

Numerical Heat Transfer

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



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, 2019-Dec.-17

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

2019年12月17日, 西安

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

13.1 Heat transfer with source term

13.2 Unsteady cooling process of a steel ball

13.3 Lid-driven flow and heat transfer

13.4 Flow and heat transfer in a micro-channel

13.5 Flow and heat transfer in chip cooling

13.6 Phase change material melting with fins

第 13 章 求解流动换热问题的Fluent软件应用举例

13.1 有内热源的导热问题

导热问题

13.2 非稳态圆球冷却问题

13.3 顶盖驱动流动换热问题

混合对流问题

13.4 微通道内流动换热问题

13.5 芯片冷却流动换热问题

微通道问题

13.6 肋片强化相变材料融化

相变传热

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.

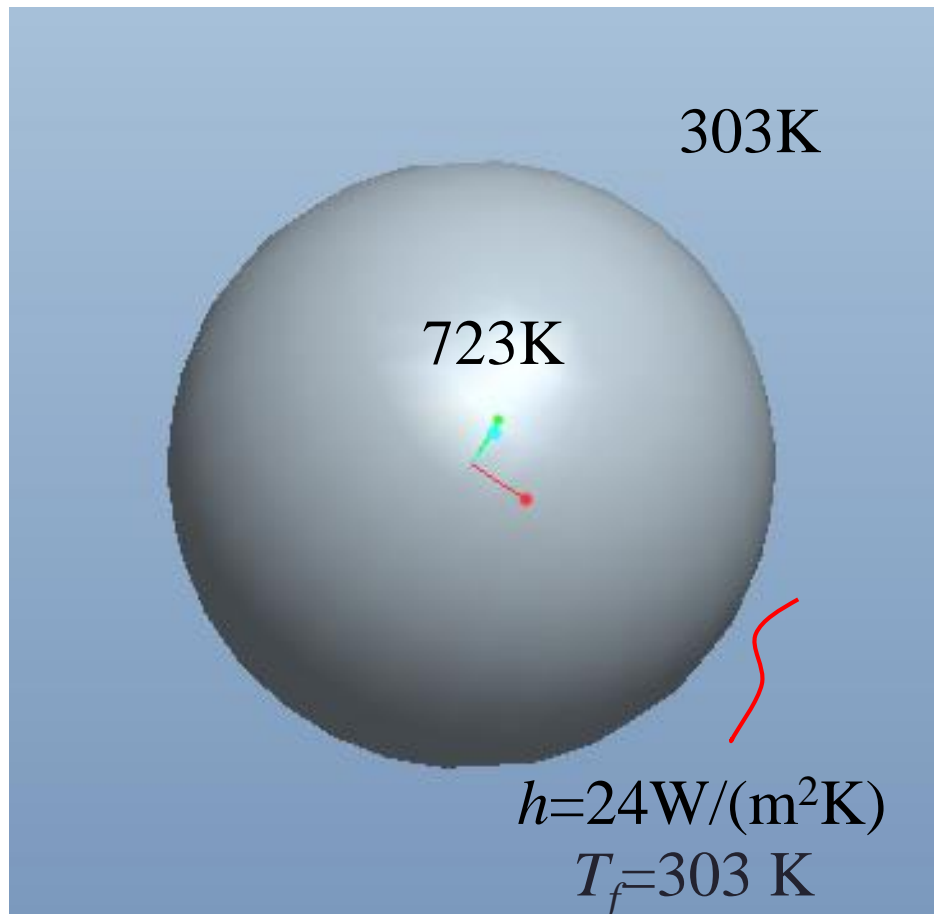
($D=5$ cm, density is 7735kg/m^3 , capacity is 480 J/(kg K), conductivity is $33\text{W}/(\text{m K})$).

- 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.

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 an unsteady heat conduction problem with given GAMA.

