Numerical Heat Transfer

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

Instructor Wen-Quan Tao; Qinlong Ren; Li Chen

CFD-NHT-EHT Center
Key Laboratory of Thermo-Fluid Science & Engineering
Xi'an Jiaotong University
Xi'an, 2021-Dec.-28

数值传热学

第 13 章 求解流动换热问题的Fluent软件基础应用举例

主讲 陶文铨

辅讲 任秦龙，陈 黎

西安交通大学能源与动力工程学院

热流科学与工程教育部重点实验室

2021年12月28日，西安

第 13 章 求解流动换热问题的Fluent软件基础应用举例

第 13 章 求解流动换热问题的Fluent软件应用举例

13.1 散热装置导热问题

13.2 非稳态圆球冷却问题

导热问题

13.3 微通道内流动换热问题

13.4 芯片冷却流动换热问题

对流传热问题

13.5 太阳能光伏板水冷问题

13.6 肋片强化相变材料融化

相变传热

Review: The 10 steps for a Fluent simulation:

1. Read and check the mesh: mesh quality.
2. Scale domain: make sure the domain size is right.
3. Choose model: write down the corresponding governing equations 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.

13.2 Unsteady cooling process of a steel ball

非稳态圆球冷却问题

Focus: **compared with previous example, this example focuses on setting of 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.

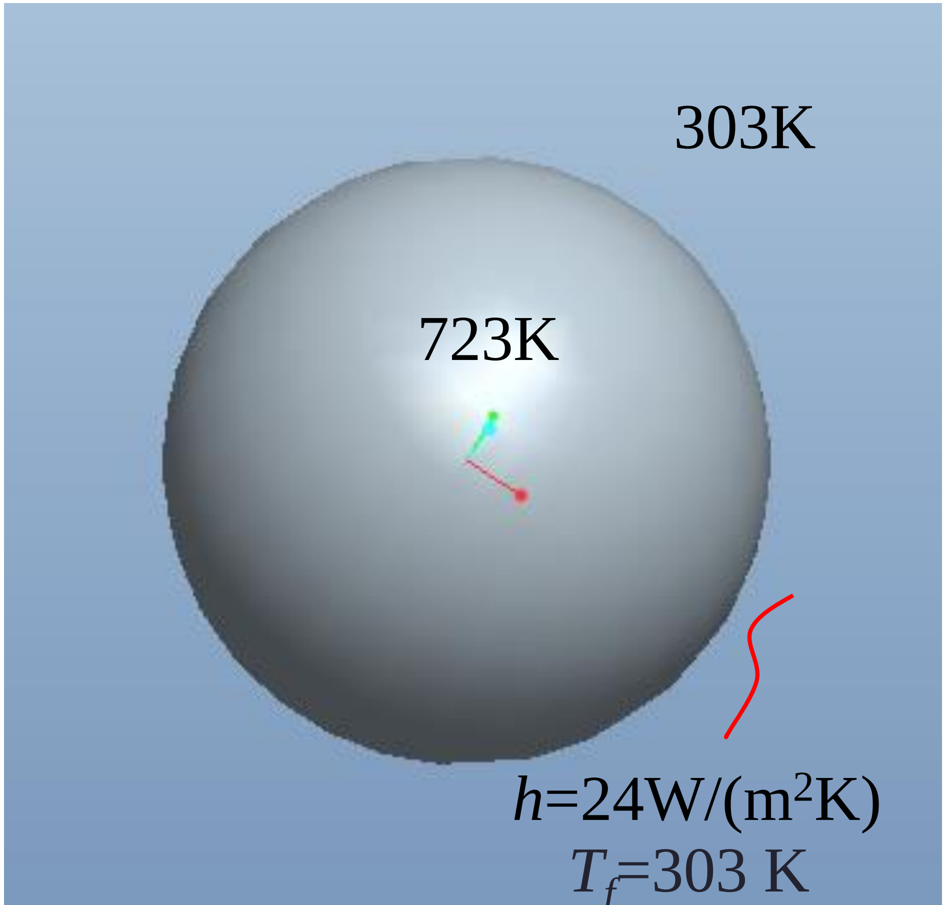
(D=5 cm, density is 7735kg/m^3, capacity is 480 J/(kg K), conductivity is 33W/(m K)).

- **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.

Fig.1 Computational domain

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

Solution:

Energy:

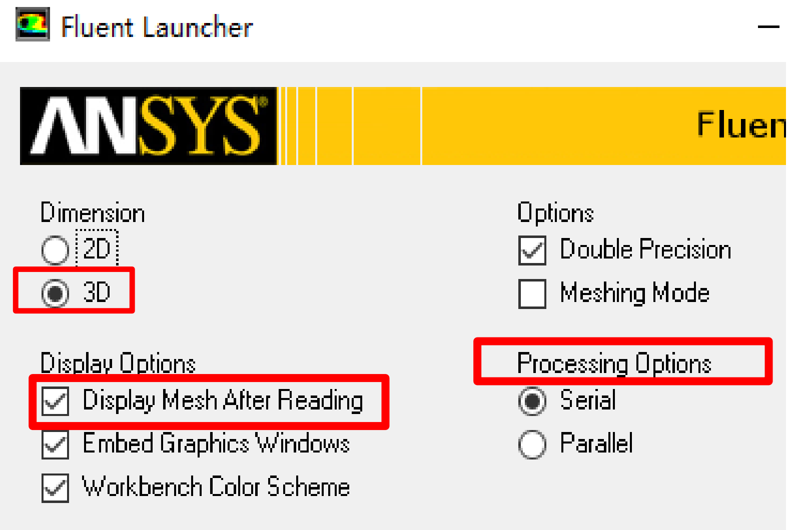
$$\frac{\partial\left(\rho C_p T\right)}{\partial t}=div(\Gamma_T gradT)$$

It is an 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.



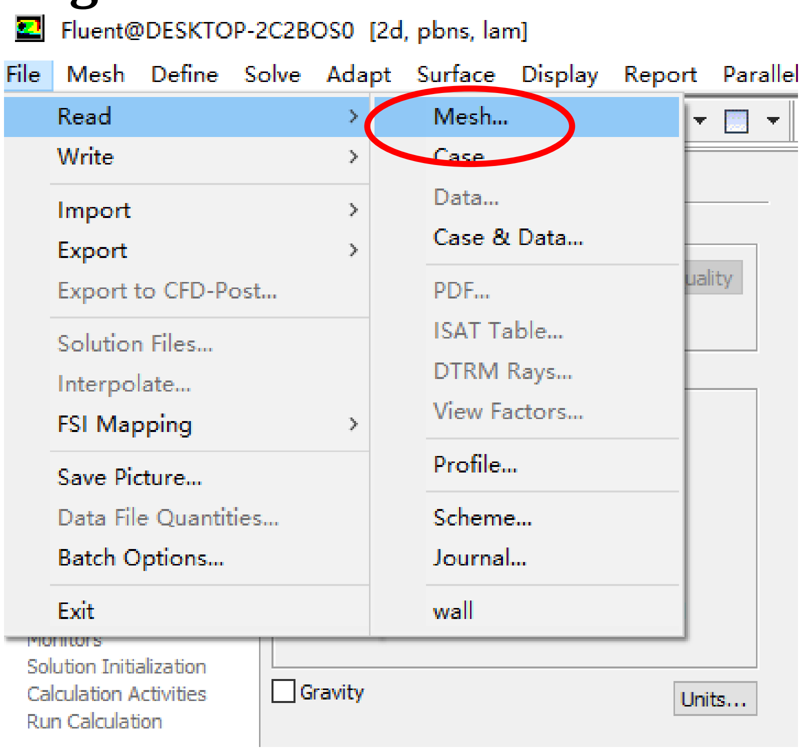
2、 If "display mesh after reading" is selected, after the Fluent is launched, the mesh will automatically shown in the interface.

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

Step 1: Read and check the mesh

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

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



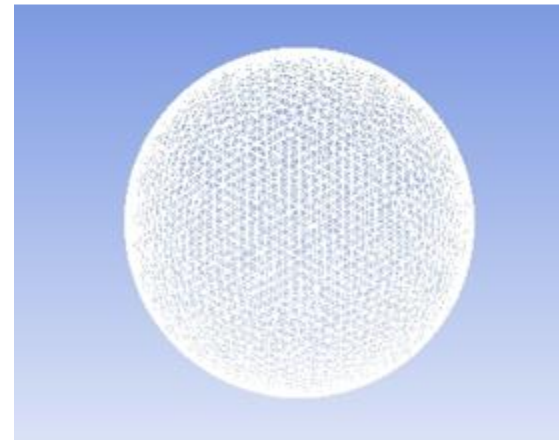
Mesh→Read

```
> 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
```

