Flow time = 0.300000011920929s, time step = 3
9997 more time steps

Updating solution at time level N... done.
iter  continuity  x-velocity  y-velocity  z-velocity  energy  surf-mon-1  surf-mon-2  time/iter
60  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5353e-08  7.2300e+02  7.2241e+02  0:00:04 20
61  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5126e-06  7.2300e+02  7.2232e+02  0:00:03 19
62  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.8373e-07  7.2300e+02  7.2232e+02  0:00:02 18
63  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  9.7828e-08  7.2300e+02  7.2232e+02  0:00:02 17
64  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  4.6966e-08  7.2300e+02  7.2232e+02  0:00:04 16
65  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.7259e-08  7.2300e+02  7.2232e+02  0:00:03 15
66  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5583e-08  7.2300e+02  7.2232e+02  0:00:02 14
67  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5166e-08  7.2300e+02  7.2232e+02  0:00:02 13
68  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5153e-08  7.2300e+02  7.2232e+02  0:00:01 12
69  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5010e-08  7.2300e+02  7.2232e+02  0:00:01 11
70  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5220e-08  7.2300e+02  7.2232e+02  0:00:03 10
iter  continuity  x-velocity  y-velocity  z-velocity  energy  surf-mon-1  surf-mon-2  time/iter
71  0.0000e+00  0.0000e+00  0.0000e+00  0.0000e+00  3.5140e-08  7.2300e+02  7.2232e+02  0:00:02 9
    
```

# The average temperature on “point-0” change by time is as below:



Convergence history of Static Temperature on point-0 (Time=1.0000e-01)

Dec 18, 2017  
ANSYS Fluent Release 16.0 (3d, pbns, lam, transient)

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

**Steel: density: 7753 kg/m<sup>3</sup>; 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	戩坤池	√	√	√	√	√		√
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)

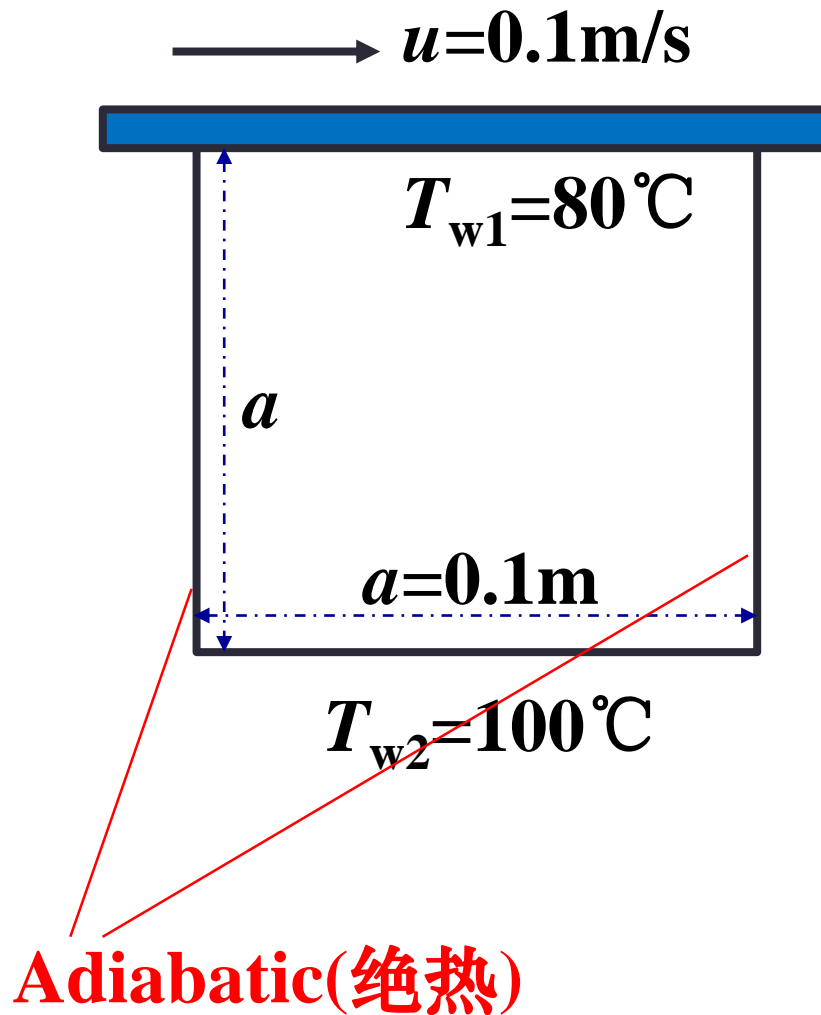
### 顶盖驱动流动换热问题

**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  $T_{w1} = 80^\circ\text{C}$  is moving with velocity  $u=0.1\text{m/s}$  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  $T_{w2} = 100^\circ\text{C}$ . 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:** 
$$\frac{\partial(\rho C_p u T)}{\partial x} + \frac{\partial(\rho C_p v T)}{\partial y} = \lambda \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right)$$

We should estimate  $Re$  to determine laminar or turbulent state.

Know:

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

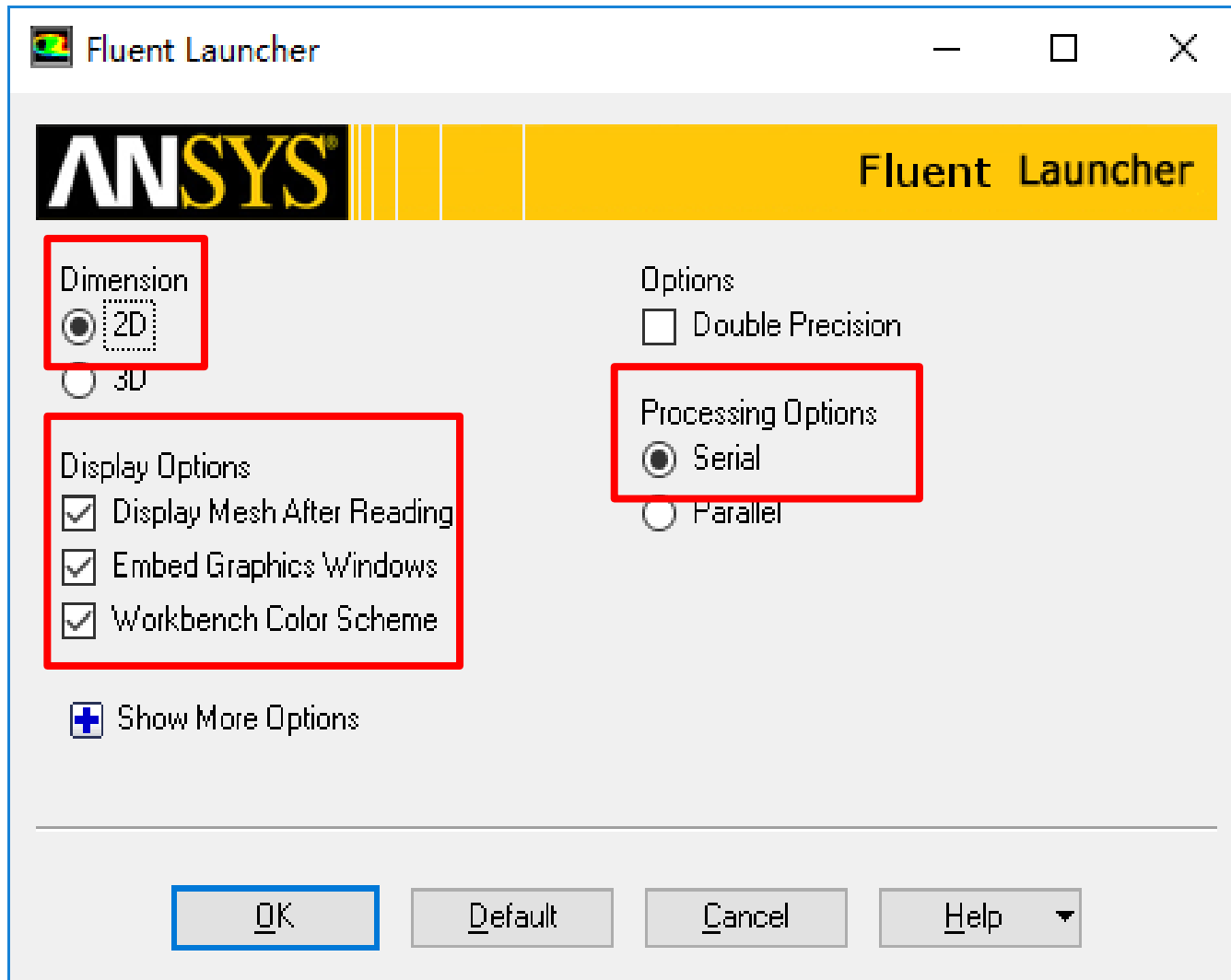
$$Re = \frac{ul}{\nu} = 452$$

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”**

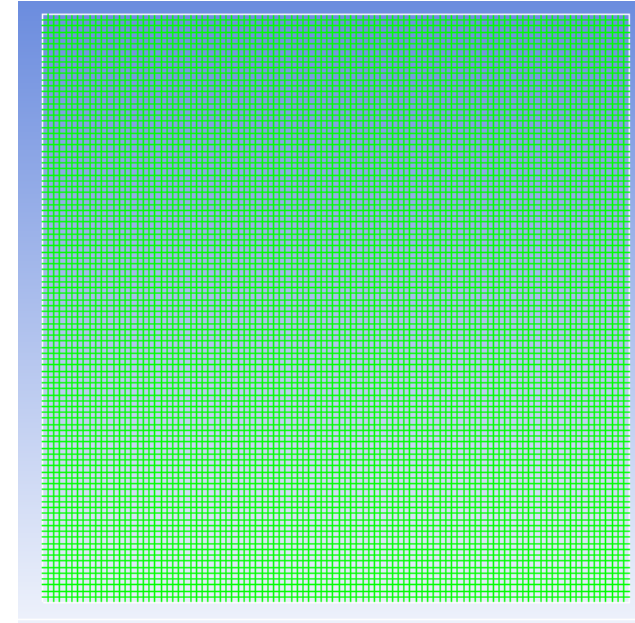
```
> Reading "E:\fluent-case\flow-5\flow2.cas"...  
Done.  
  9801 quadrilateral cells, zone  8, binary.  
 19404 2D interior faces, zone  9, binary.  
   99 2D wall faces, zone 10, binary.  
   99 2D wall faces, zone 11, binary.  
  198 2D wall faces, zone 12, binary.  
 10000 nodes, binary.  
 10000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



```

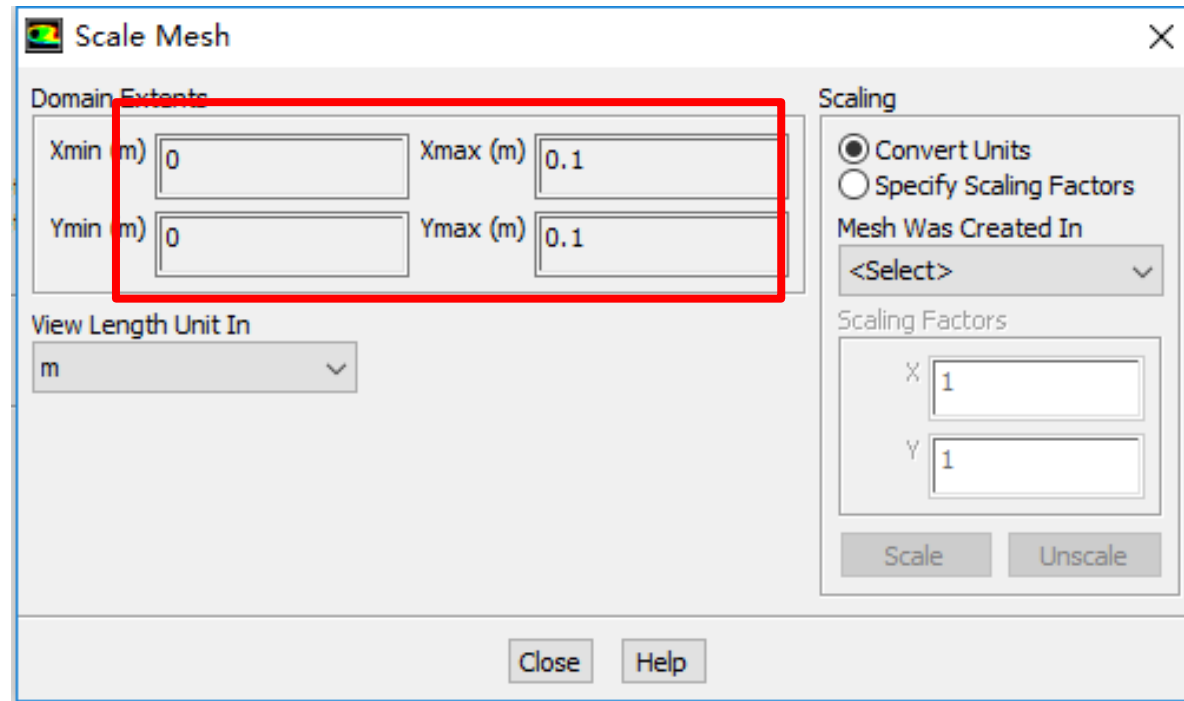
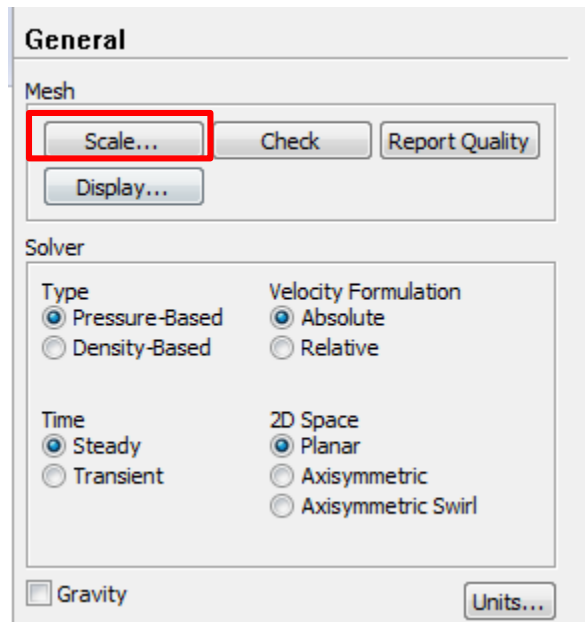
Domain Extents:
  x-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
  y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
Volume statistics:
  minimum volume (m3): 1.020304e-06
  maximum volume (m3): 1.020304e-06
  total volume (m3): 1.000000e-02
Face area statistics:
  minimum face 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 = 1.000000e+00
  Maximum Aspect Ratio = 1.41422e+00
    
```