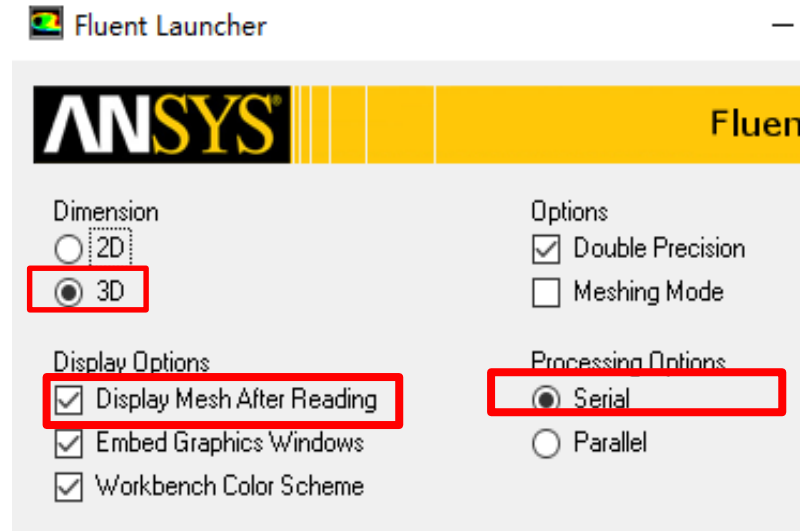
Remark: here we write the energy governing equation in the improved form with nominal density ρC_p . The improved form is adopted in our general teaching code as well in Fluent.

Start the Fluent software

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

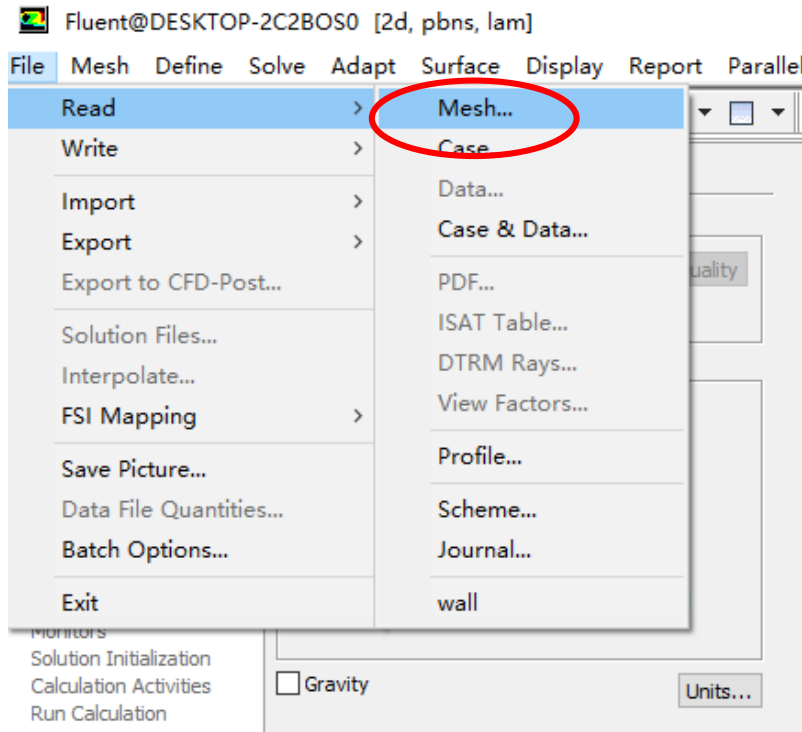
2、 If “display mesh after reading” is selected, after the Fluent is launched, the mesh will automatically shown in the interface.

3、 For most cases the single precision version of Fluent is sufficient. For heat transfer problem, **if the thermal conductivity 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

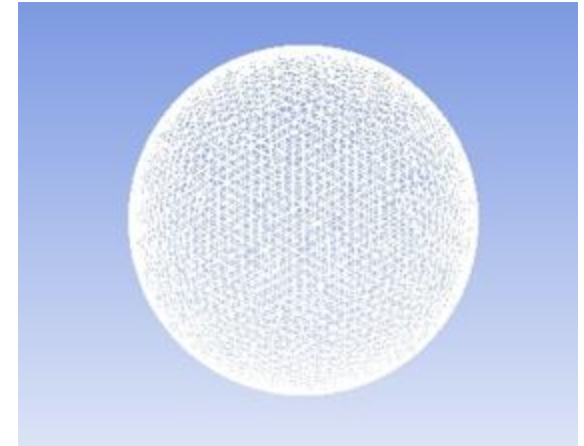
Check quality and topological information of the mesh