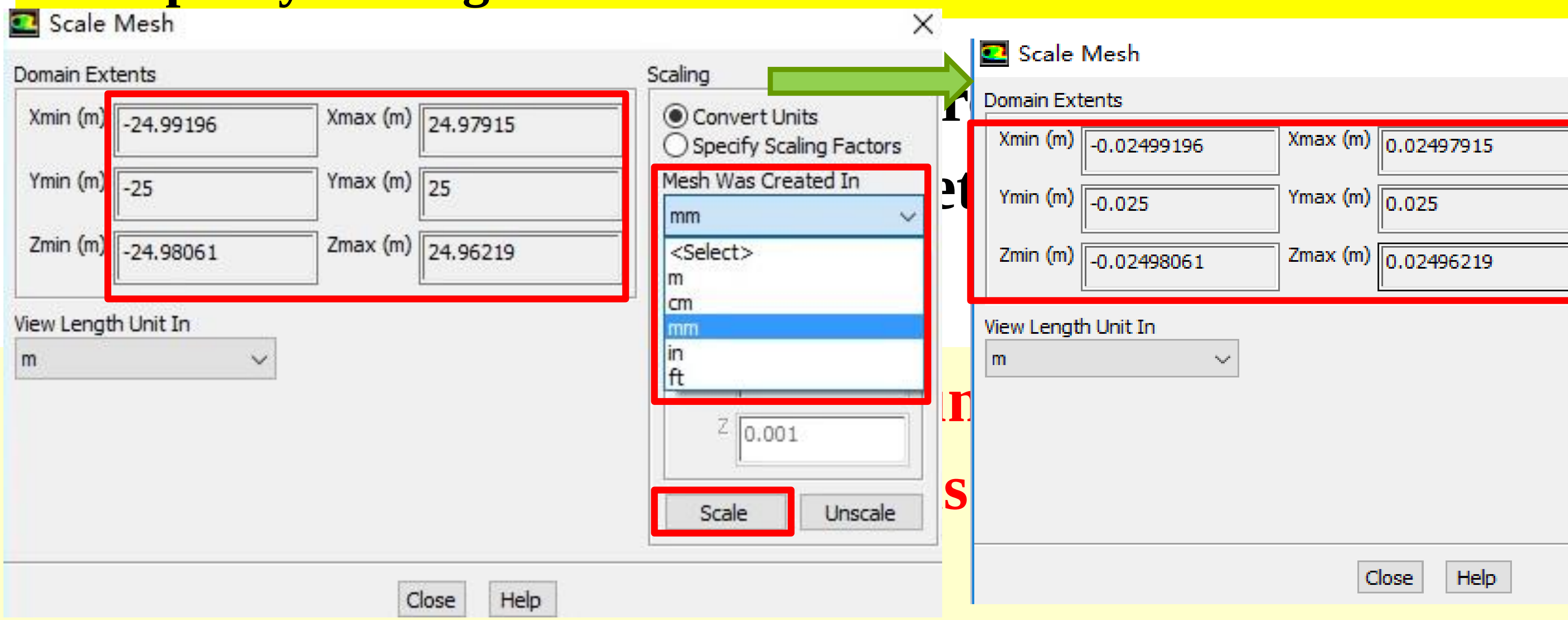
```
WARNING: Mesh check failed.
```

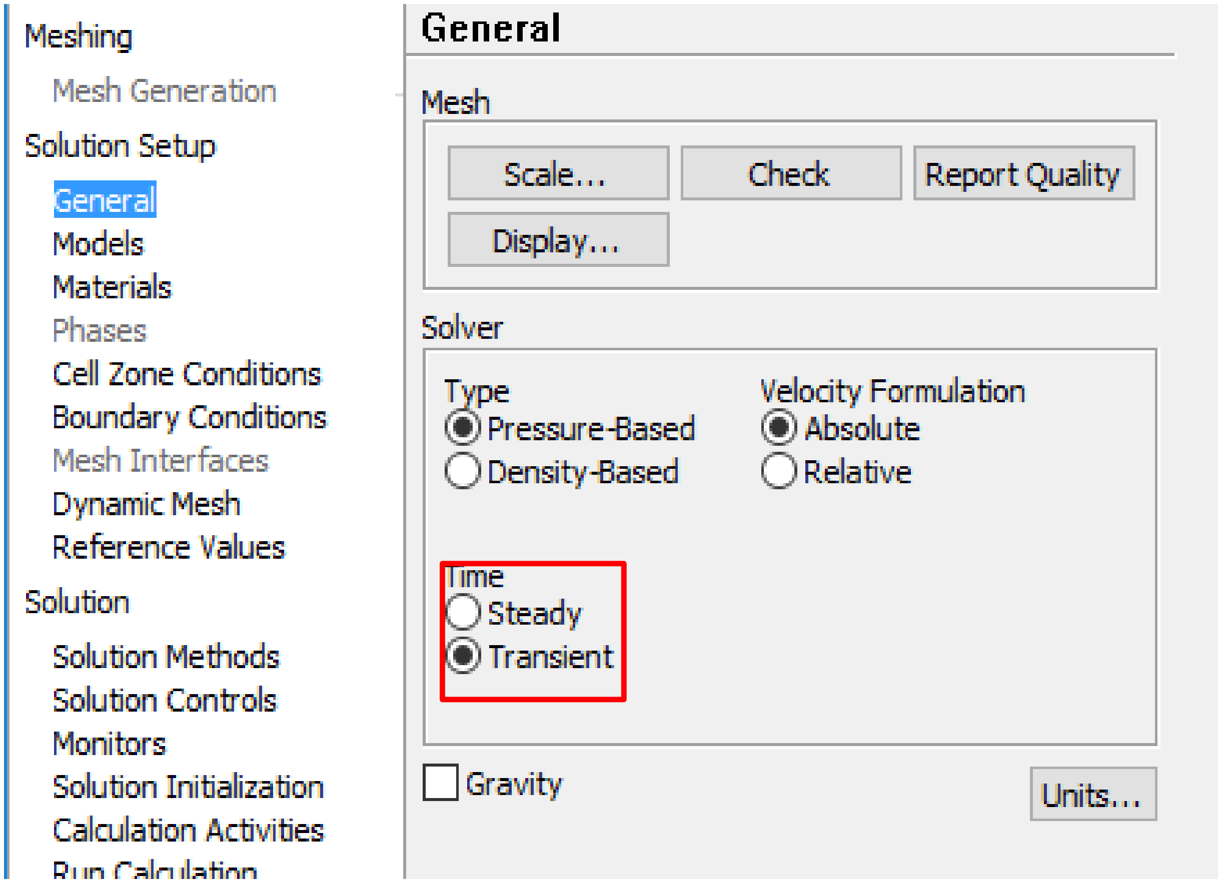
```
WARNING: The mesh contains high aspect ratio quadrilateral,
hexahedral, or polyhedral cells.
```

2st step: Scale the domain size

General→Scale

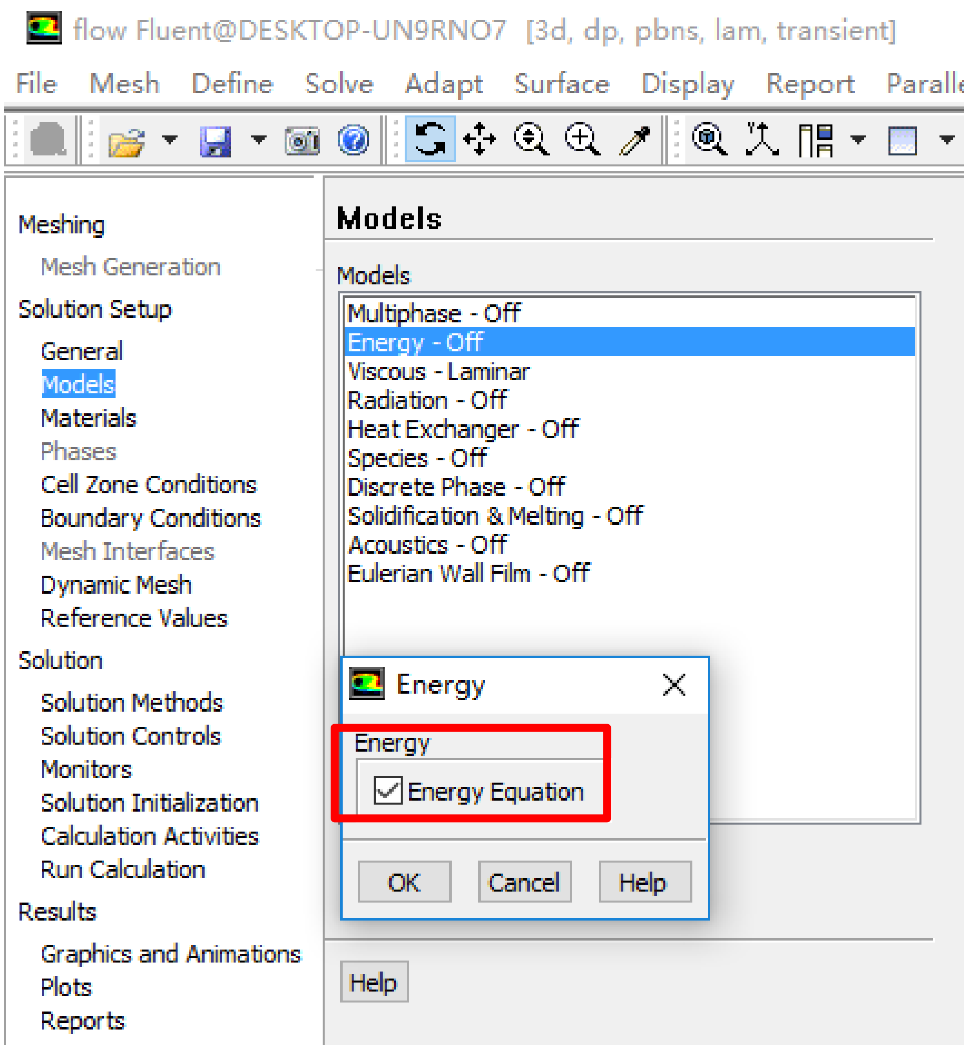
- **You also can scale the domain size use "Convert Units" or " Specify Scaling Factors" command.**





Choose the "transient" for a unsteady problem!

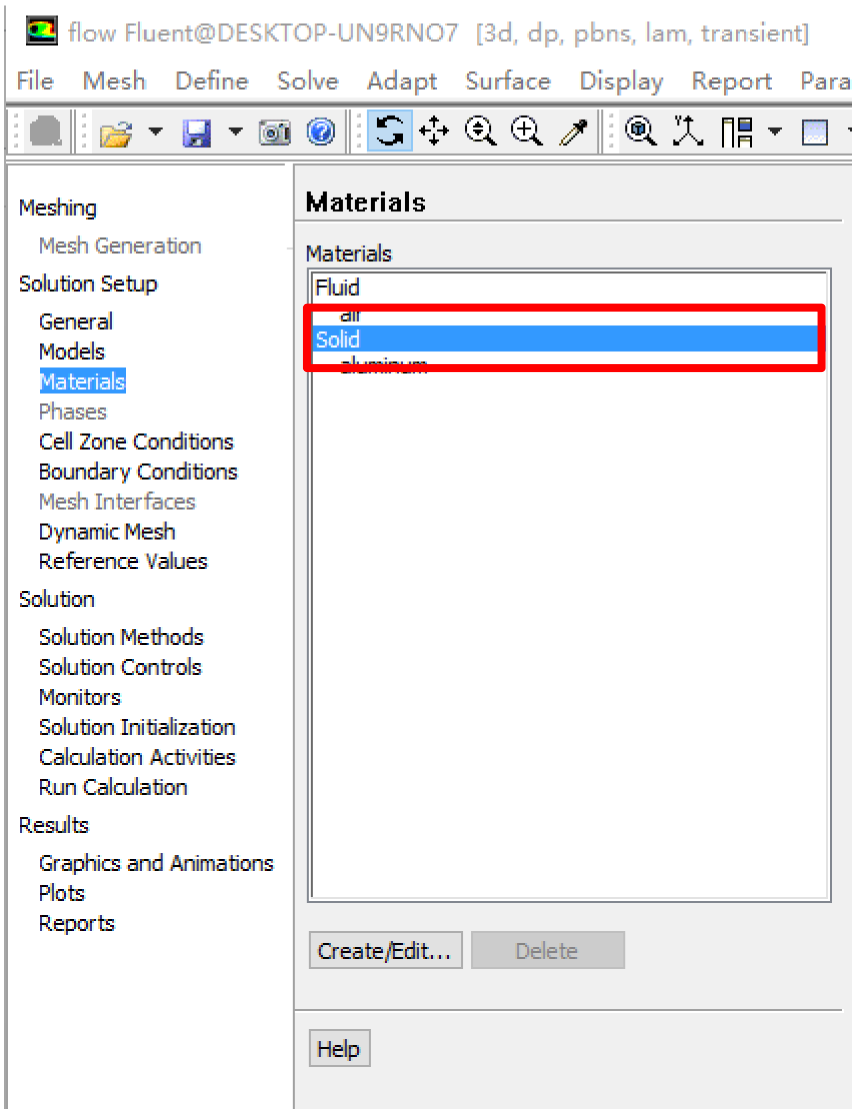
Step 3: Choose the physicochemical model



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

The energy equation is activated.

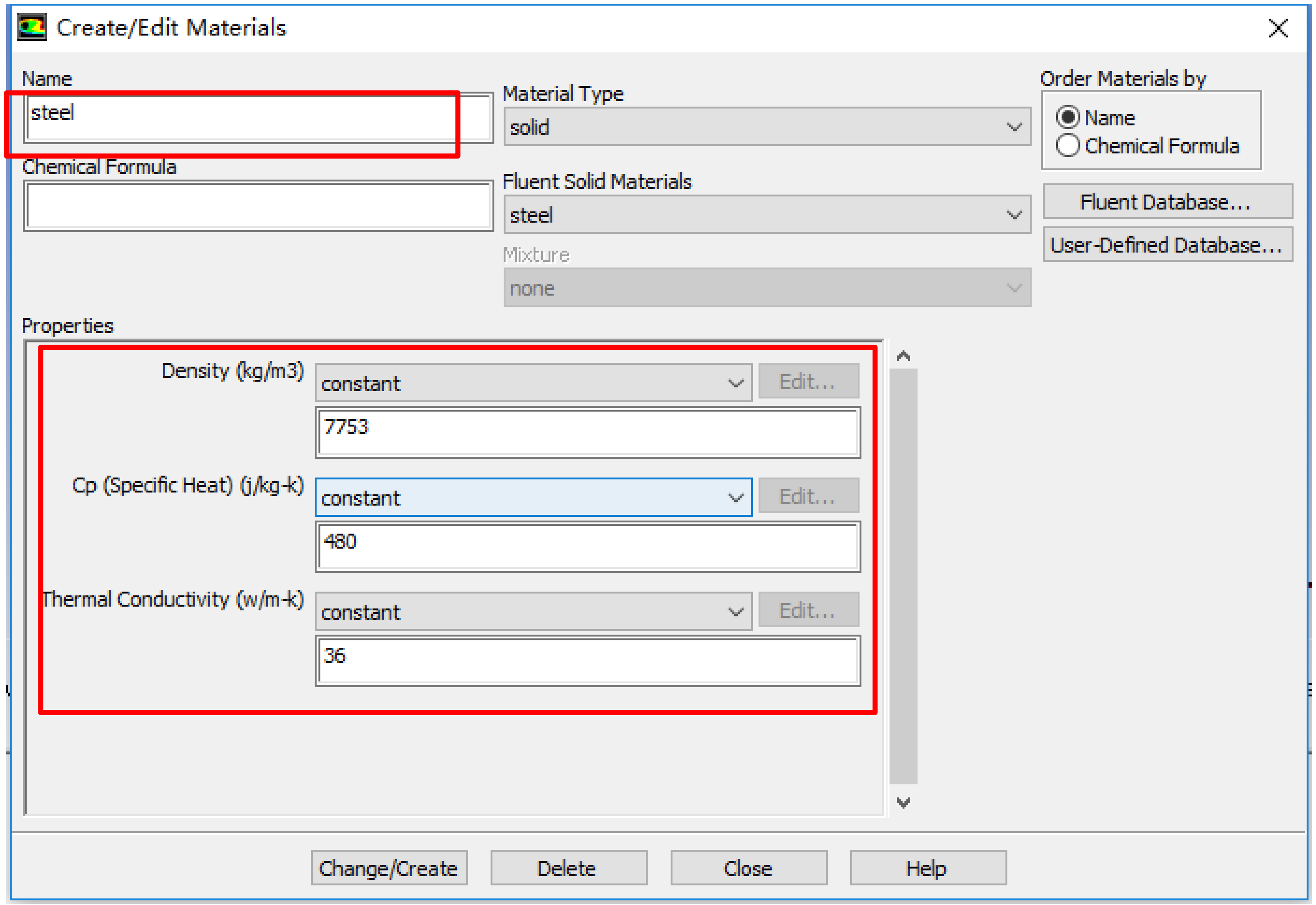
Step 4: Define the material properties



The default fluid in Fluent is air.