## 2st step: Scale the domain size

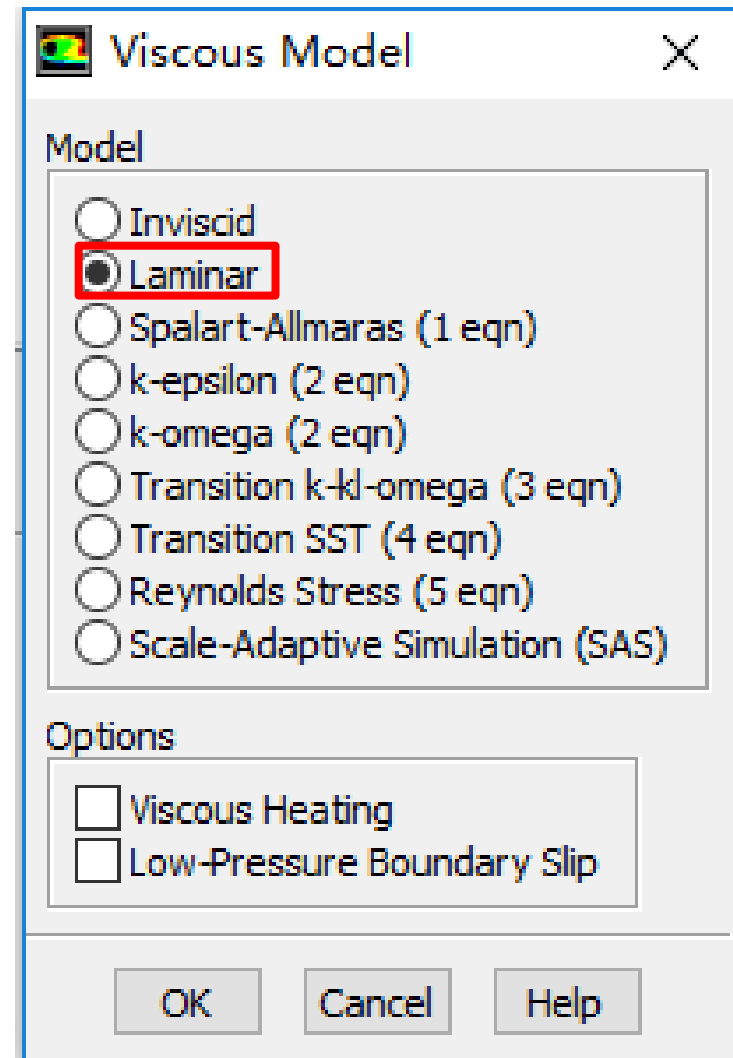
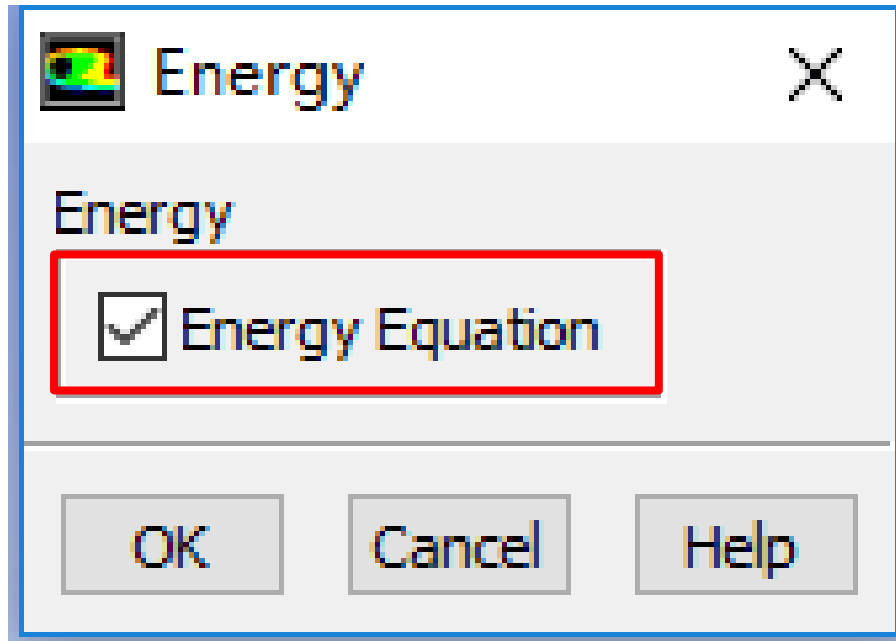
### General→Scale



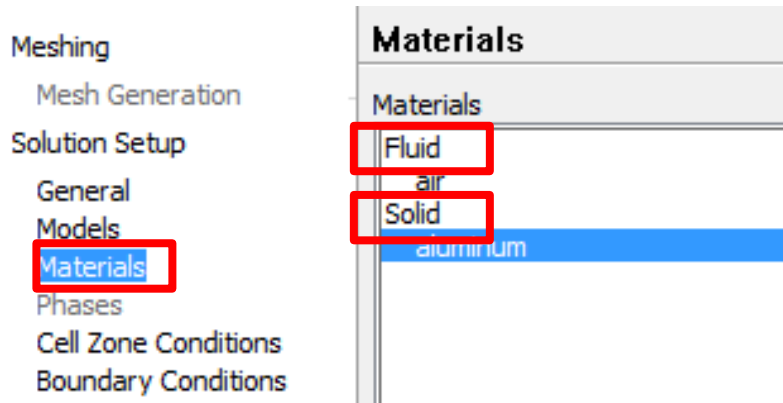
**The mesh is generated in ICEM with unit of m. So we do not need to scale the size for this problem.**

## Step 3: Choose the physicochemical model

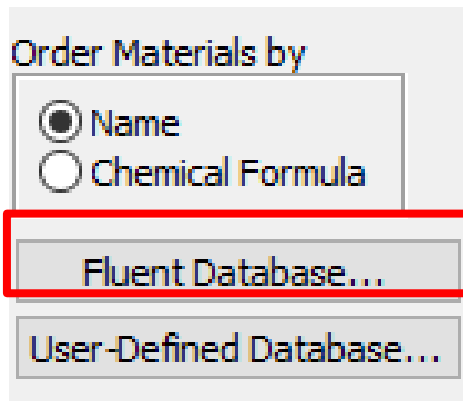
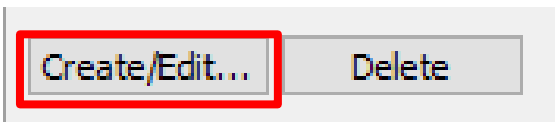
Based on the governing equations you are going to solve, select the related model in Fluent.



## 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

Cell Zone Conditions

Zone

inner

Phase

mixture

Type

fluid

Edit...

Copy...

Profiles...

Parameters...

Operating Conditions...

Display Mesh...

Type

fluid

Fluid

Zone Name

inner

Material Name

air

Edit...

Frame Motion  Source Terms

Mesh Motion  Fixed Values

Porous Zone

Reference Frame | Mesh Motion | Porous Zone | Embedded LES | Reaction

This page is not applicable under current settings.

## 6st step: Define the Boundary conditions

### Boundary Conditions

Zone

bottom-wall  
fixed-wall  
int\_solid  
move-wall

Phase

mixture

Type

interior

ID

9

Edit...

Copy...

Profiles...

Parameters...

Operating Conditions...

Display Mesh...