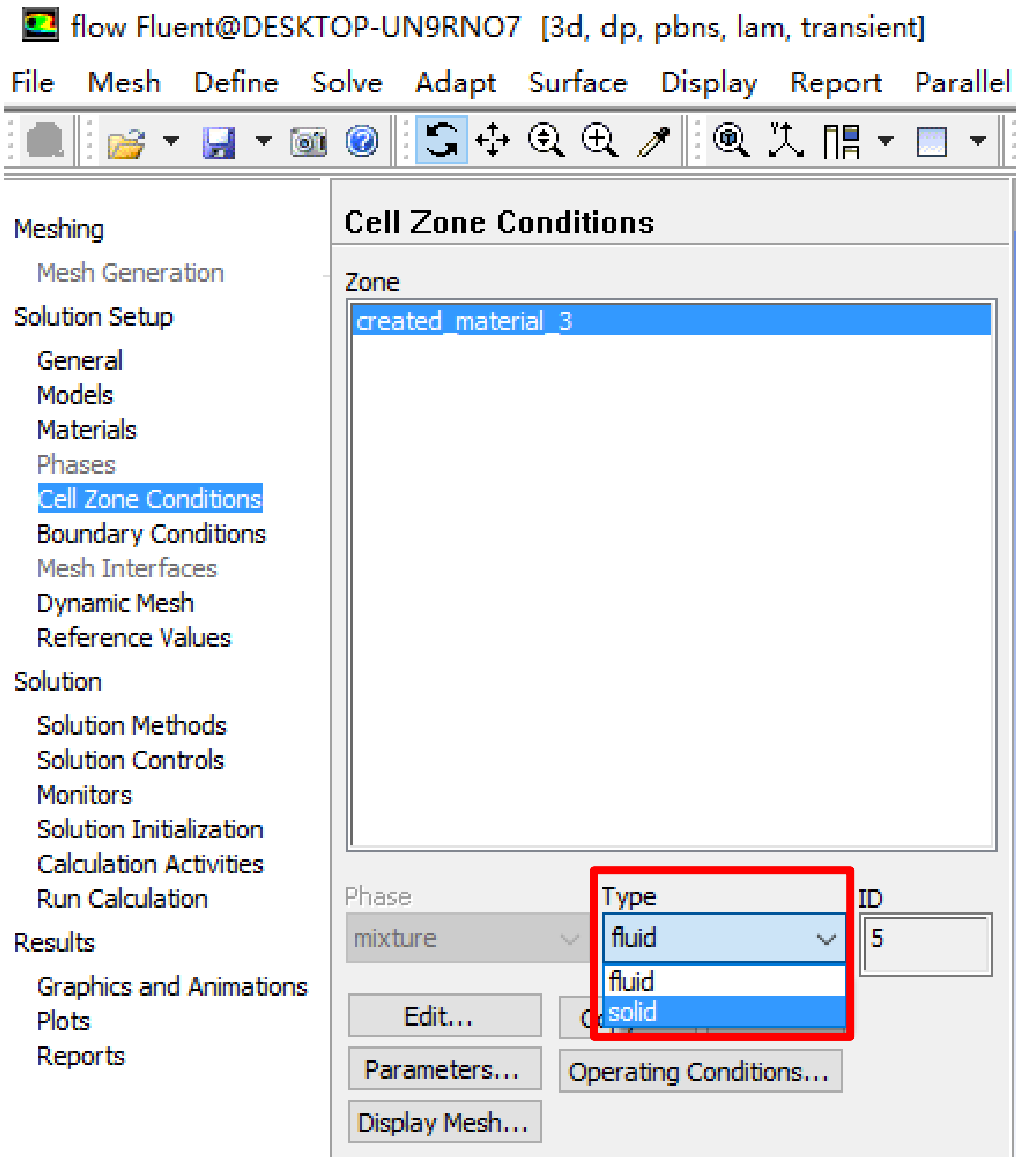
The default solid in Fluent is Aluminum.

For Example 2, steel material should be added.

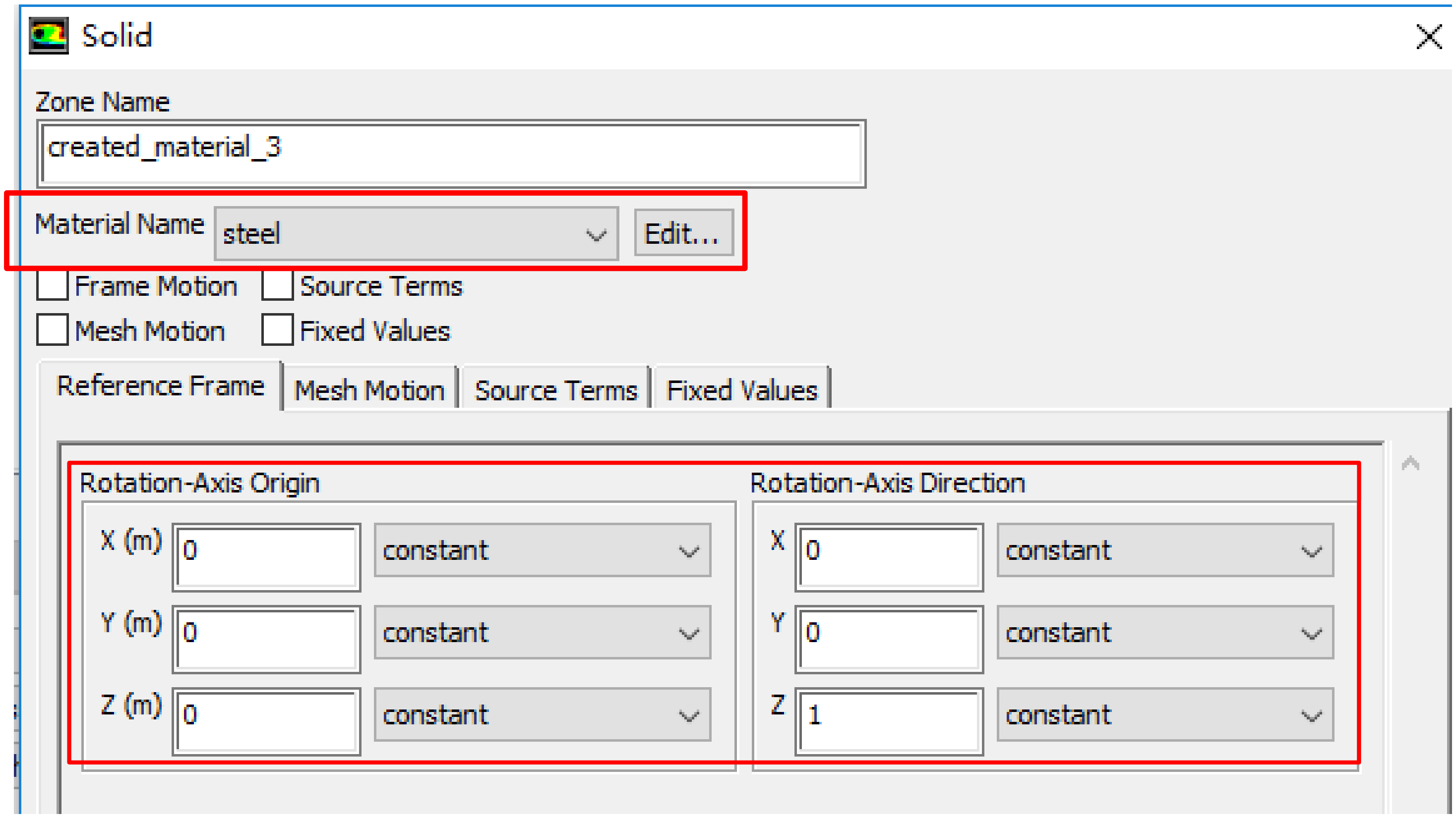


The properties of steel are manually inputted.

Step 5: Define zone condition

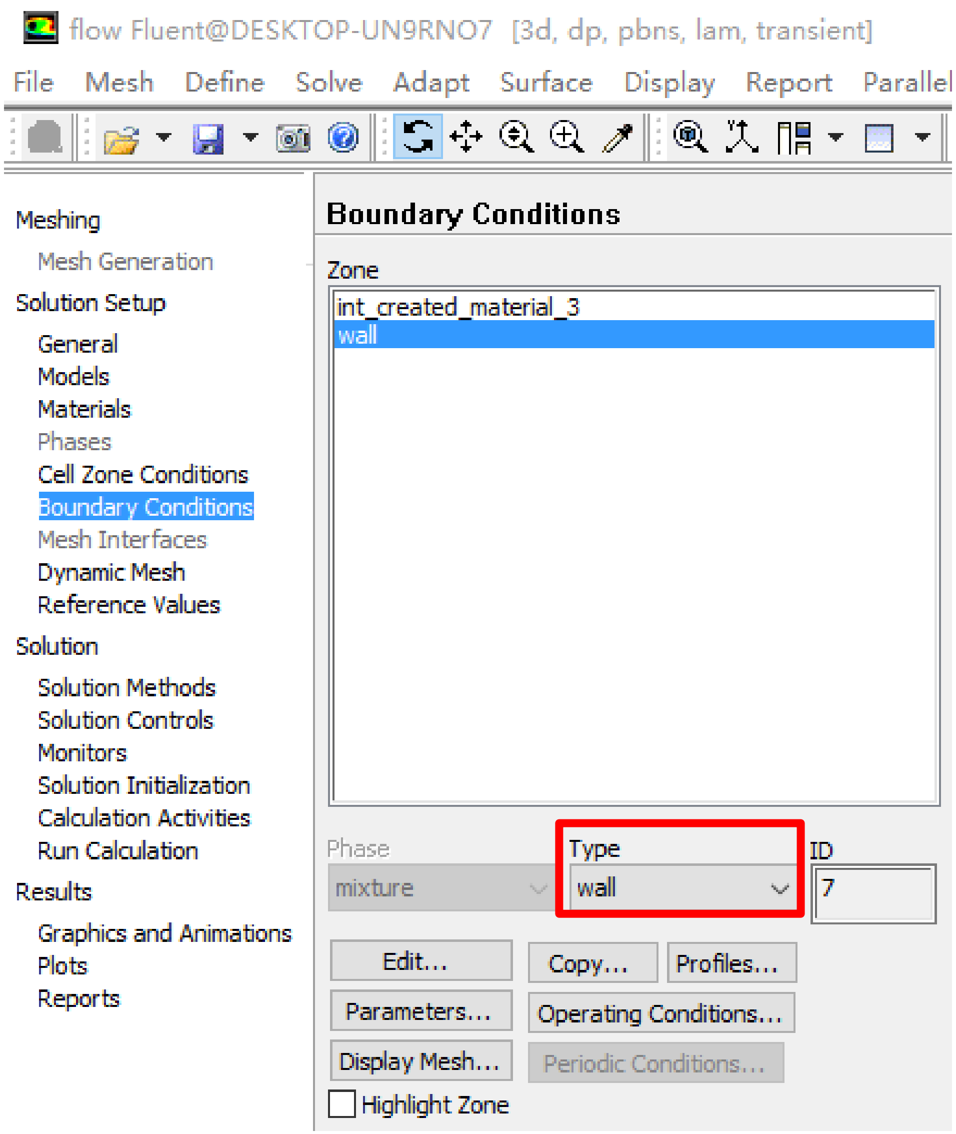


In this step, we define the cell zone conditions. The cell zone is a ball made of steel, so you should choose the type "solid".

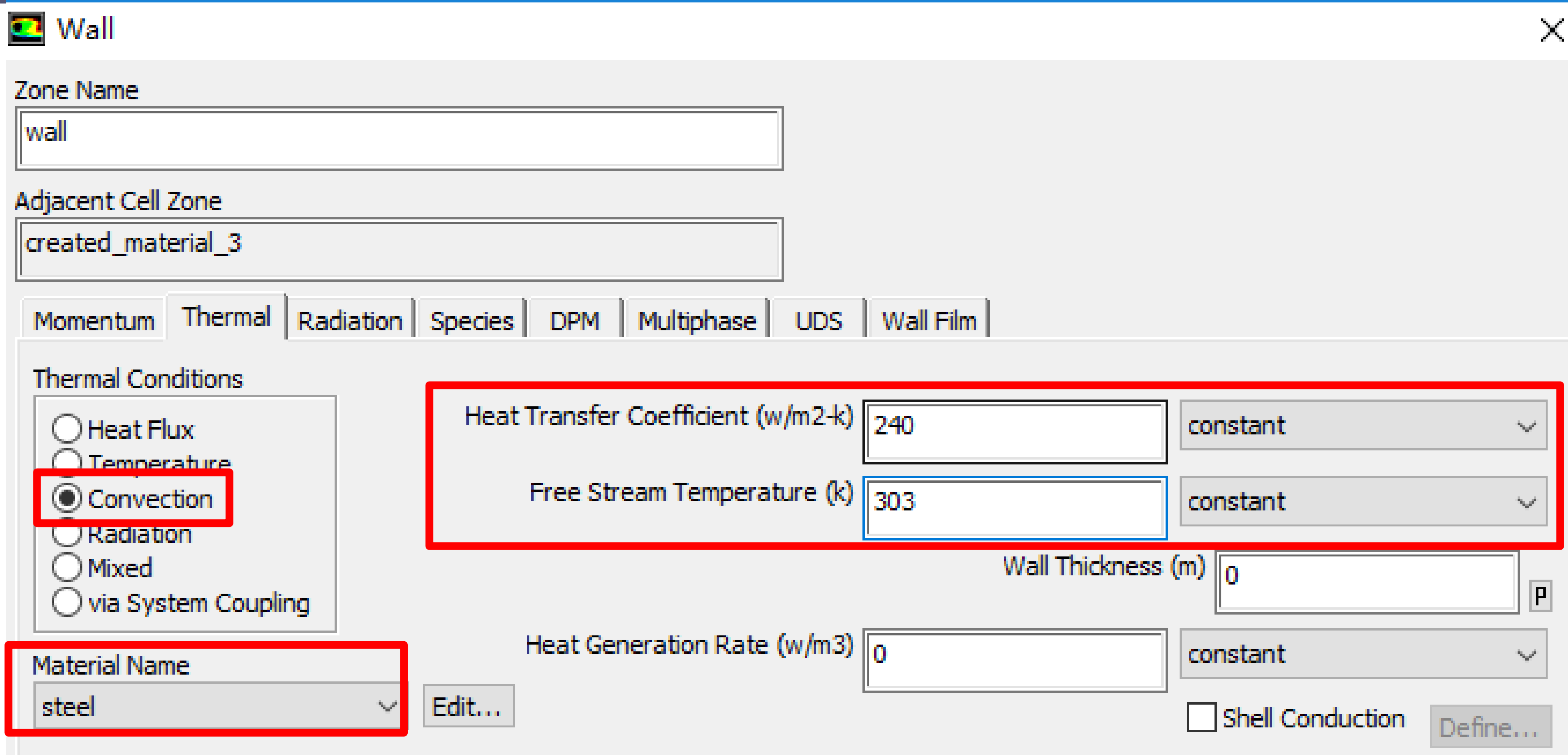


Be sure the material is steel and others keep as default.