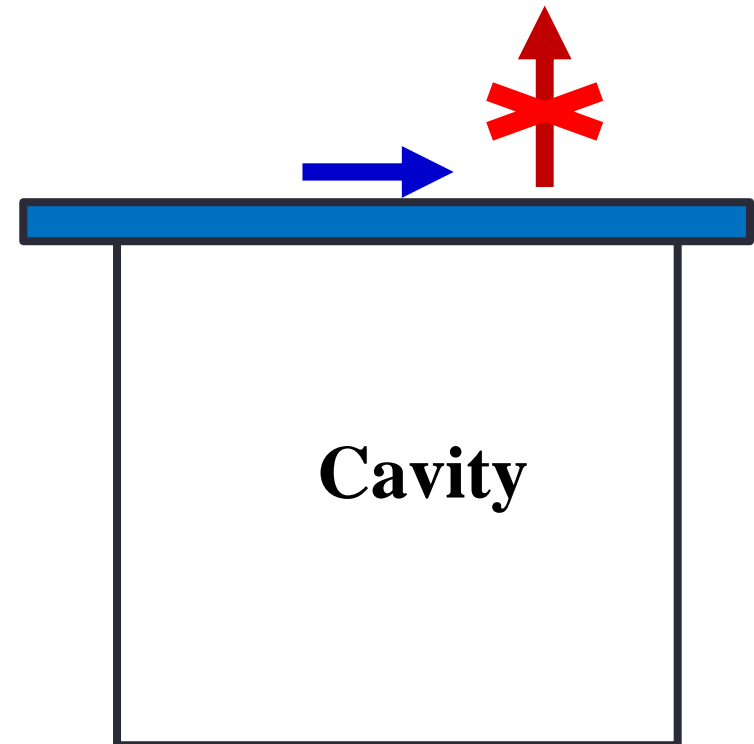
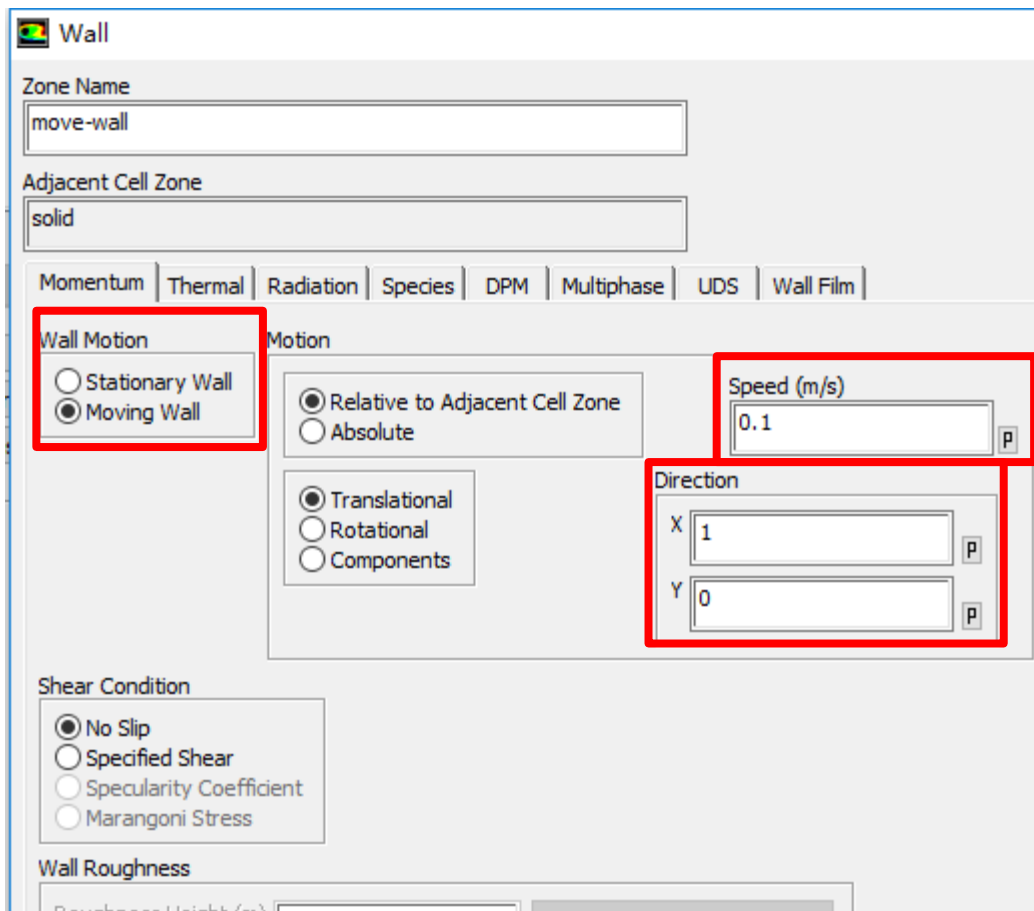
Periodic Conditions...

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.

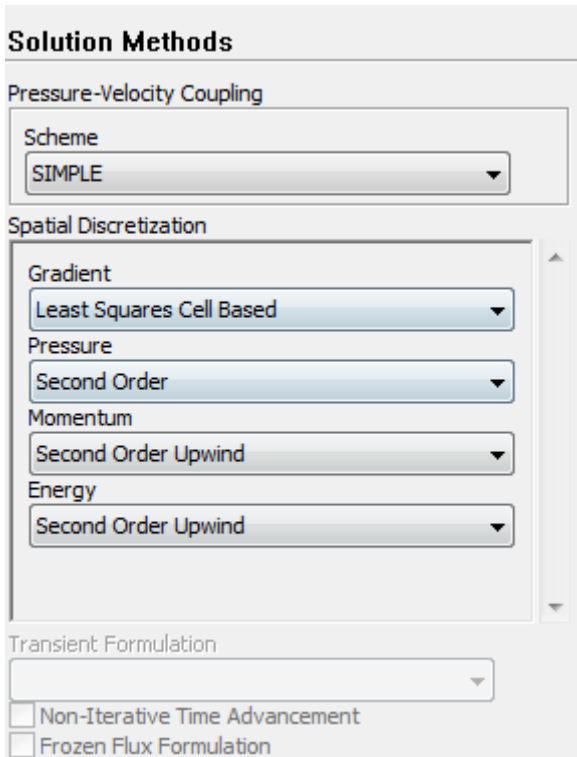
The top wall is moving. We will discuss it in detail.

“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

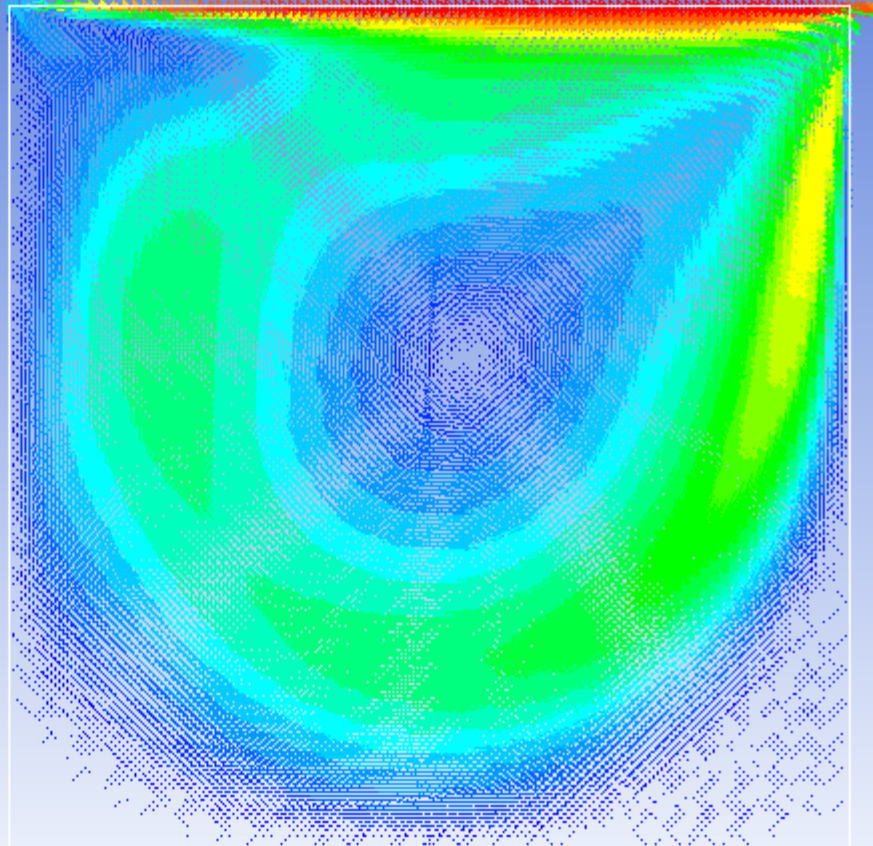
## Step 10: Post-processing results

# Velocity Vector

1: Velocity Vectors Colored B ▾



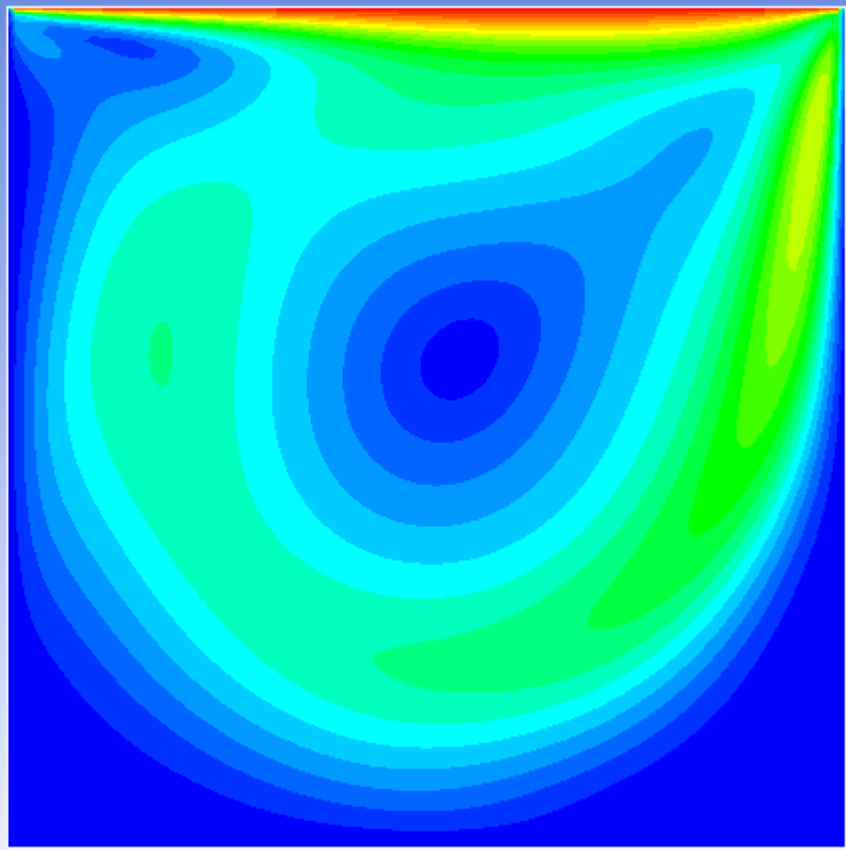
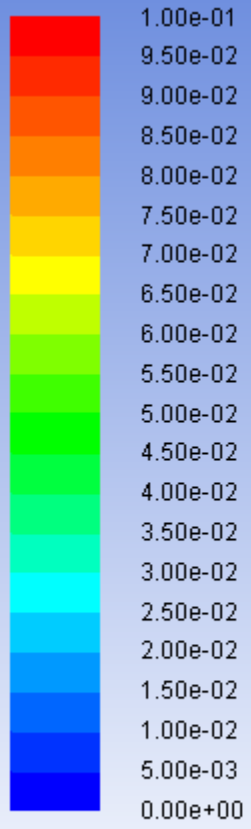
- 9.40e-02
- 8.93e-02
- 8.46e-02
- 7.99e-02
- 7.52e-02
- 7.05e-02
- 6.58e-02
- 6.11e-02
- 5.64e-02
- 5.17e-02
- 4.70e-02
- 4.23e-02
- 3.76e-02
- 3.29e-02
- 2.82e-02
- 2.35e-02
- 1.88e-02
- 1.41e-02
- 9.40e-03
- 4.70e-03
- 8.34e-07



Velocity Vectors Colored By Velocity Magnitude (m/s)

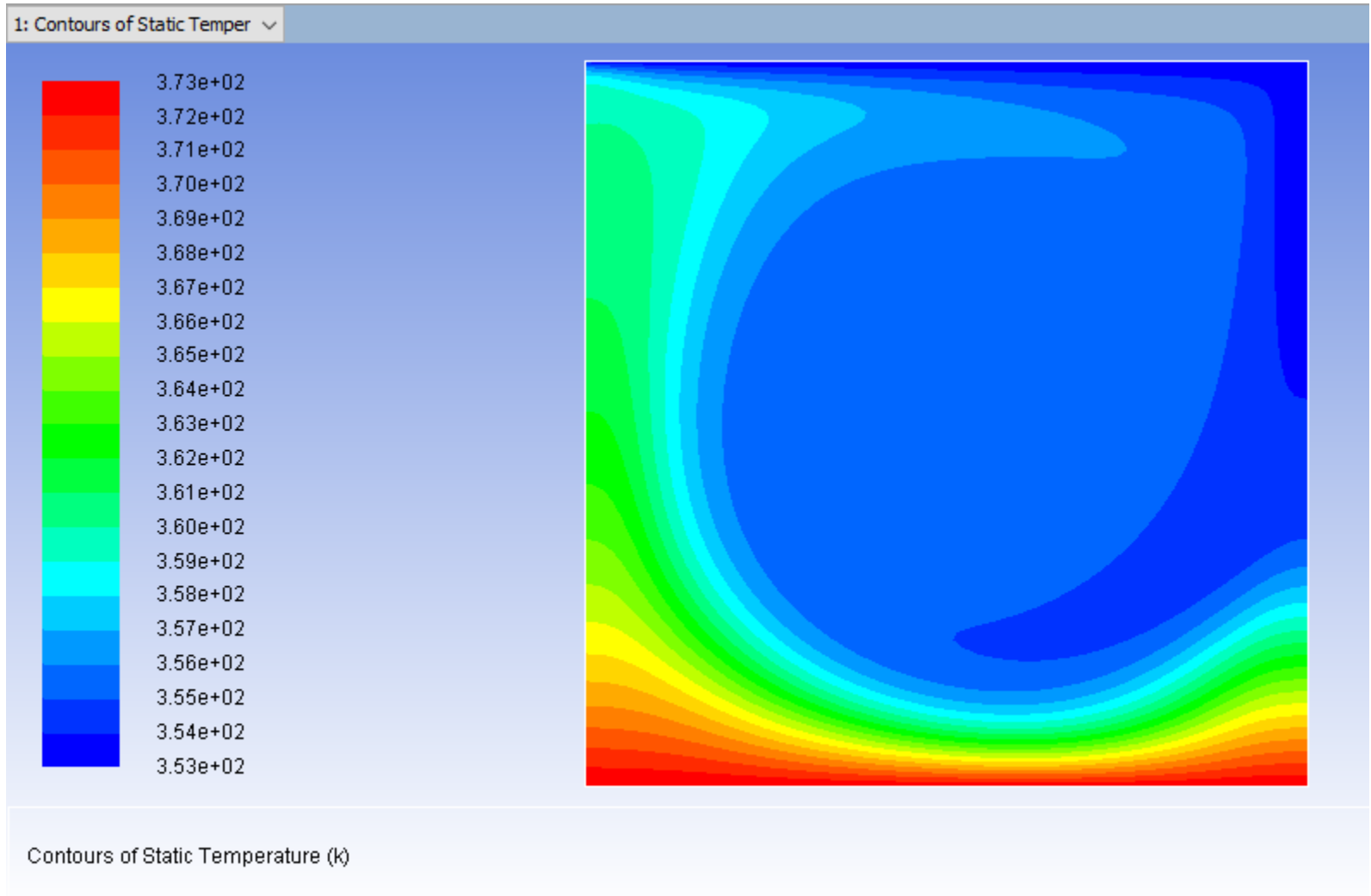
# Velocity magnitude

1: Contours of Velocity Magn



Contours of Velocity Magnitude (m/s)

# 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**

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



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

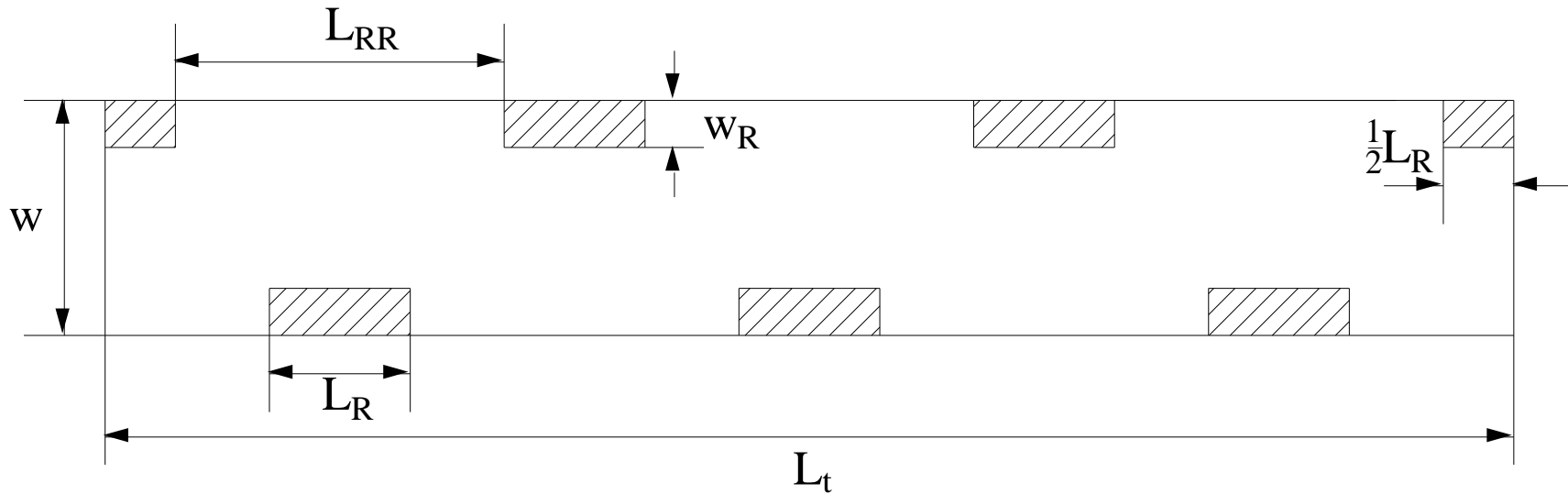
### 微通道内流动换热问题

**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  $T_f=20^\circ\text{C}$  flows into the inlet of a micro channel with rectangular ribs (MC-RR) with velocity  $u=0.1\text{m/s}$ . The side walls of MC-RR are heated with a uniform heat flux  $q = 30\text{W/cm}^2$ .

**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**

<b>Geometrical Parameters</b>	$W$	$L_{RR}$	$W_R$	$L_R$	$L_t$
<b>Value/mm</b>	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:

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

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:

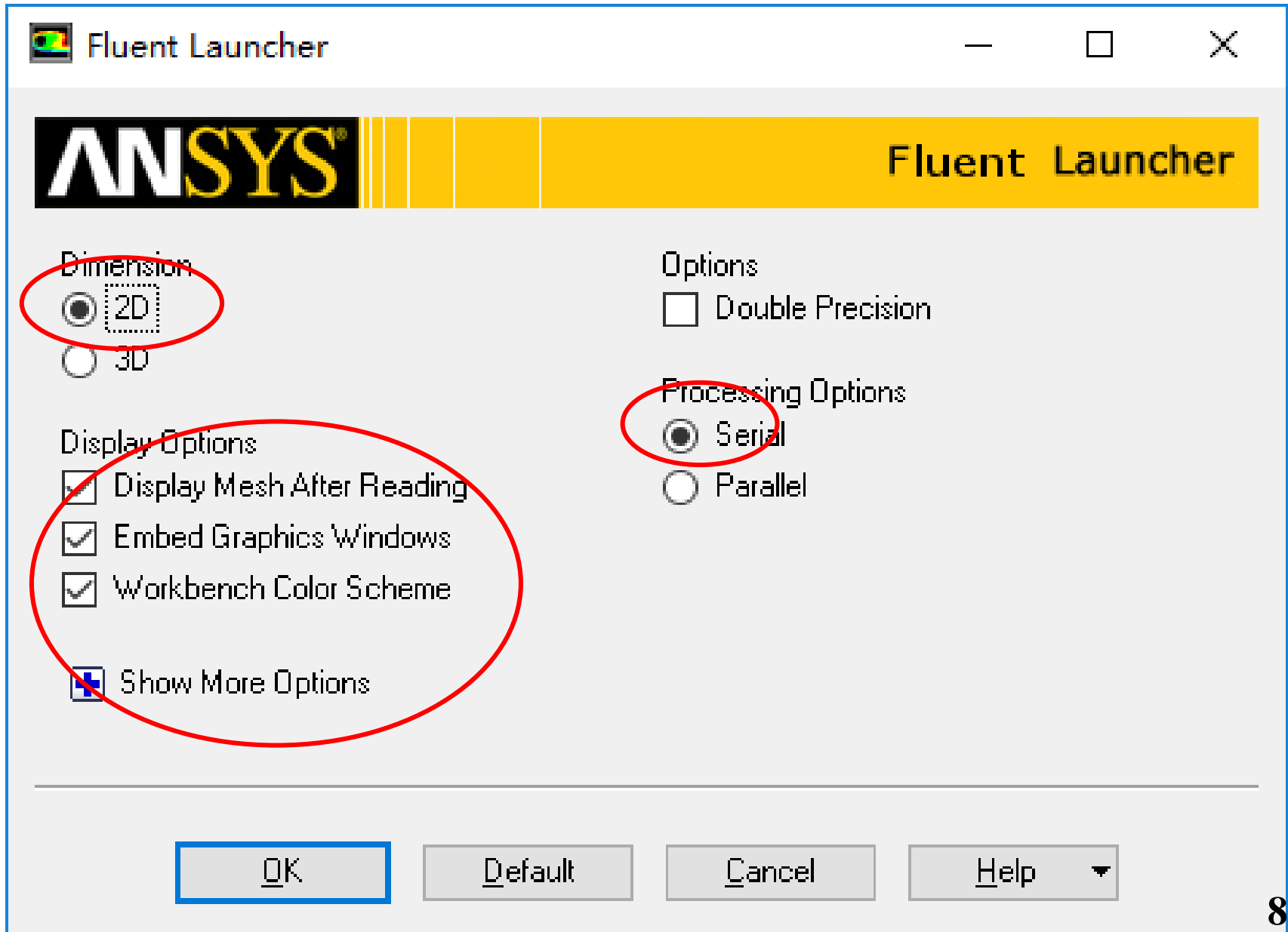
$$0 = k_s \left( \frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} \right)$$

where  $T_s$  is solid temperature and  $k_s$  is solid thermal conductivity

## Boundary condition:

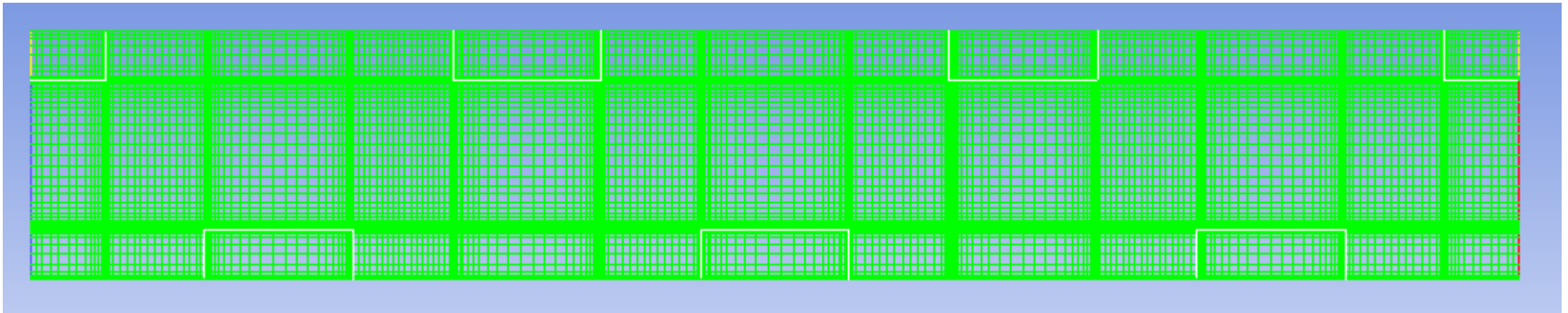
1.	channel inlet $x=0$ $u = u_f$ For fluid $T_f = T_{in} = 293.15\text{K}$
2.	Channel outlet $x=3\text{mm}$ $P_f = P_{out} = 1\text{atm}$
3.	fluid/solid surface $u = v = 0$ $-k_s \left( \frac{\partial T_s}{\partial n} \right) = -k_f \left( \frac{\partial T_f}{\partial n} \right)$ $T_f = T_s$ where $(n)$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