Step 6: Define the boundary condition

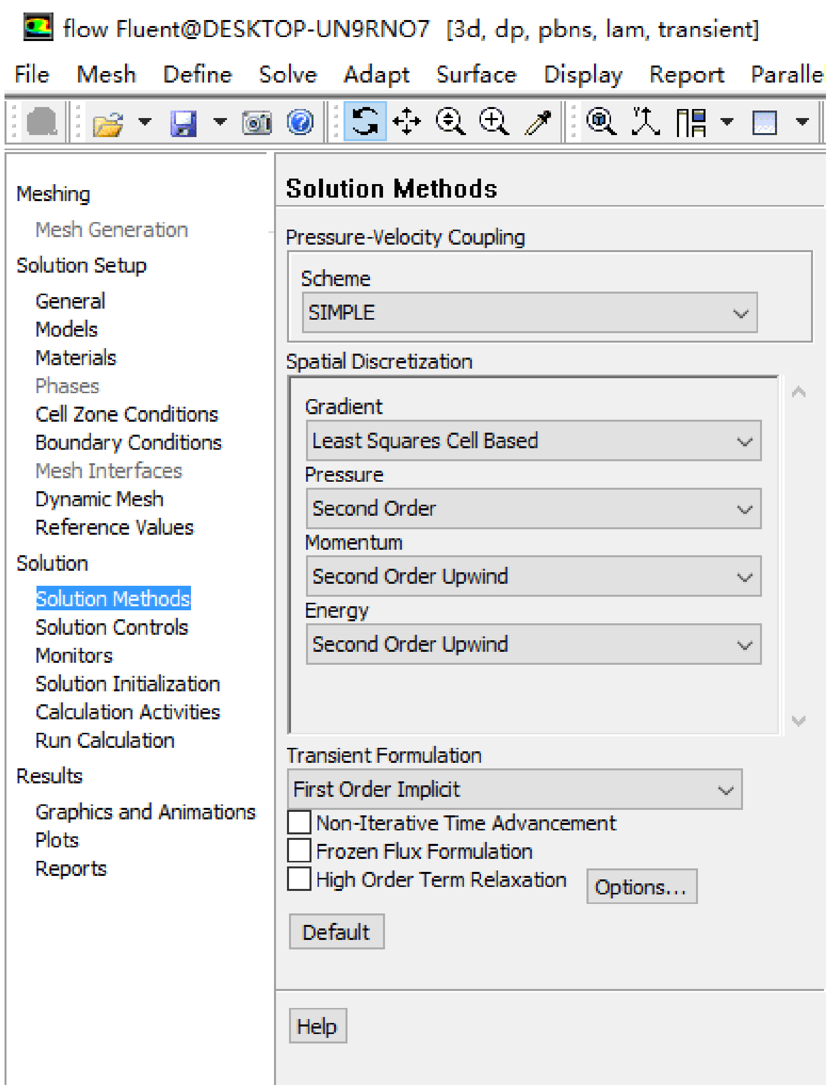


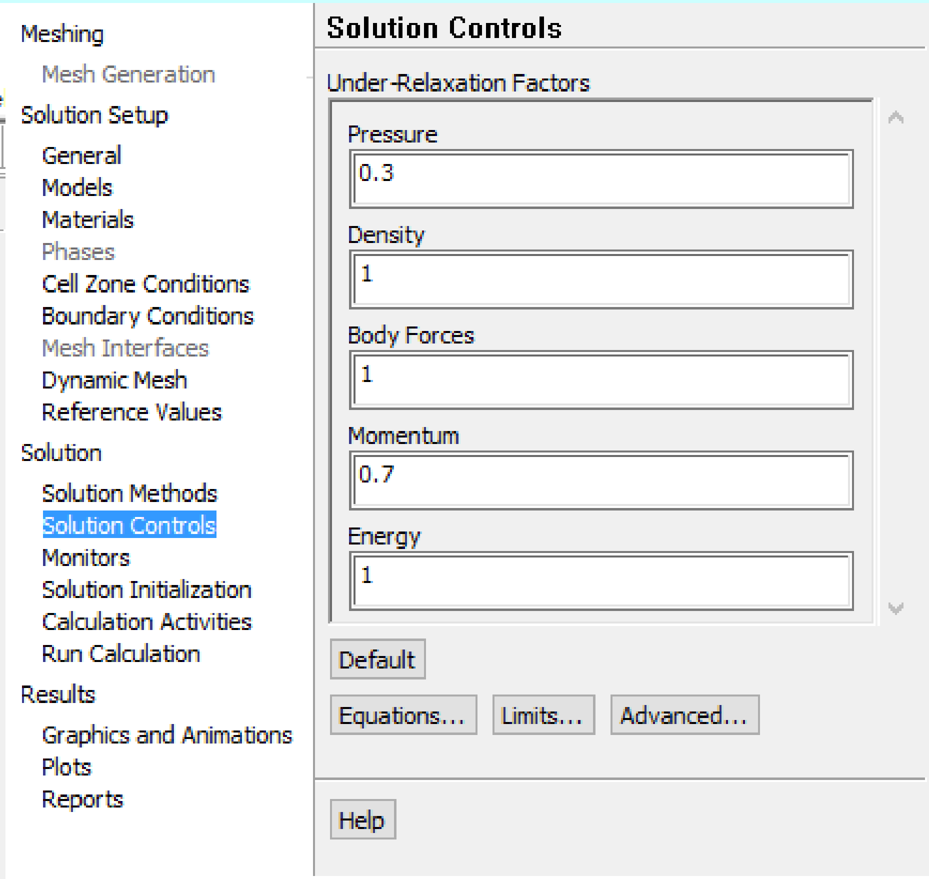
Now, you need to define the "Boundary conditions" Firstly, ensure the "type" is "wall". Then click the "edit" to edit the BC.



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

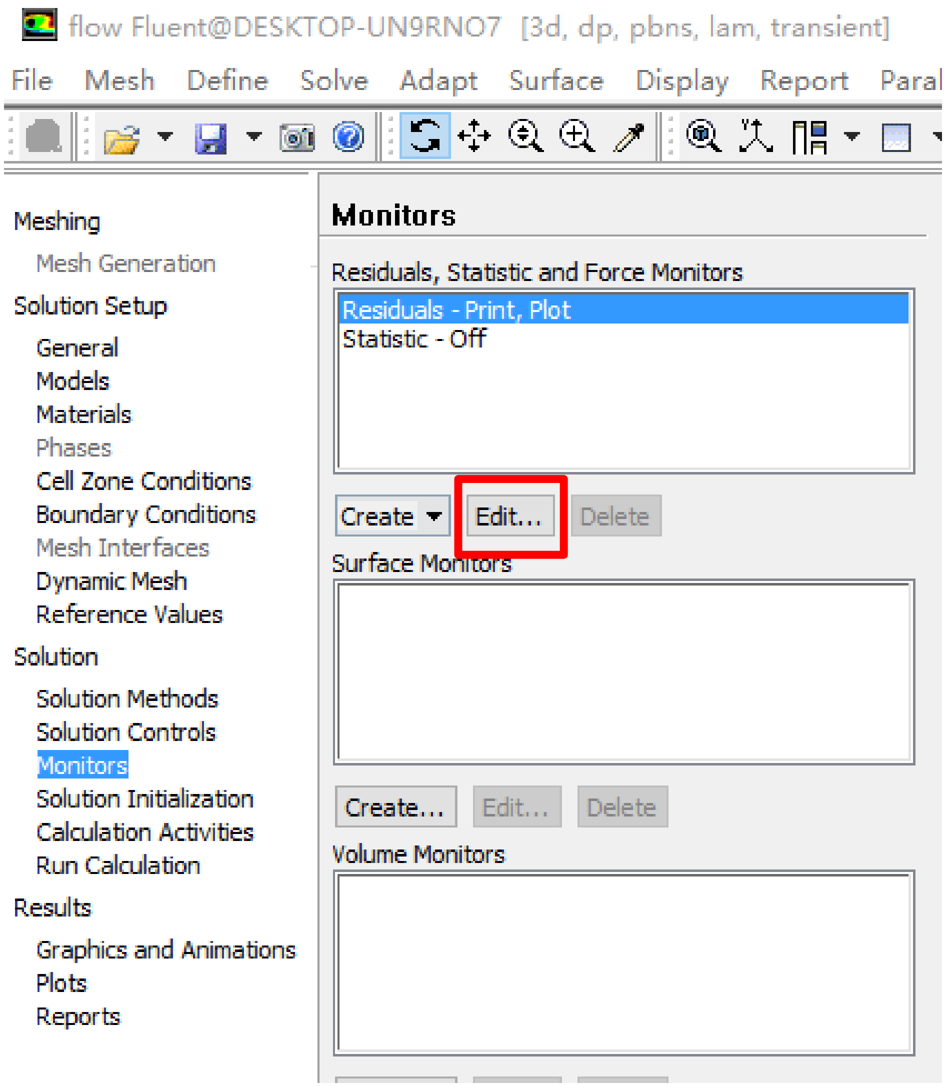
Step 7: Solution setup: algorithm and scheme





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

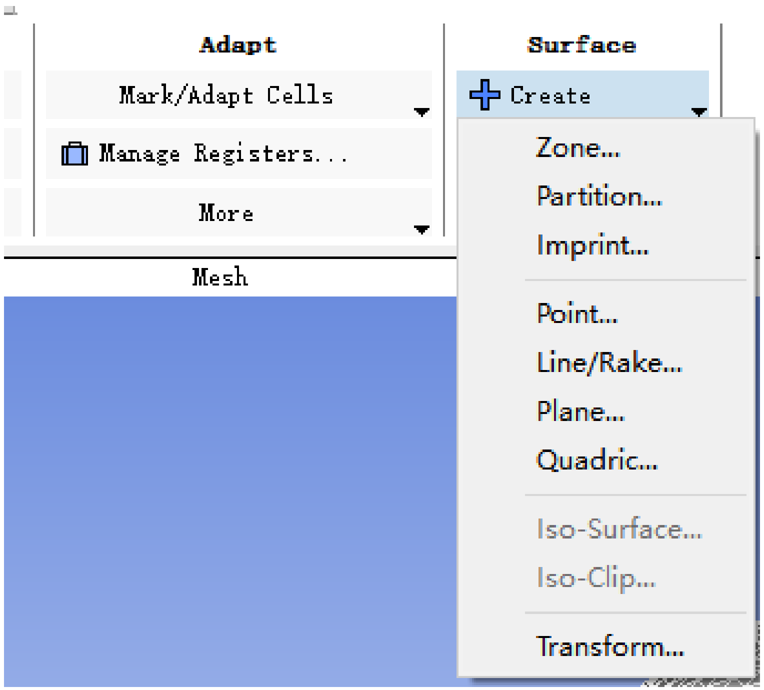
Step 7: Solution setup: monitors

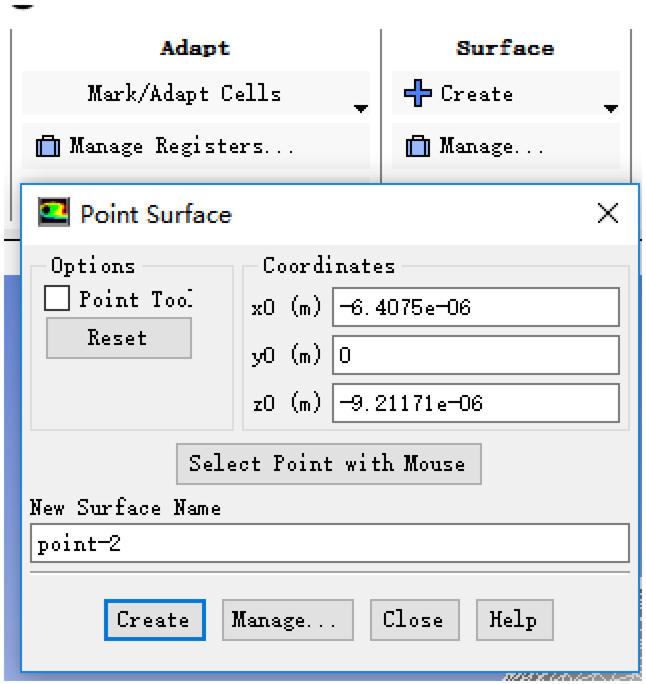


In this step, the residual can be changed.