# 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

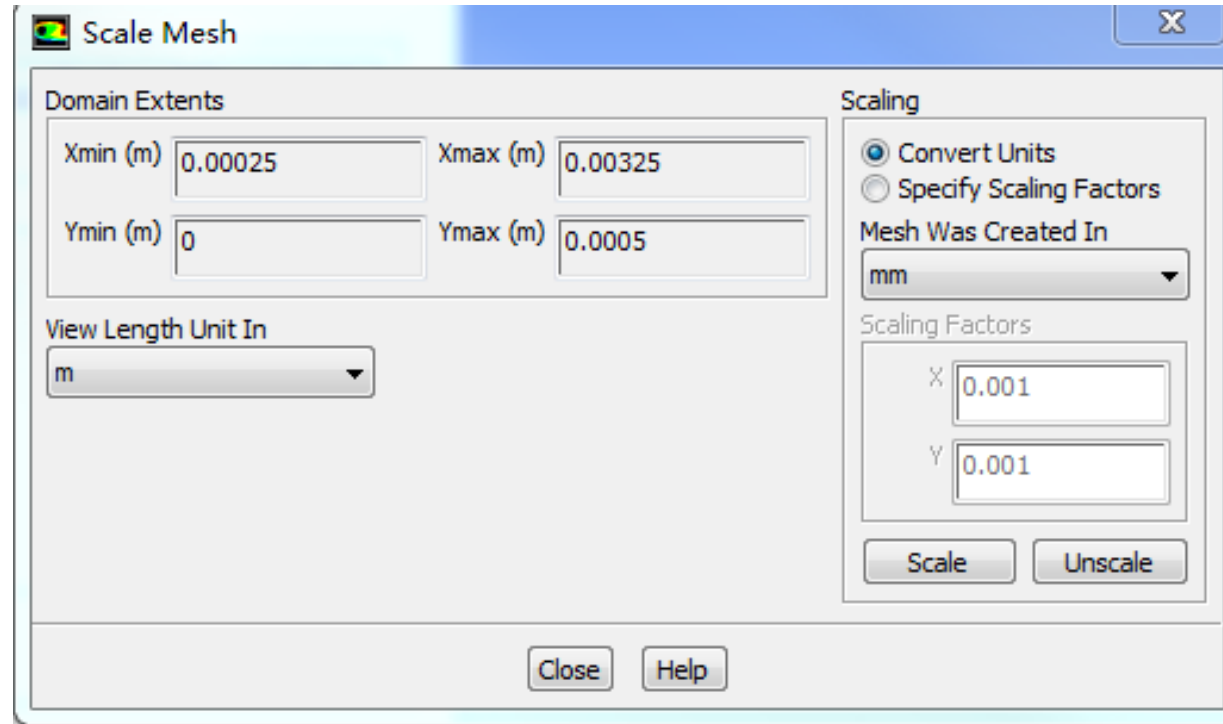
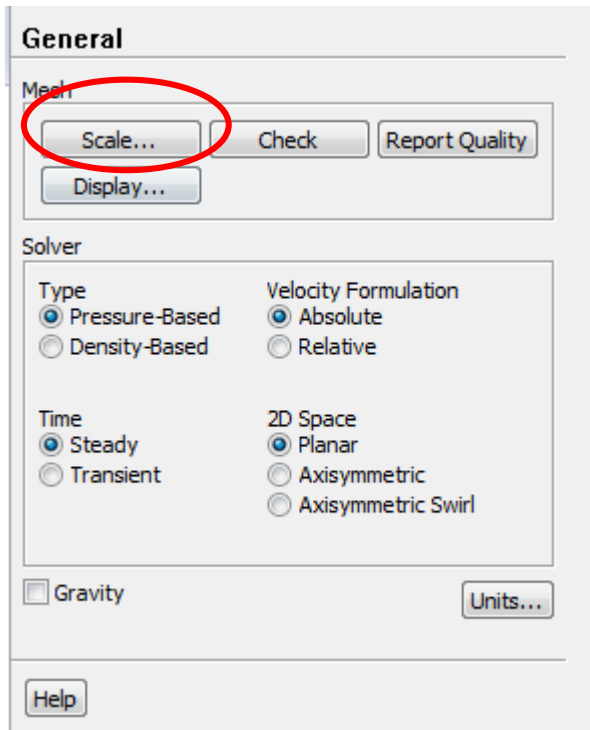
maximum face area (m2): 2.495997e-05

Checking mesh.....

Done.

# Step 2: Scale the domain size

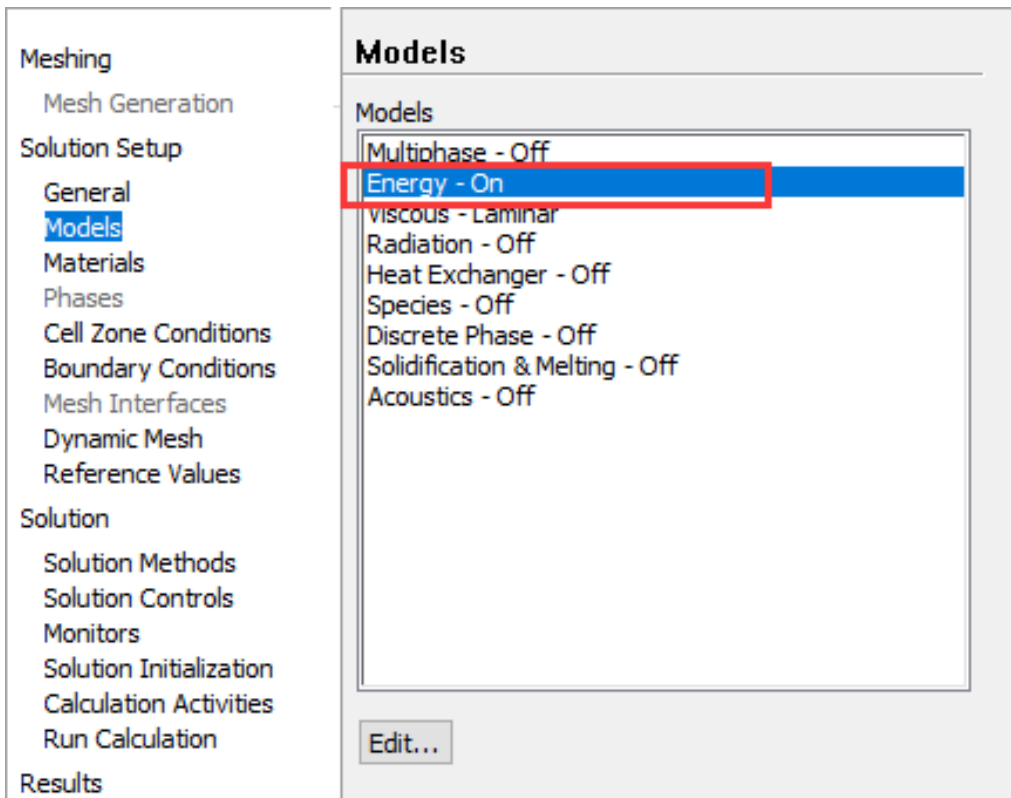
General → Scale



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.**

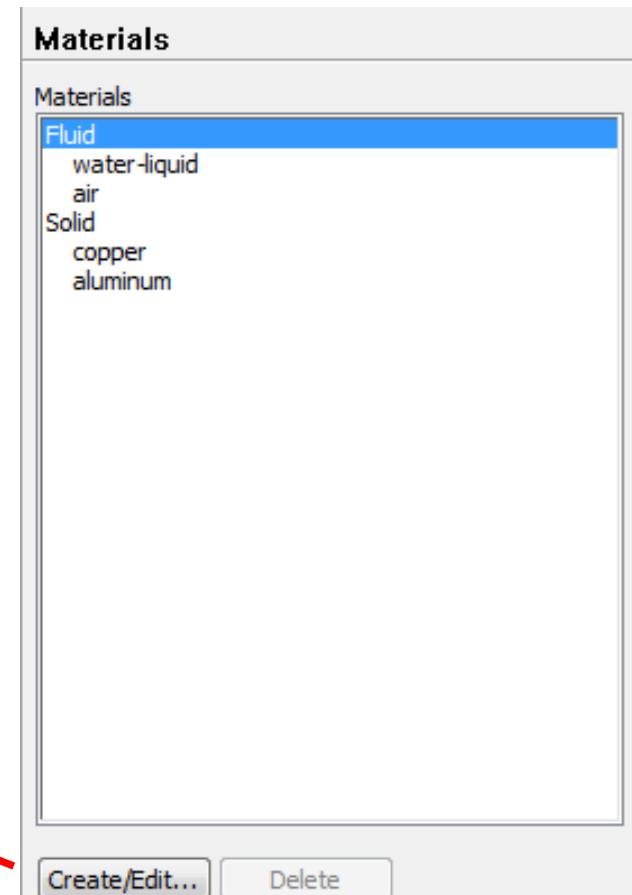
## Step 4: Define the material properties

Define the properties required for modeling! For pure heat conduction problem studied here,  $\rho$ ,  $C_p$  and  $\lambda$  should be defined.

**Solution Setup** → **Materials**

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

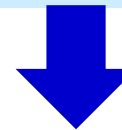
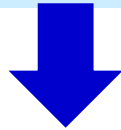
Click the **Create/Edit** button to add **Water** and **Copper**.



# Step 5: Define zone condition

**Solution Setup → Cell Zone Condition**

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



Fluid

Zone Name  
fluid

Material Name **water-liquid** Edit...

Frame Motion    Source Terms  
 Mesh Motion    Fixed Values  
 Porous Zone

Reference Frame | Mesh Motion | Porous Zone | Embedded LES | Reaction | Source Terms | Fixed Values | Multiphase

Solid

Zone Name  
solid

Material Name **copper** Edit...

Frame Motion    Source Terms  
 Mesh Motion    Fixed Values

Reference Frame | Mesh Motion | Source Terms | Fixed Values

Origin

constant  
 constant

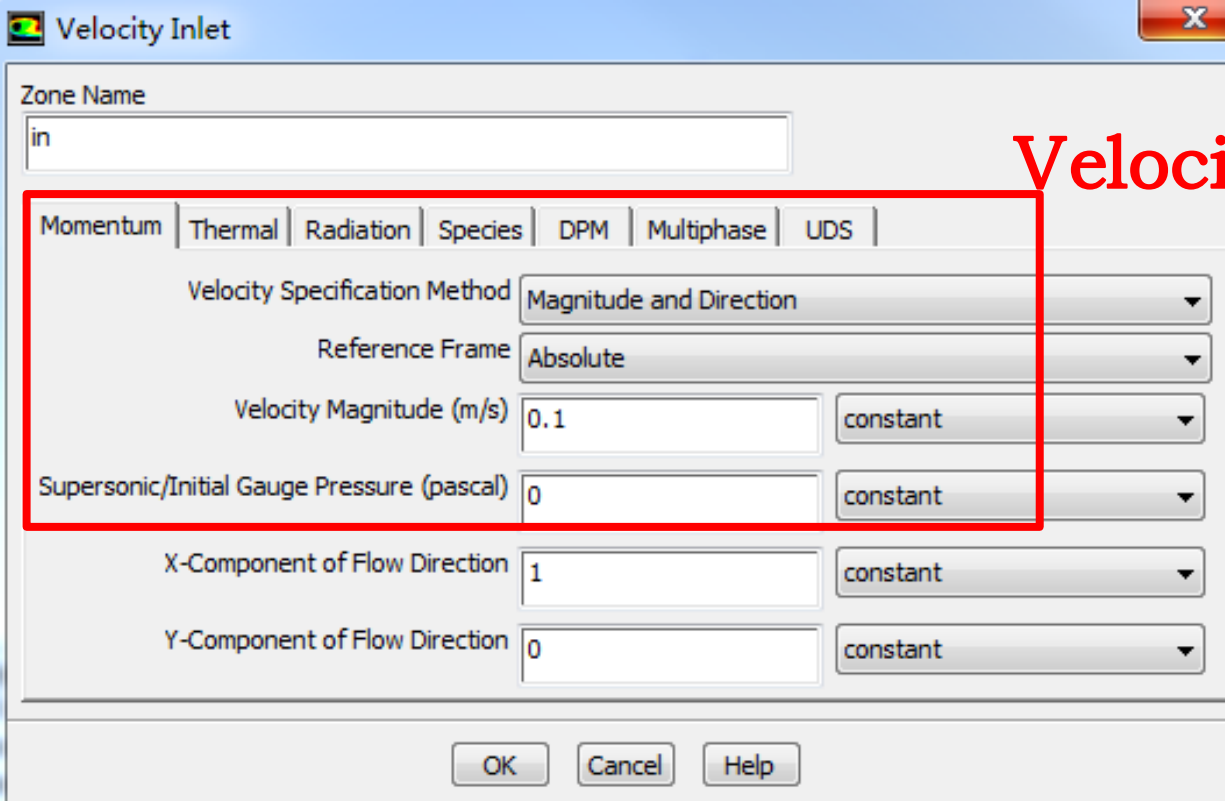
Zone Name  
fluid

Material Name **water-liquid** Edit...

Frame Motion    Source Terms

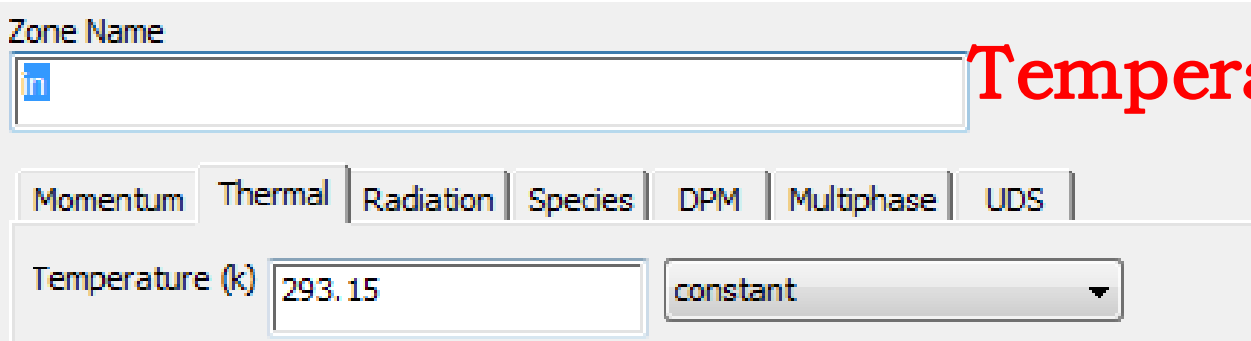
# Step 6: Define the boundary condition

Inlet



Velocity Inlet dialog box showing boundary condition settings. The 'Momentum' tab is selected and highlighted with a red box. The 'Velocity Specification Method' is set to 'Magnitude and Direction', 'Reference Frame' is 'Absolute', 'Velocity Magnitude (m/s)' is 0.1, and 'Supersonic/Initial Gauge Pressure (pascal)' is 0. The 'X-Component of Flow Direction' is 1 and 'Y-Component of Flow Direction' is 0. The 'Zone Name' is 'in'. Buttons for 'OK', 'Cancel', and 'Help' are at the bottom.

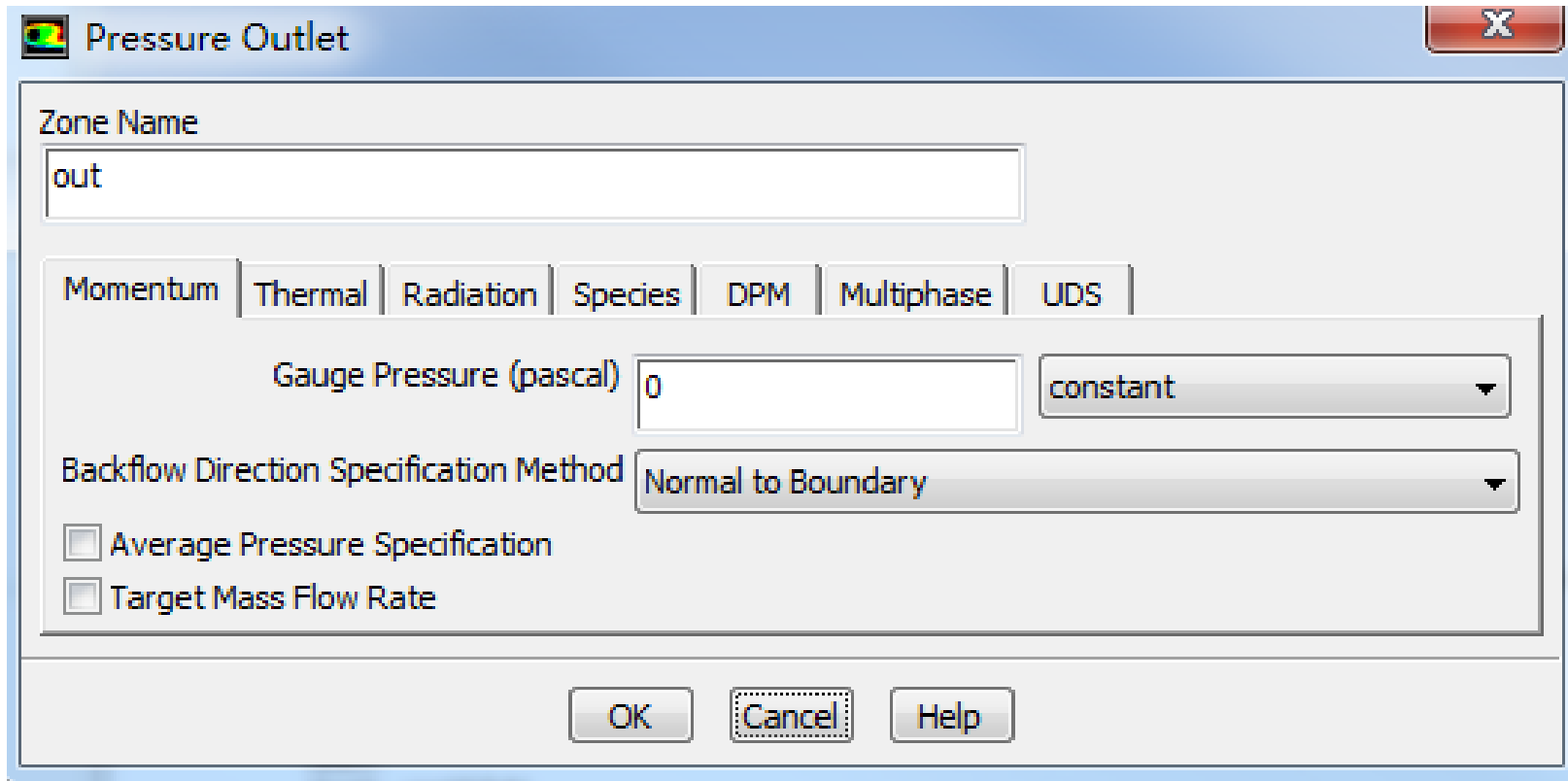
Velocity



Temperature dialog box showing boundary condition settings. The 'Thermal' tab is selected. The 'Temperature (k)' is set to 293.15 and the method is 'constant'. The 'Zone Name' is 'in'. Buttons for 'OK', 'Cancel', and 'Help' are at the bottom.

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  $0.5\rho U^2$

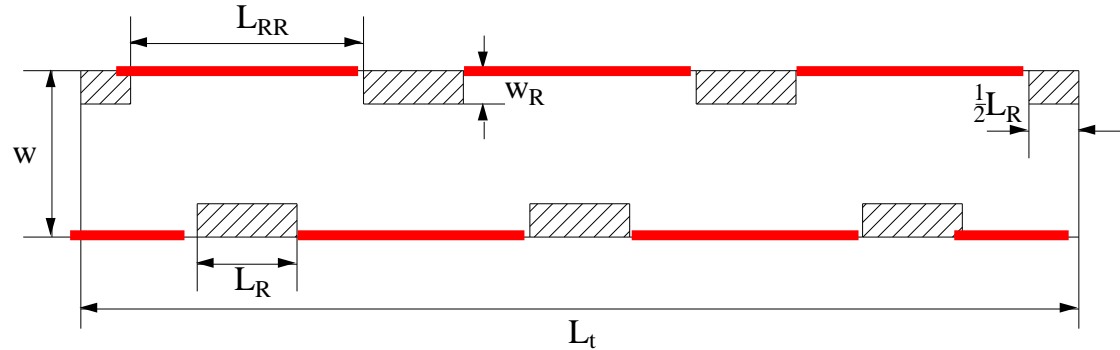
**Is related to the velocity.**

**Total pressure (总压):**

**= Static pressure + dynamic pressure**

# Wall

$$q = 30W/cm^2$$



Wall
X

Zone Name

Adjacent Cell Zone

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

Thermal Conditions

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

Heat Flux (w/m<sup>2</sup>)  constant

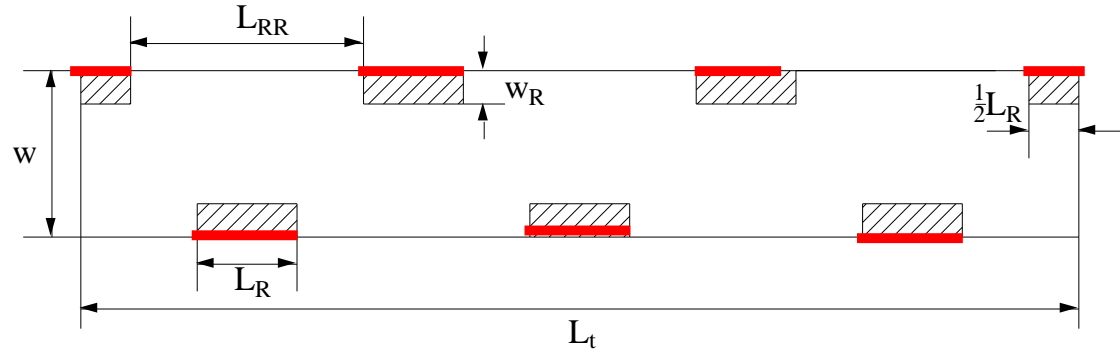
Wall Thickness (m)  P

Heat Generation Rate (w/m<sup>3</sup>)  constant

Material Name  
 Edit...

# Wall

$$q = 30W/cm^2$$



Wall

Zone Name: fluid\_wall

Adjacent Cell Zone: fluid

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

Thermal Conditions

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

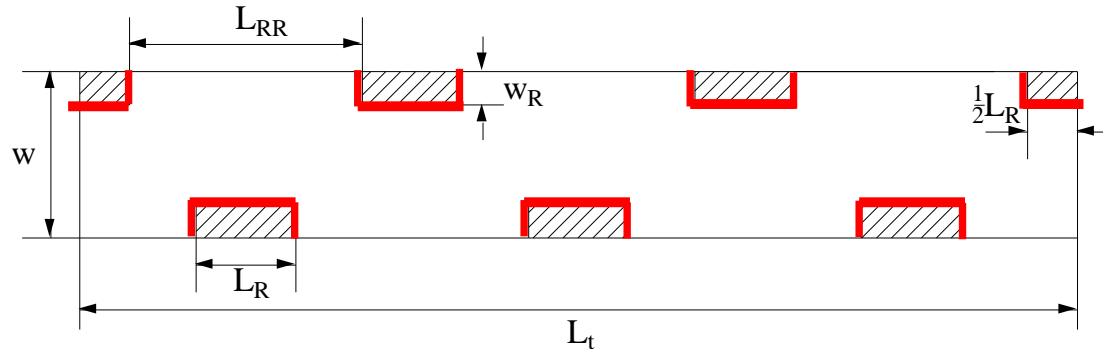
Heat Flux (w/m<sup>2</sup>): 300000 constant

Wall Thickness (m): 0 P

Heat Generation Rate (w/m<sup>3</sup>): 0 constant

Material Name: copper Edit...