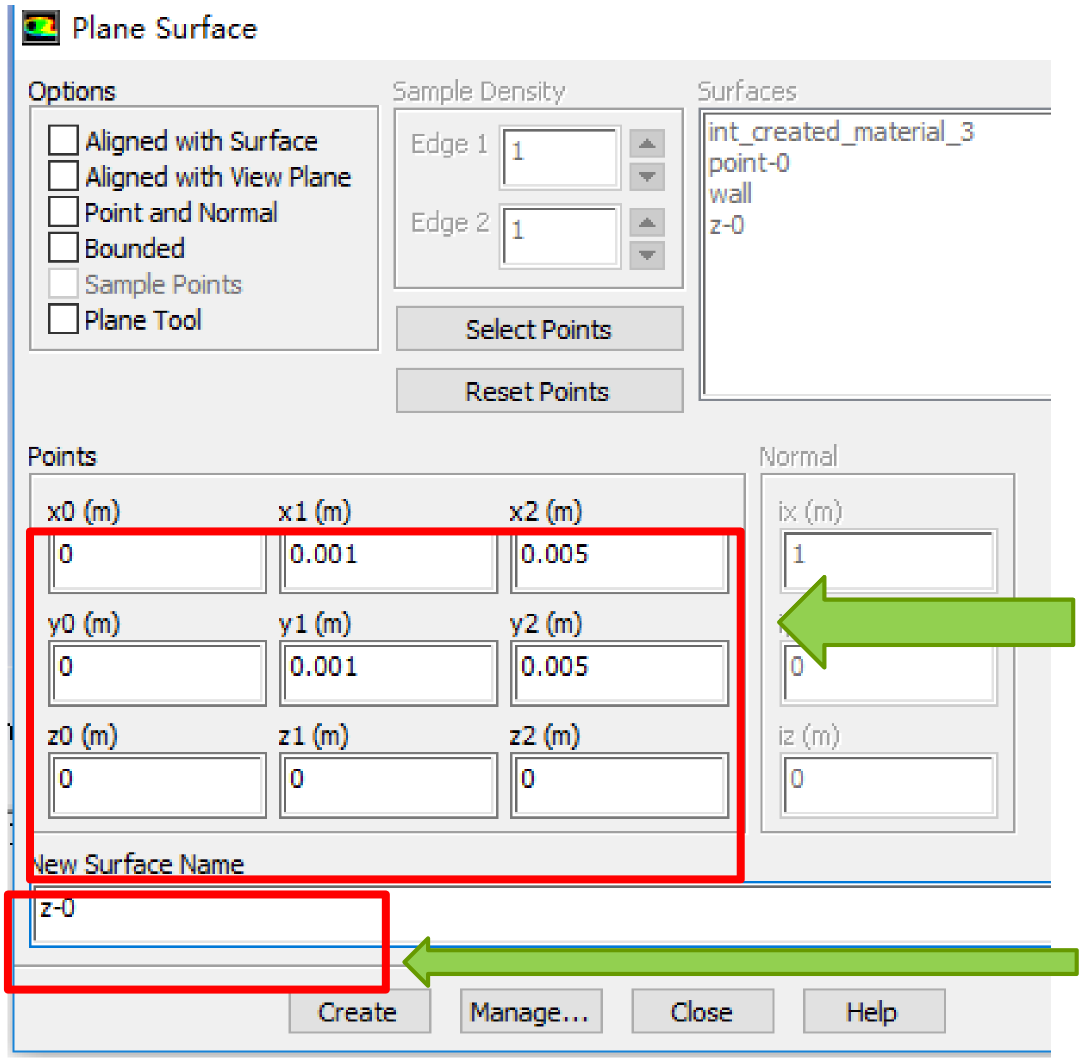
You also can define a point, a line or a surface to monitor related variables.

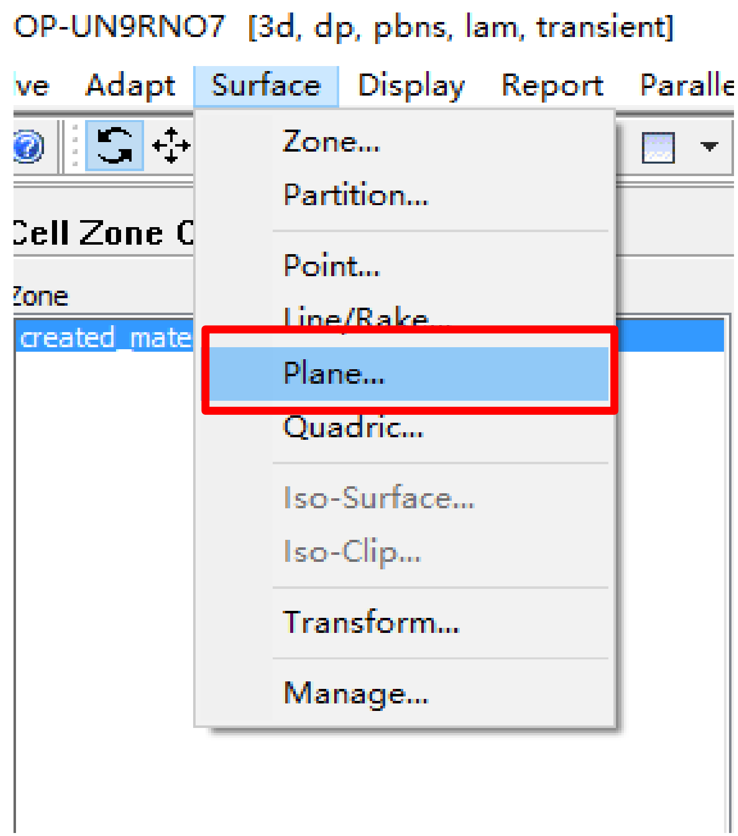
Here, you can create a point by clicking "surface" and choose "point", the "point" dialog will display.

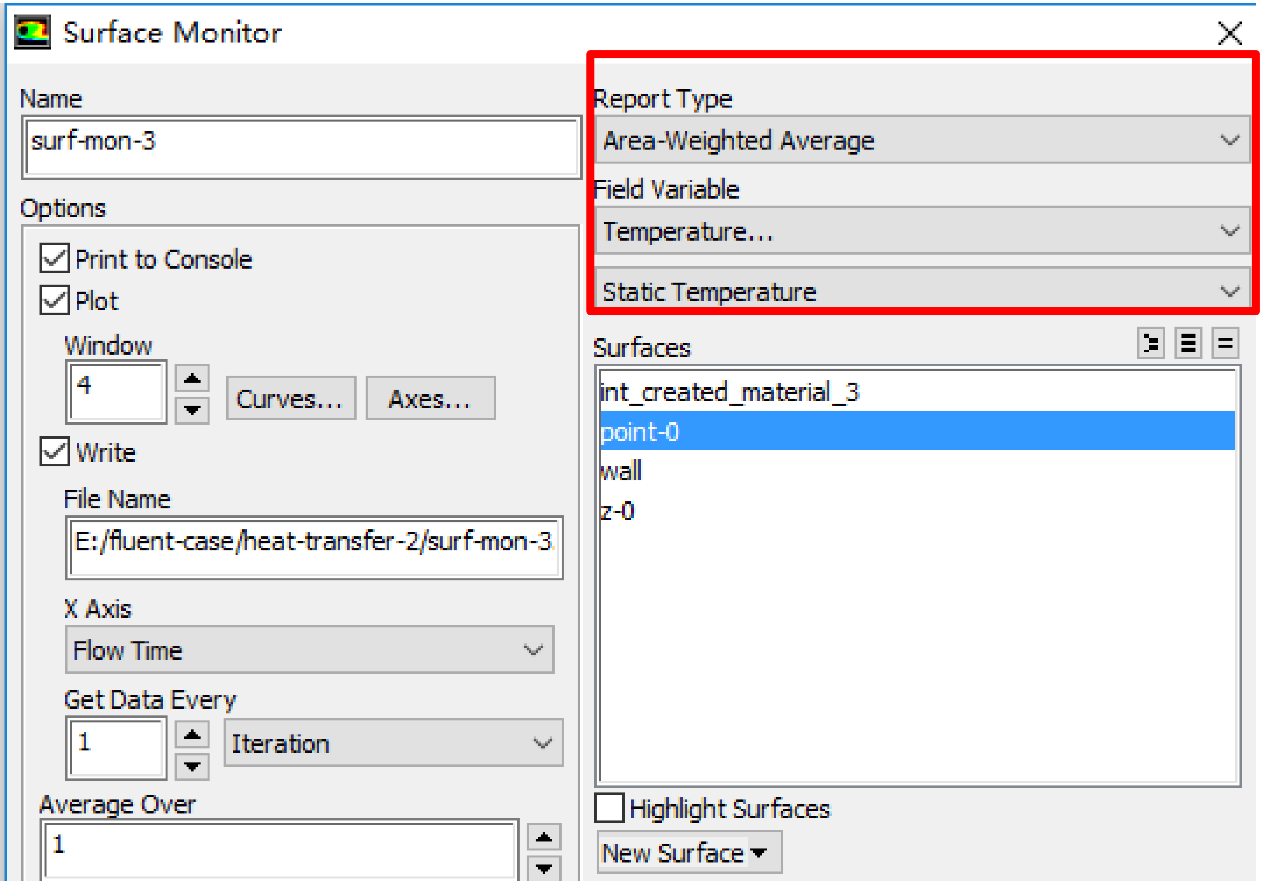




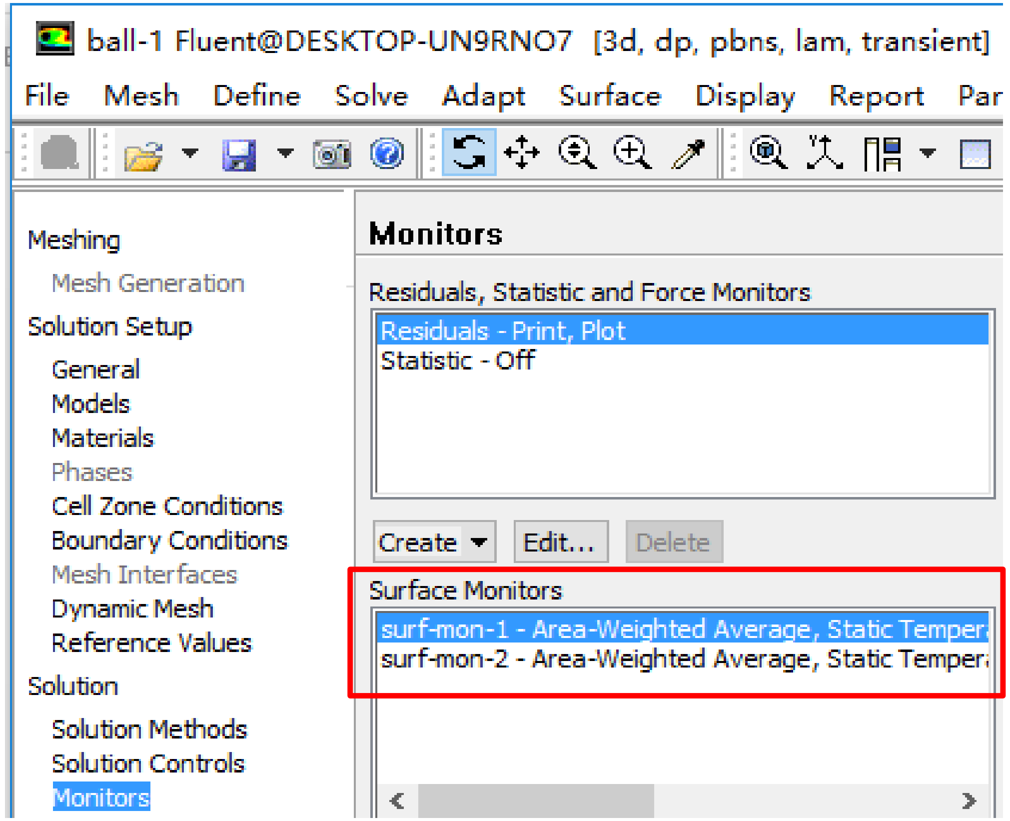
You can also create Plane by defining three points in the surface.