## 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”.

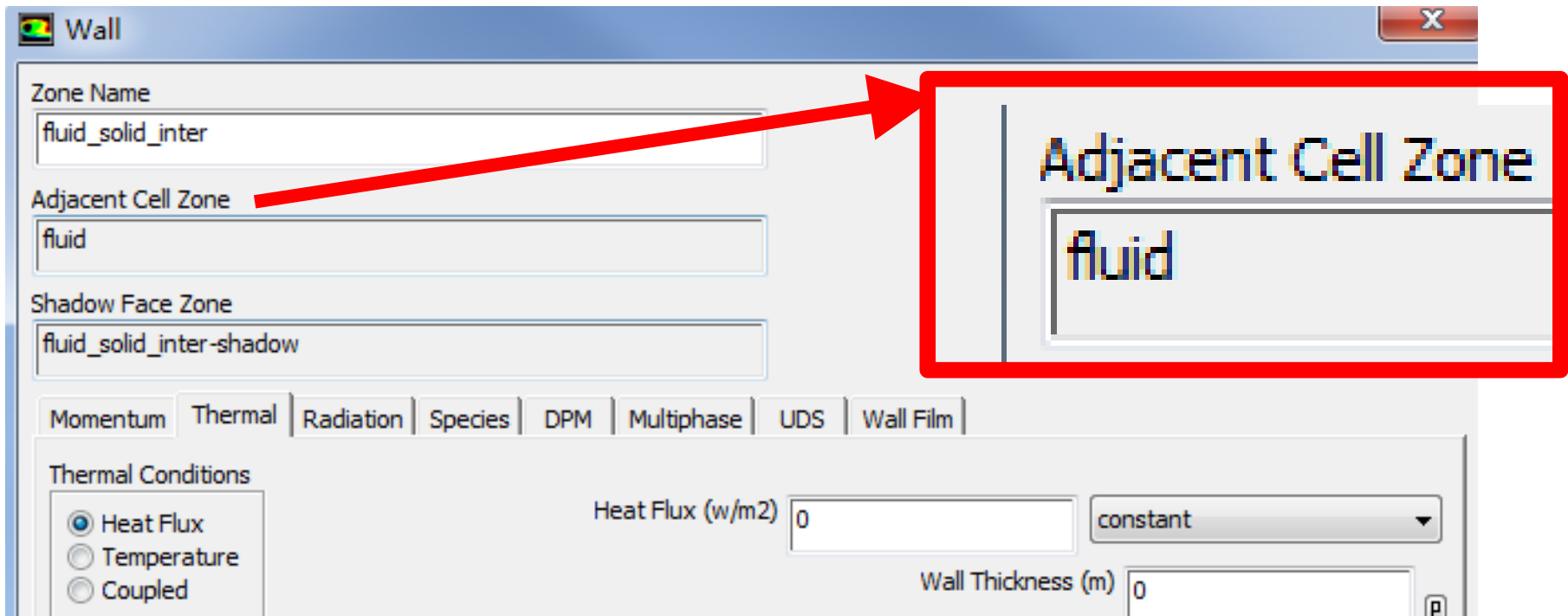
## Thermal Conditions

- Heat Flux
- Temperature
- Coupled

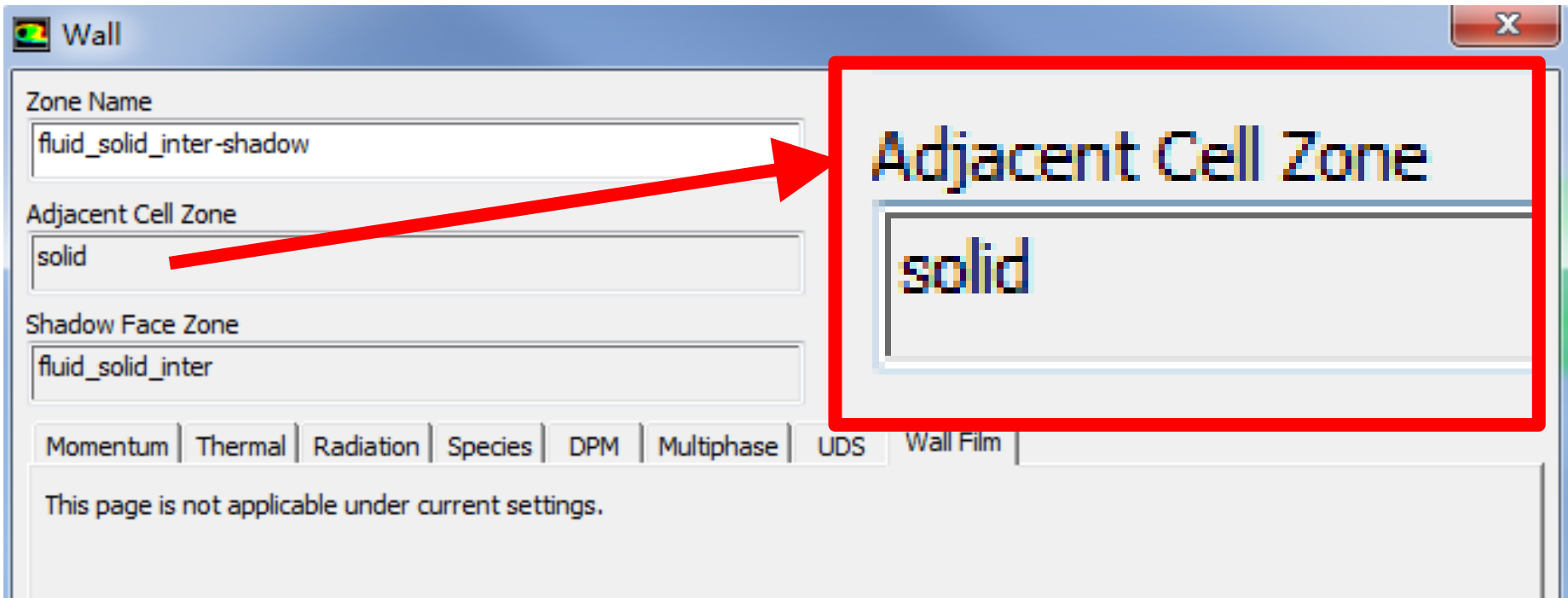
- Heat flux
- Temperature
- Coupled

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.**

**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

Thermal Conditions

- Heat Flux
- Temperature
- Coupled

Thermal Conditions

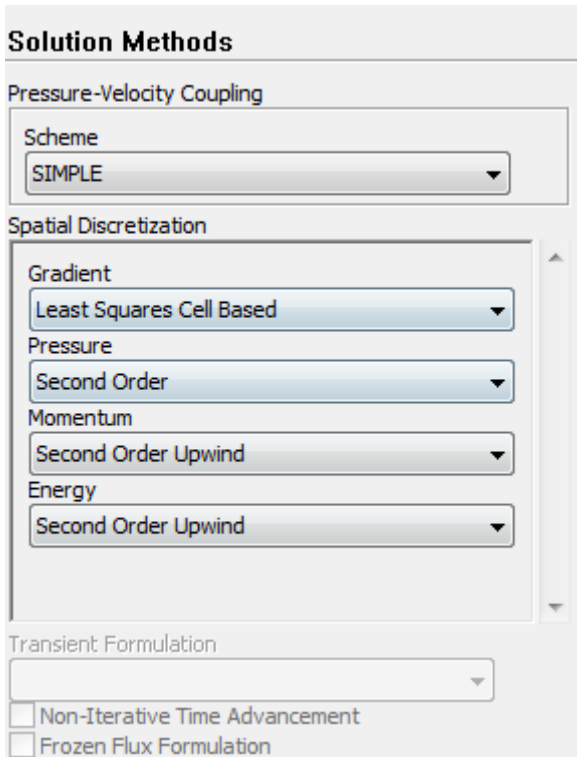
- Heat Flux
- Temperature
- Coupled

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**.

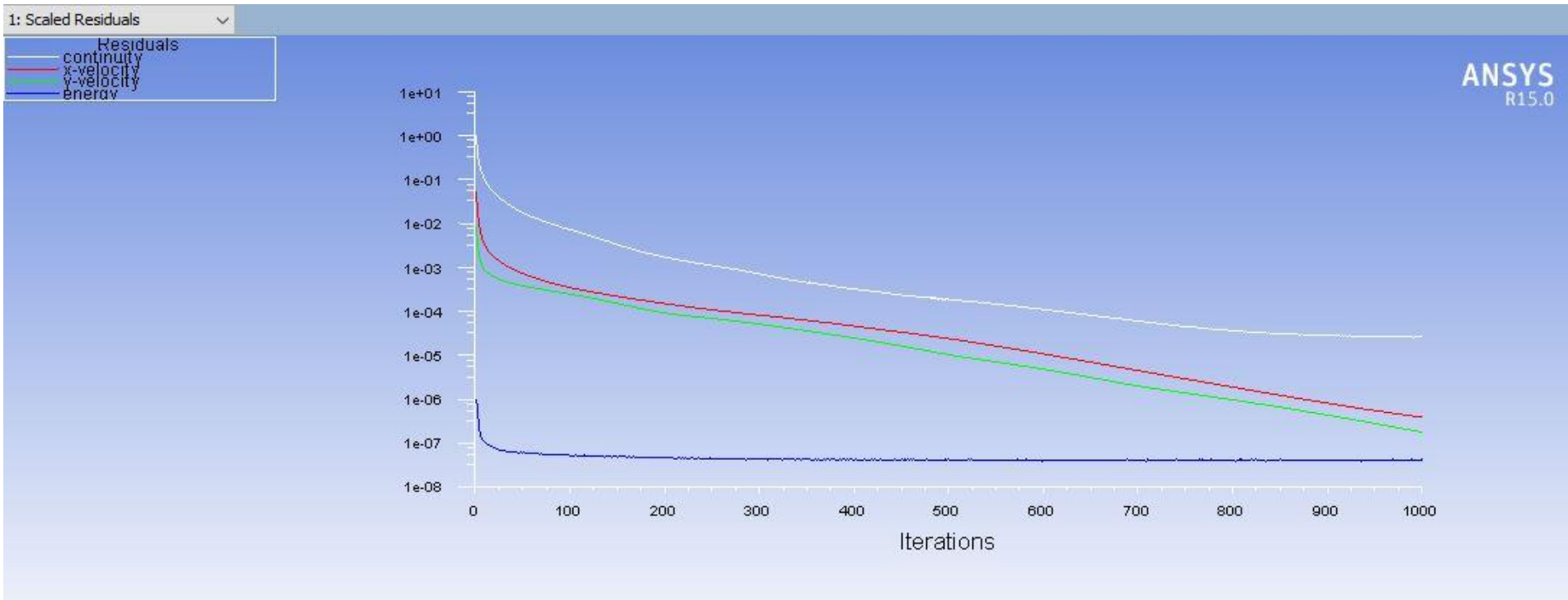
## 8st step: Initialization

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

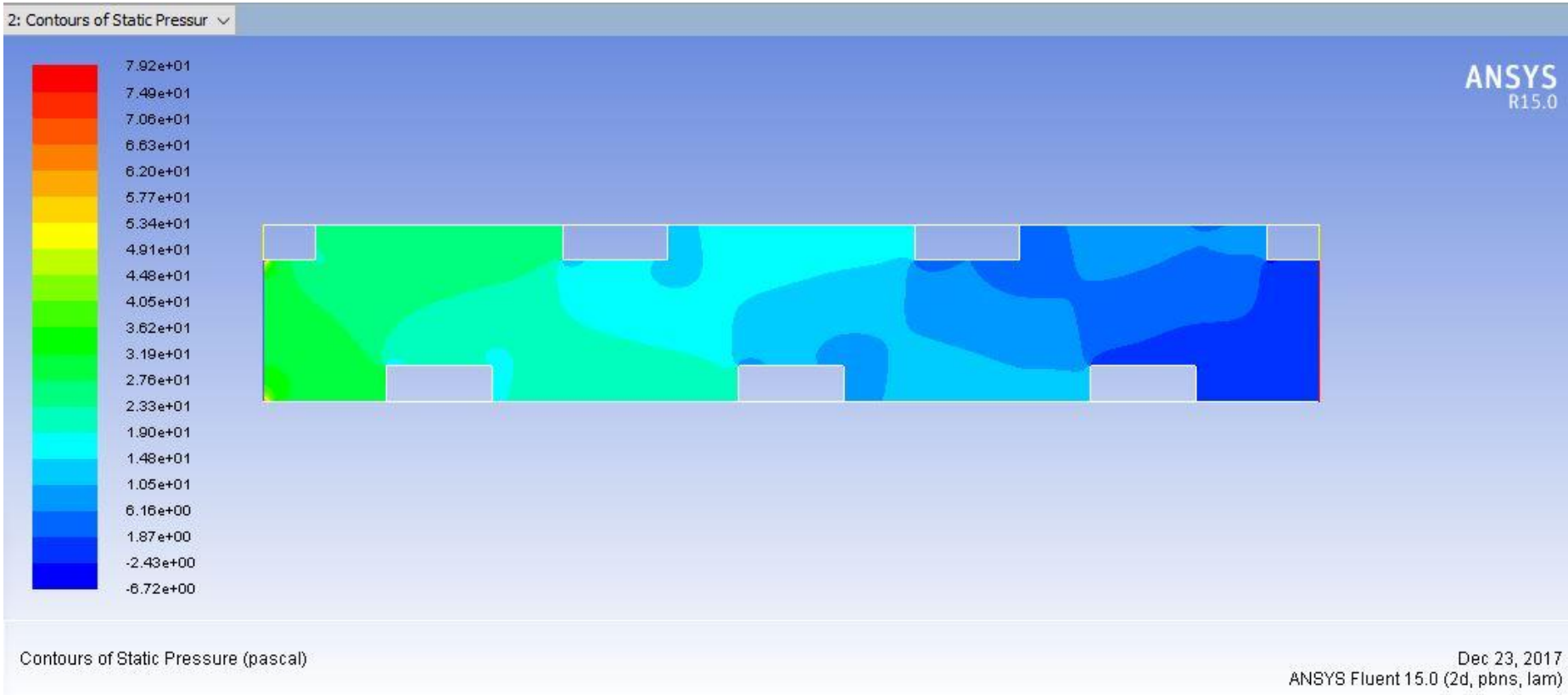


ANSYS  
R15.0

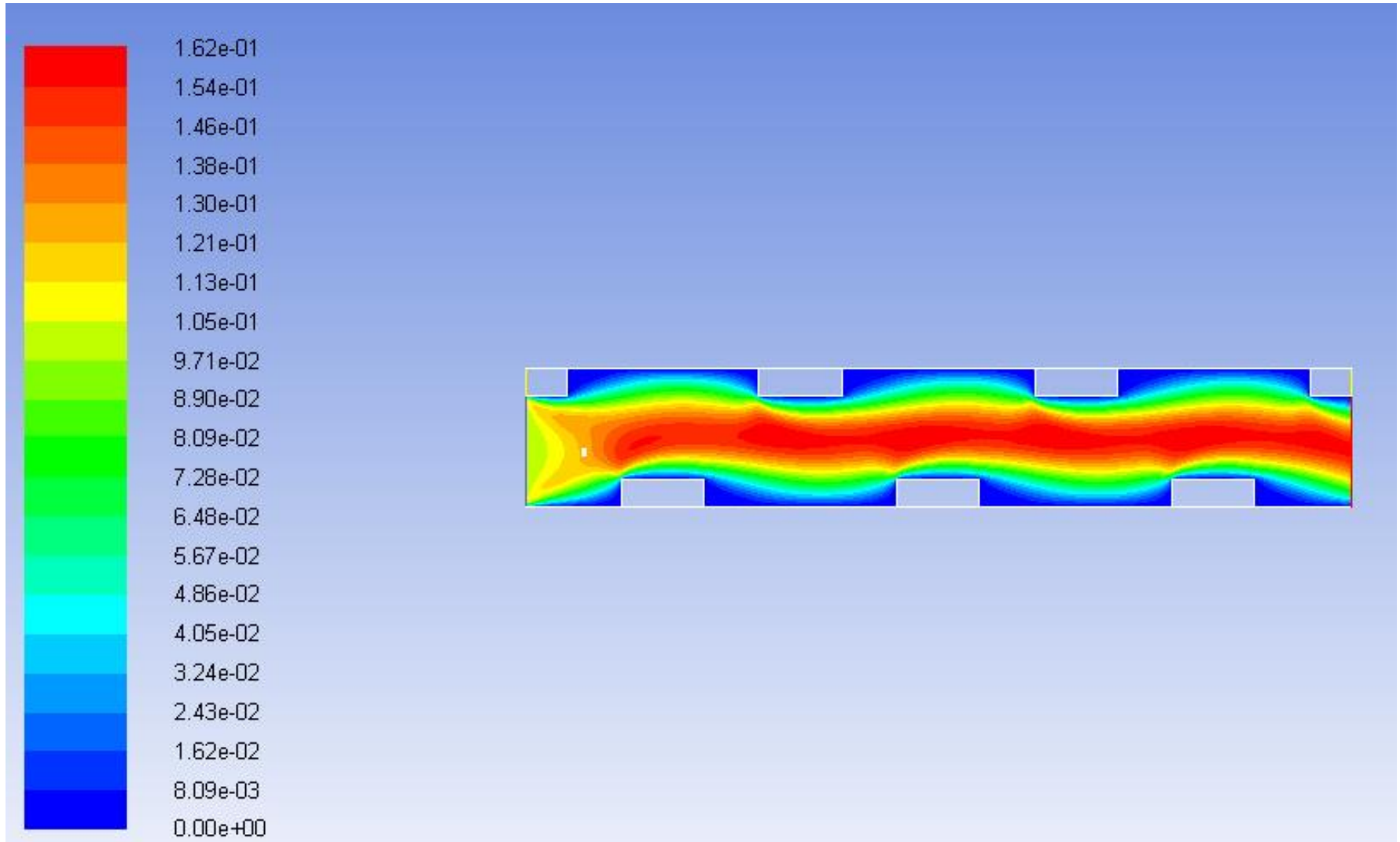
Scaled Residuals

Dec 23, 2017  
ANSYS Fluent 15.0 (2d, pbns, lam)

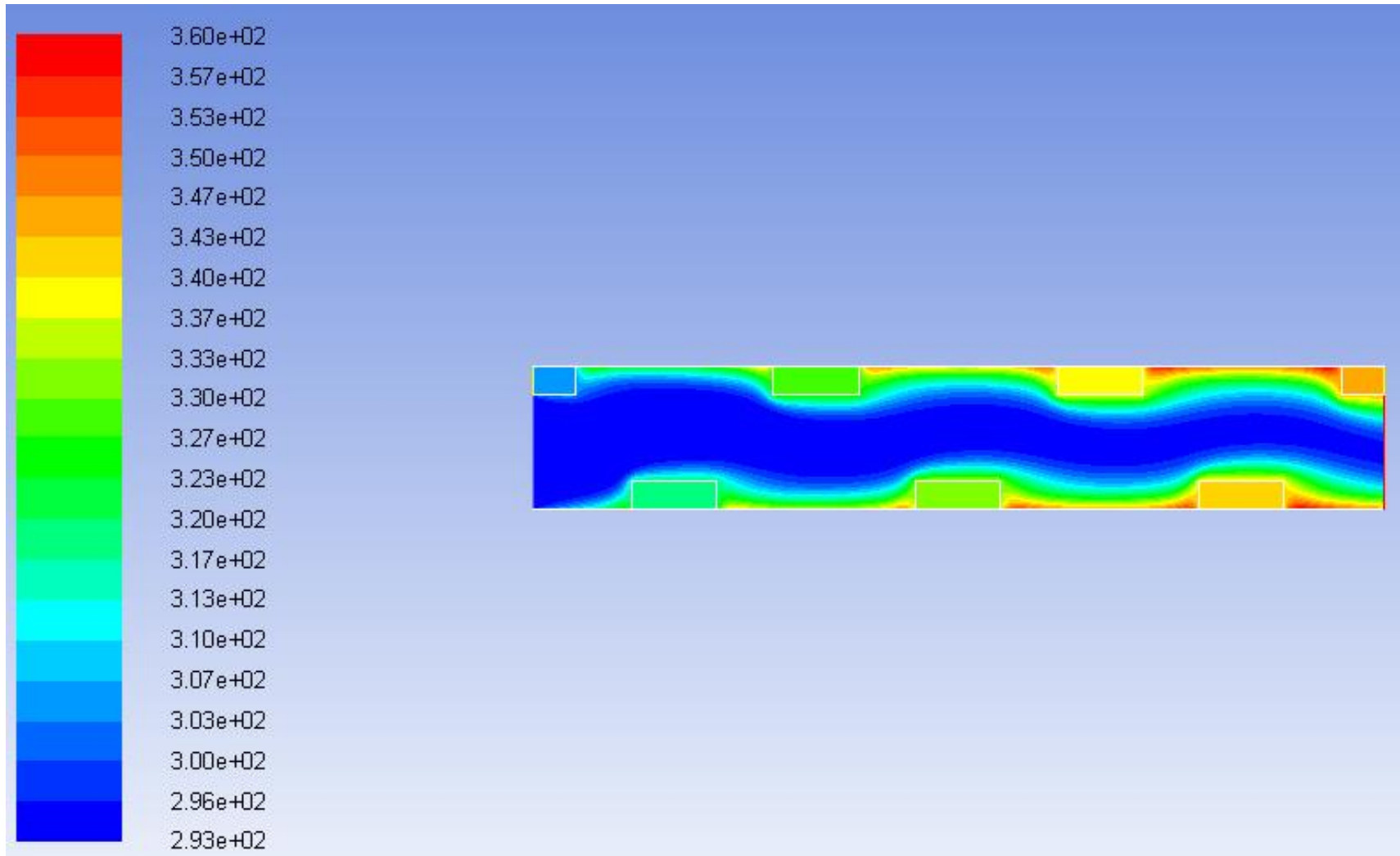
# Contours of static pressure (Pa)



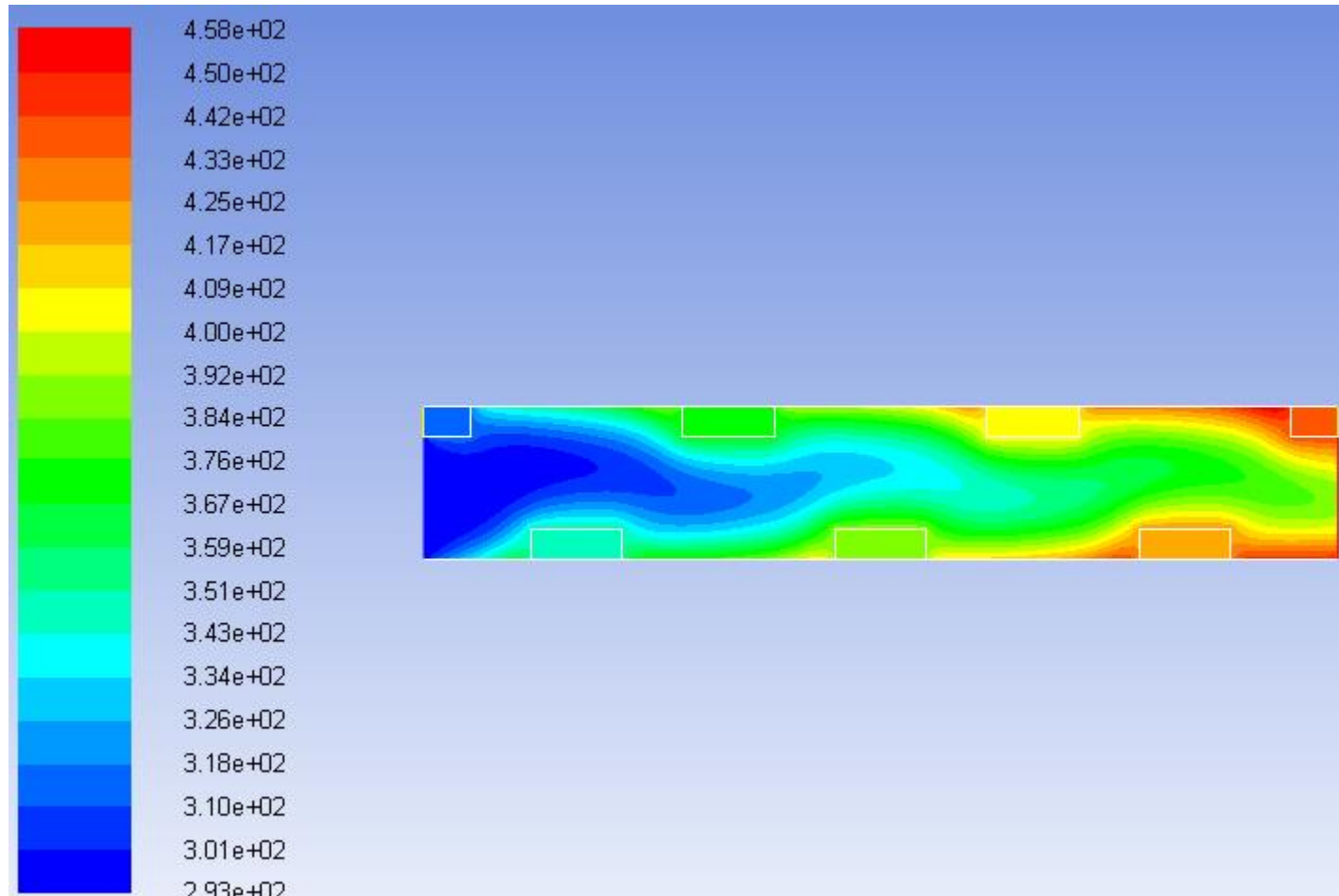
# Velocity magnitude



# 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](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....**