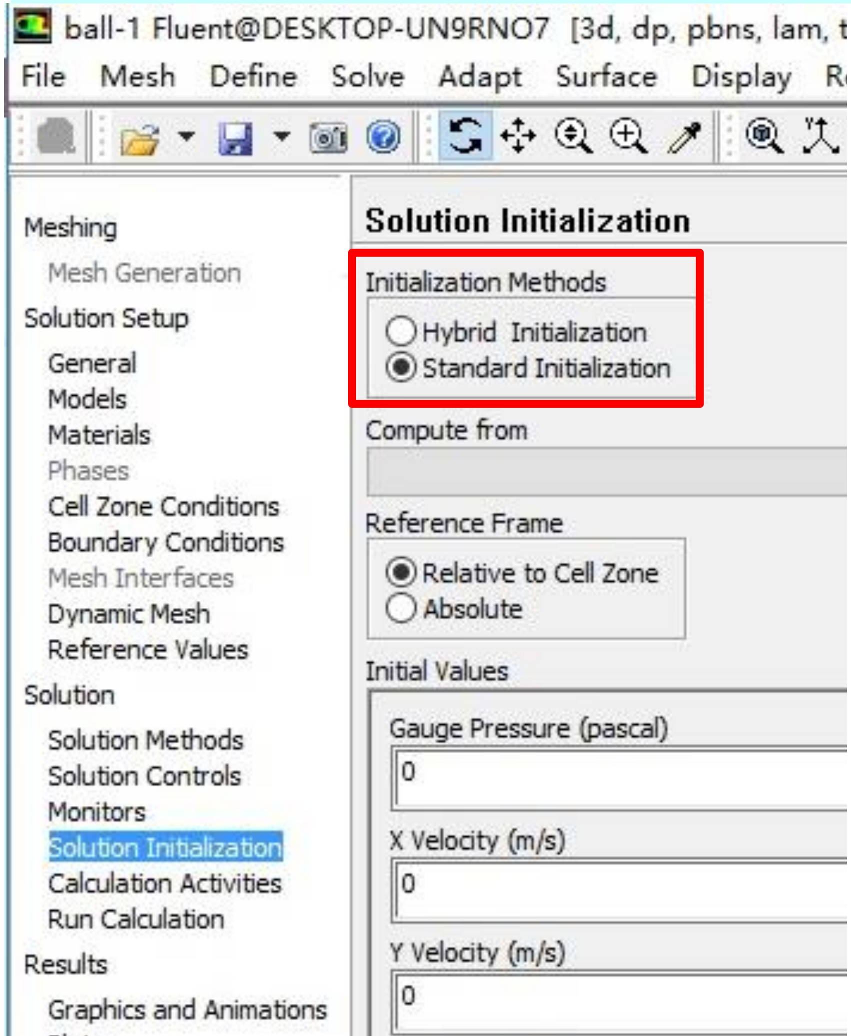
Next, you can create the monitors in the "Monitors" dialog. Select the "Report type", the variable you want to monitor, and the position you want to monitor.



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



The "Standard Initialization" use the initial value of one curve so the initialization is quick, but the speed of convergence is slow.

The "Hybrid Initialization" is opposite, it initializes slow but the speed of convergence is fast.

Select "Standard Initialization" and "Compute from" "all zones".

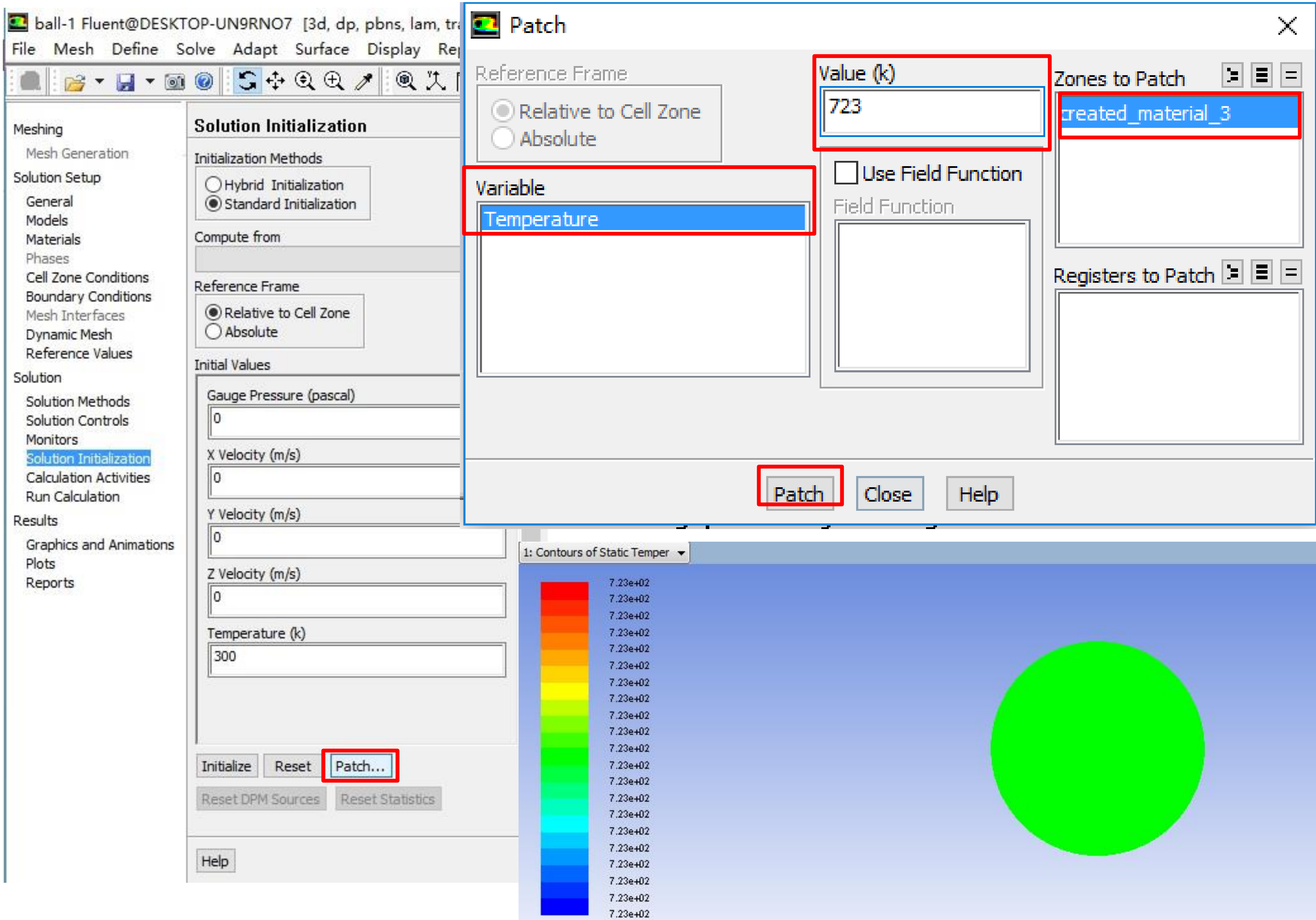
Patching (修补）Values in Selected Cells

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

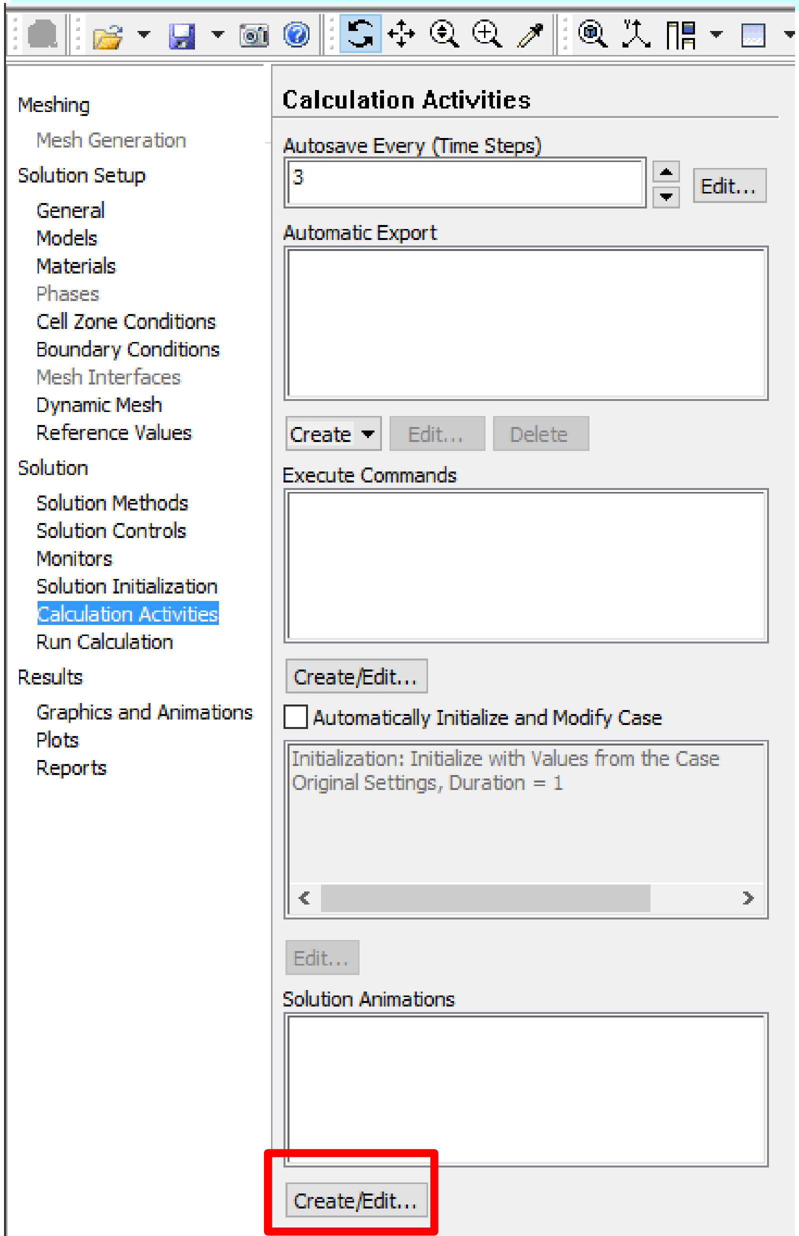
For multiphase flow, you may also want to define the volume of fraction for a phase in a particular sub-region.

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.

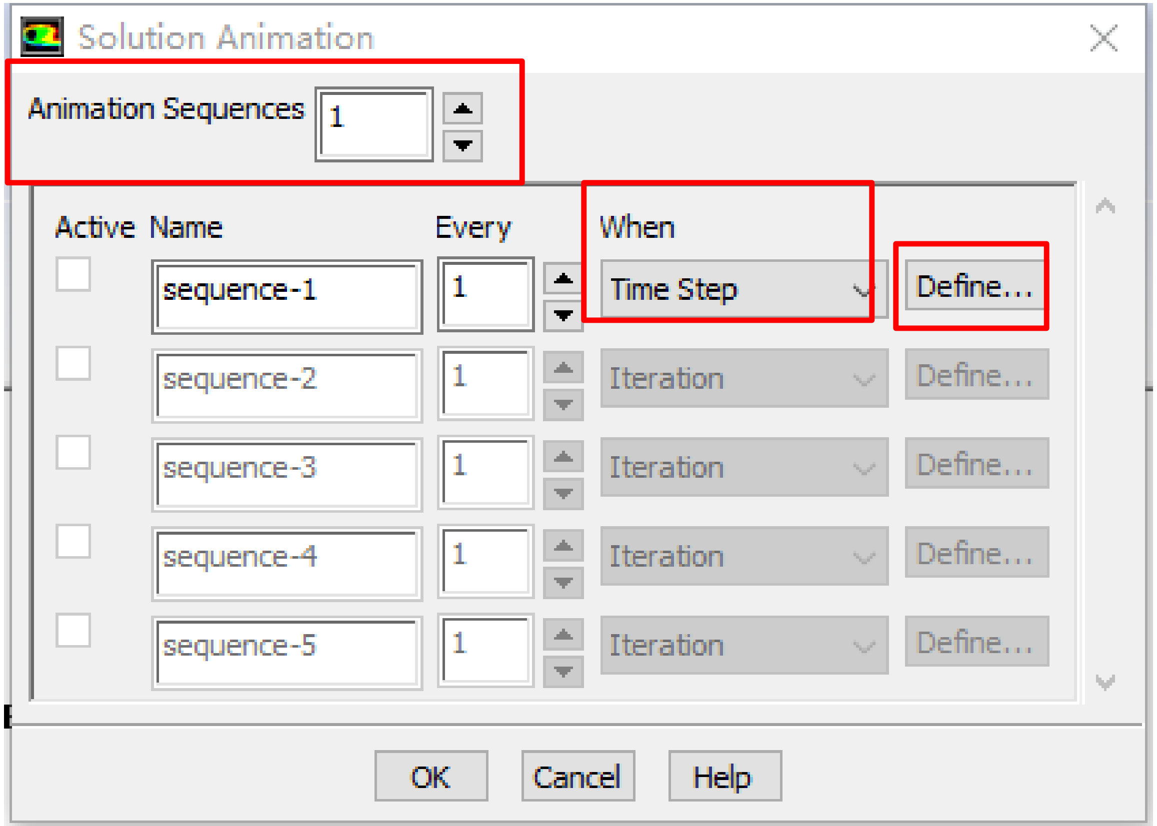

ball-1 Fluent@DESKTOP-UN9RNO7 [3d, dp, pbns, lam, tr
File Mesh Define Solve Adapt Surface Display Re
Meshing
Mesh Generation
Solution Setup
General
Models
Materials
Phases
Cell Zone Conditions
Boundary Conditions
Mesh Interfaces
Dynamic Mesh
Reference Values
Solution
Solution Methods
Solution Controls
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.

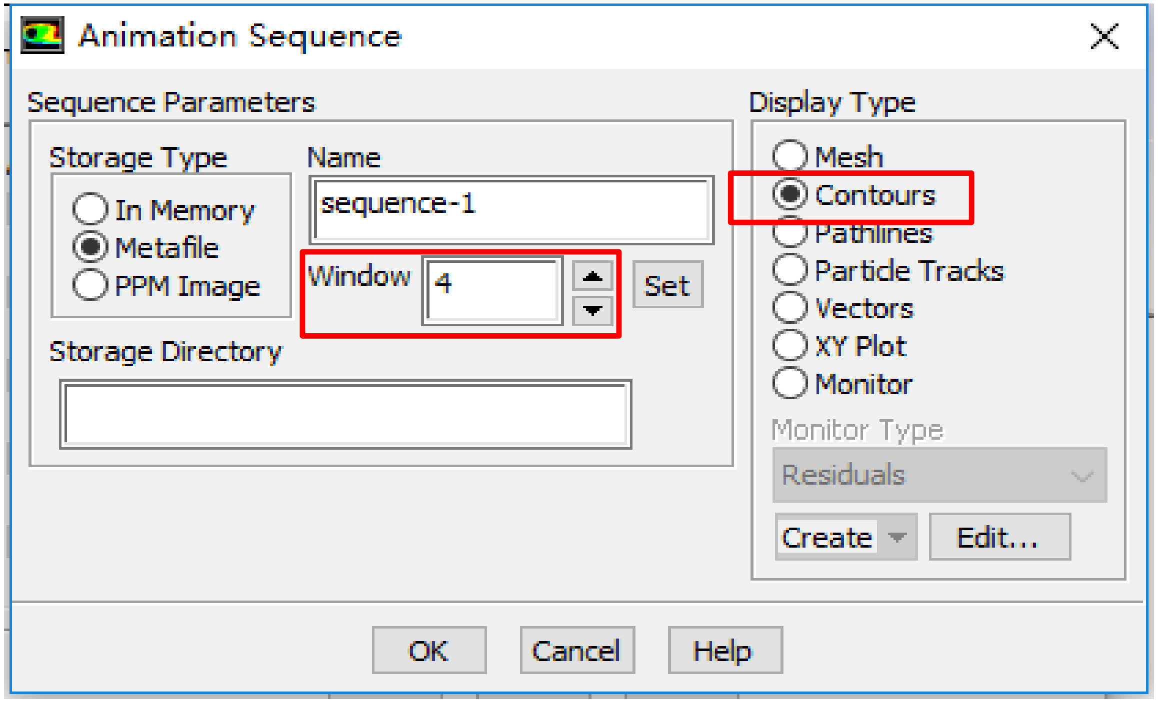
In the "Calculation Activities" dialog, click "Change/Create" in "Solution Animations".



Set the "Animation Sequences" as 1.

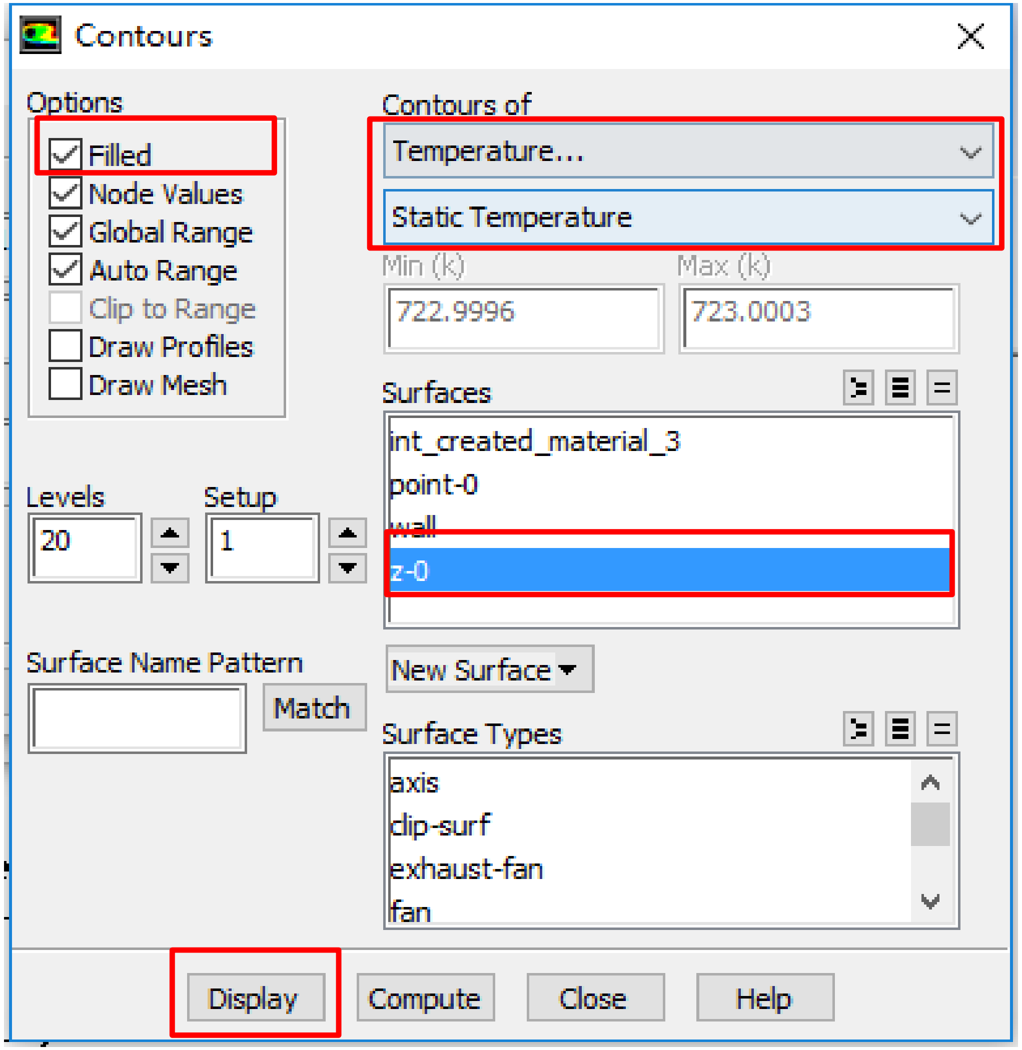
Select "Time Step" in "When".

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.

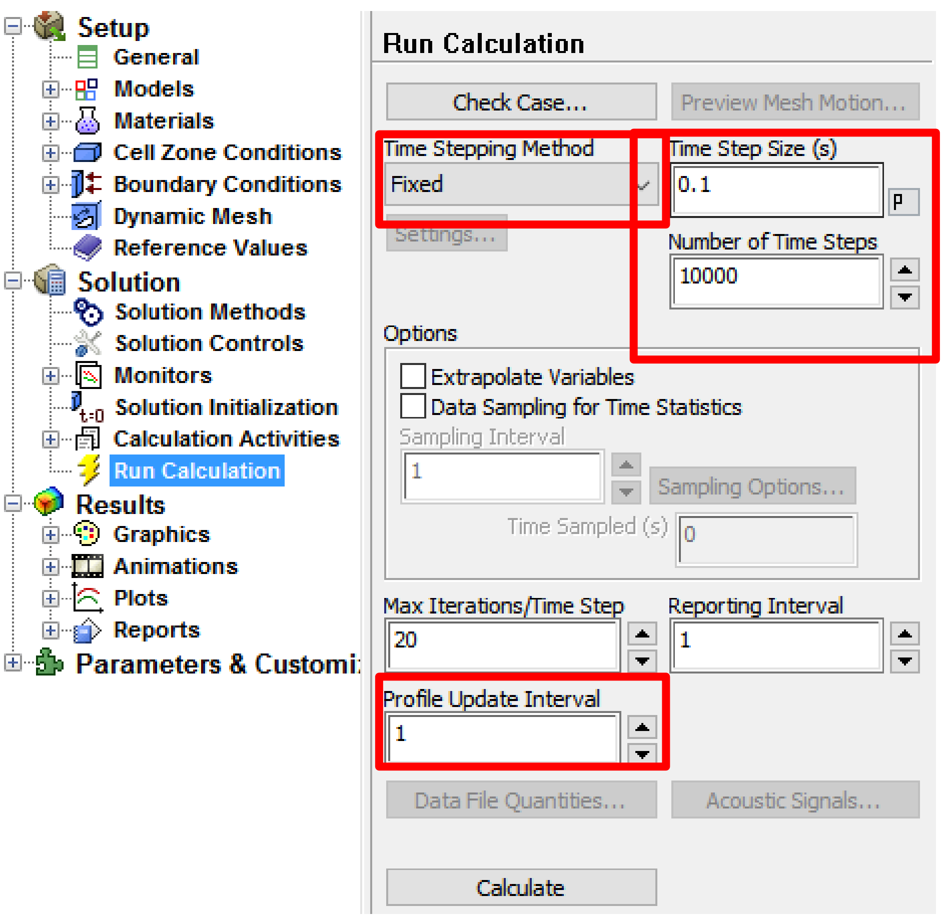


In "Contours" dialog, we choose "Temperature", select "Filled", and choose the surface: z-0.

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

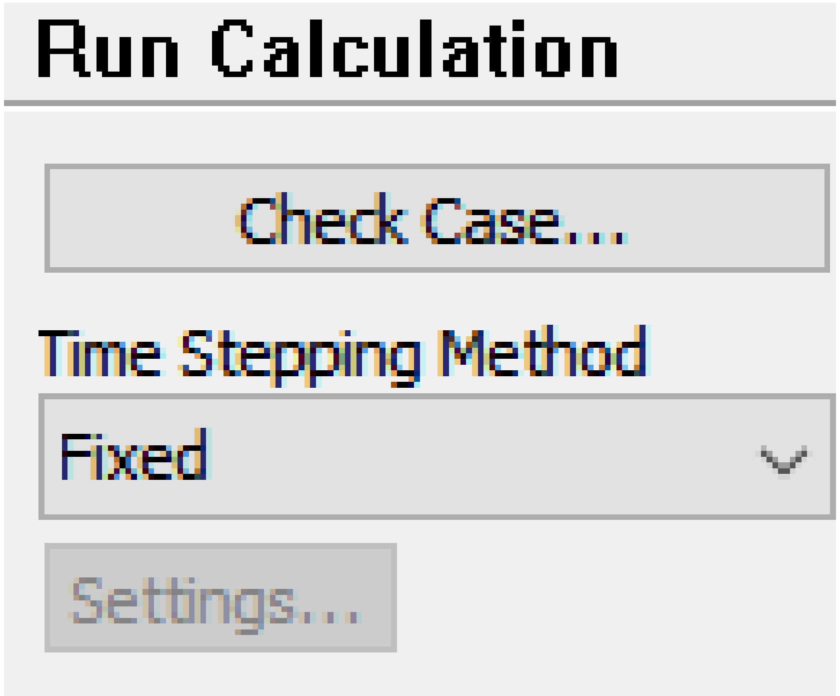
Step 9: Run the simulation

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

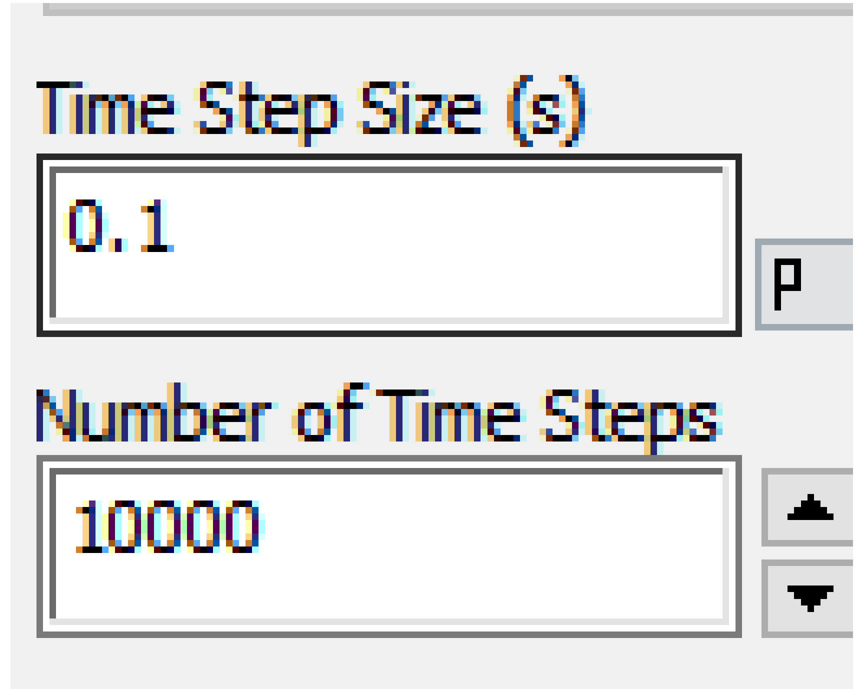


You need to select the time stepping method, set the time step size, and the max iteration per time step.

Time stepping method



Time step size



Iteration per time step

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

OUTER ITERATION

ITER=1

⋮

ITER=ITER+1

⋮

TIME=TLAST

ITER=LAST

NT=1

⋮

NT=i

- Two line iterations in x direction
- Two line iterations in y direction
- Two block corrections in x,y direction

NTIMES(NF)

Number of specified iteration cycles

Outer Iteration

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.

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

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

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

Sufficient condition for iteration convergence of Jakob and Gauss-Seidel iteration.

1. Sufficient condition－Scarborough criterion

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

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

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

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

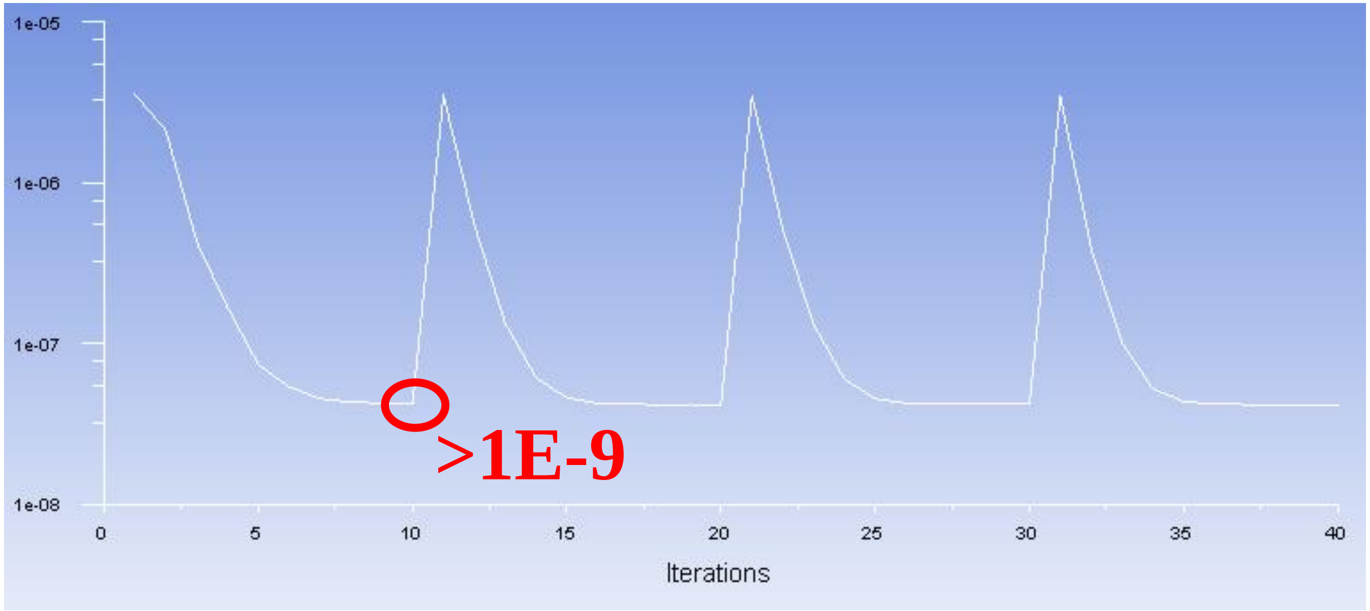
However, Δt will affect the accuracy of the simulation results.

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.

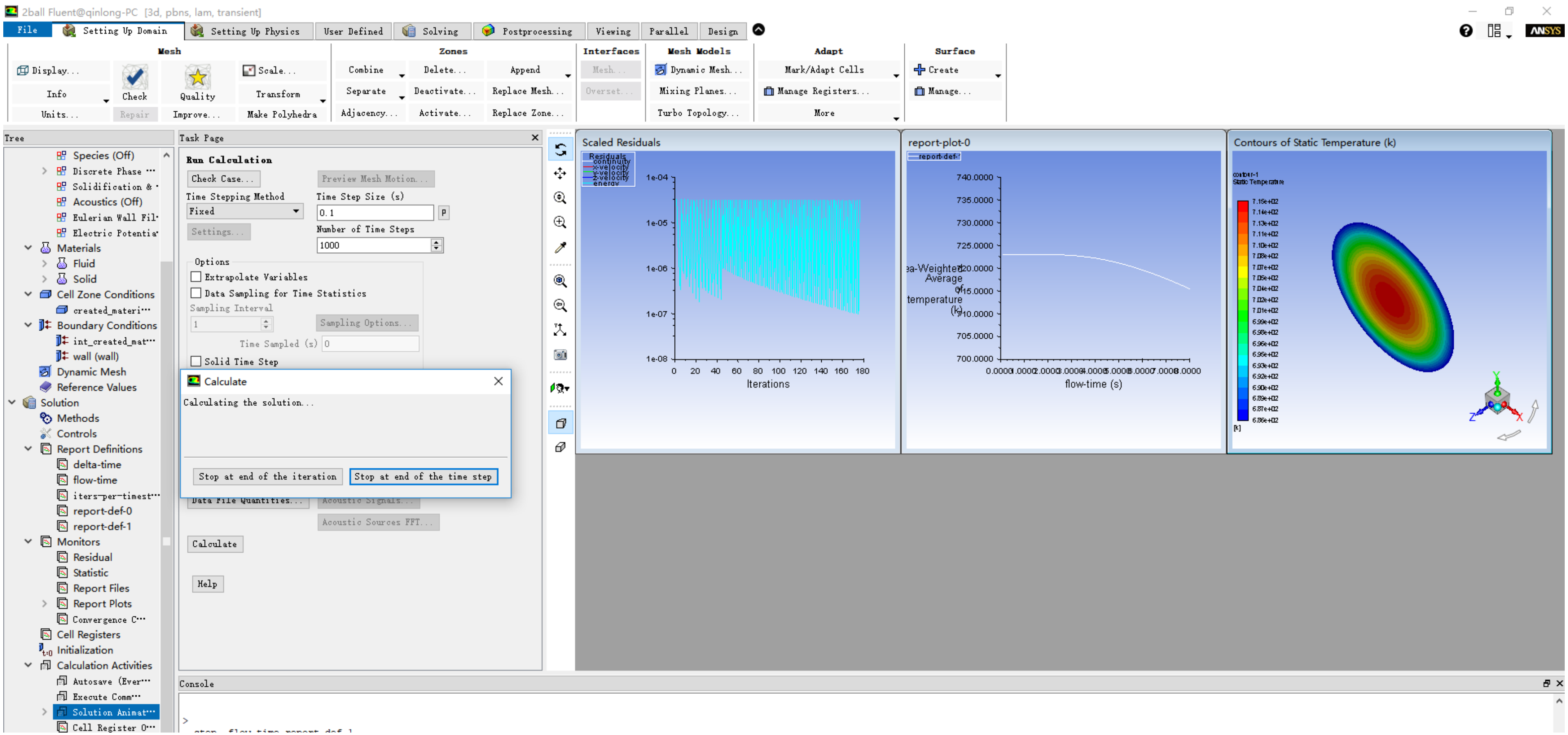
Usually, Δt should be small at beginning and then can be increased after 5-10 time steps.

Time stepping method

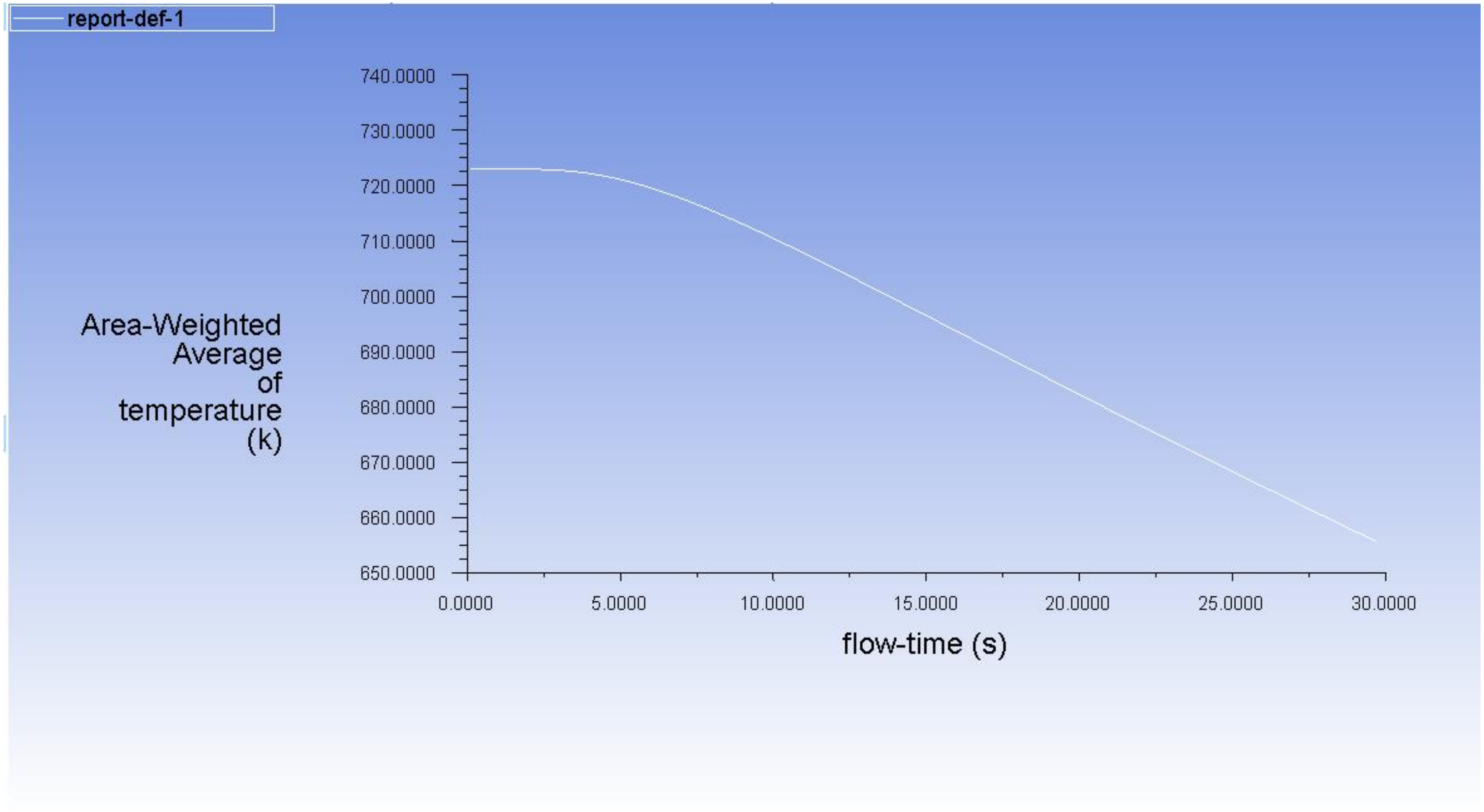
Here for Example 2, you can simply set the time stepping method as fixed, indicating the time step size is not changed during the iteration.

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



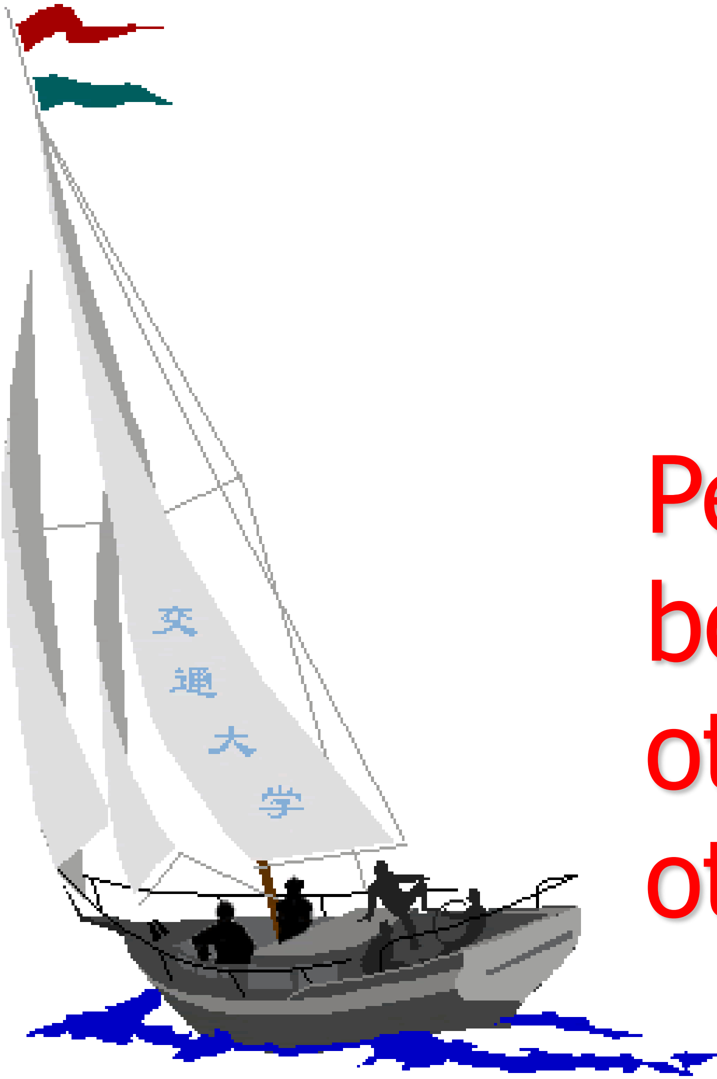
The average temperature on “point-0” change by time is as below:



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)



同舟共济
渡彼岸!

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