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

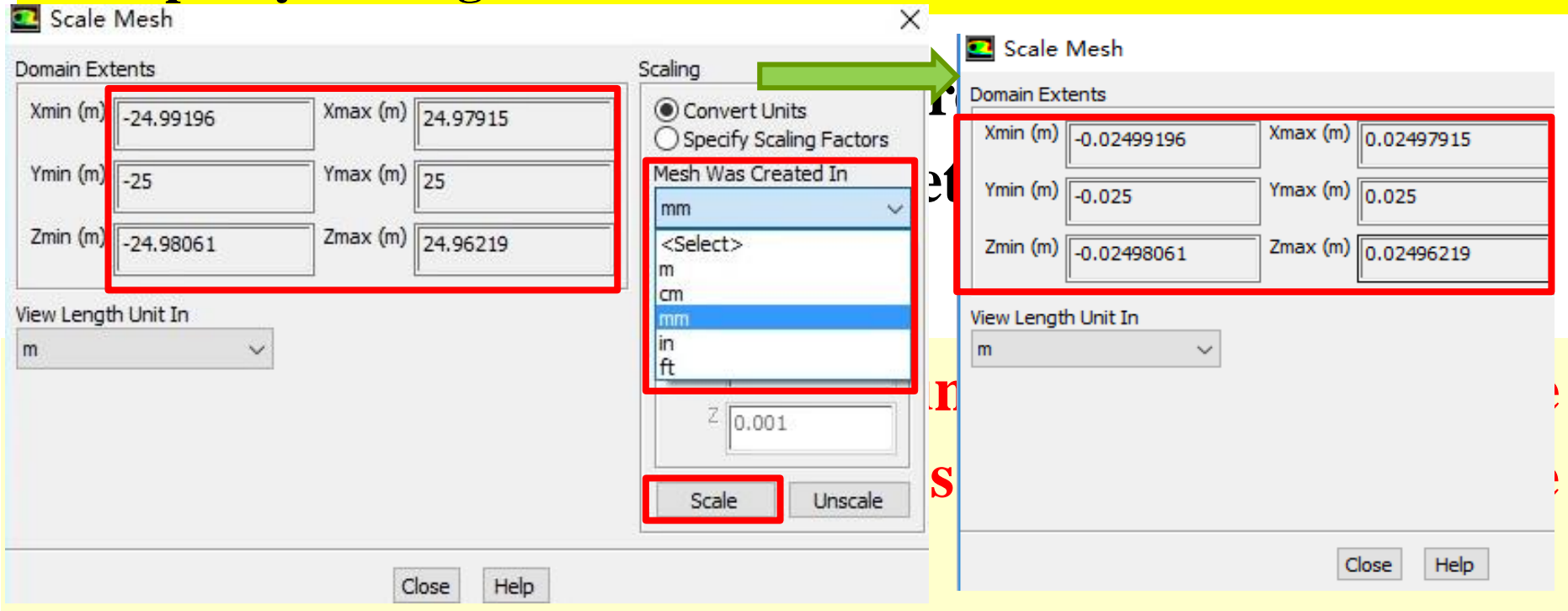
WARNING: Mesh check failed.

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

2st step: Scale the domain size

General → Scale

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



Scale Mesh

Domain Extents

Xmin (m)	-24.99196	Xmax (m)	24.97915
Ymin (m)	-25	Ymax (m)	25
Zmin (m)	-24.98061	Zmax (m)	24.96219

View Length Unit In: m

Scaling

Convert Units
 Specify Scaling Factors

Mesh Was Created In

- mm
- <Select>
- m
- cm
- mm
- in
- ft

Z: 0.001

Scale Unscale

Close Help

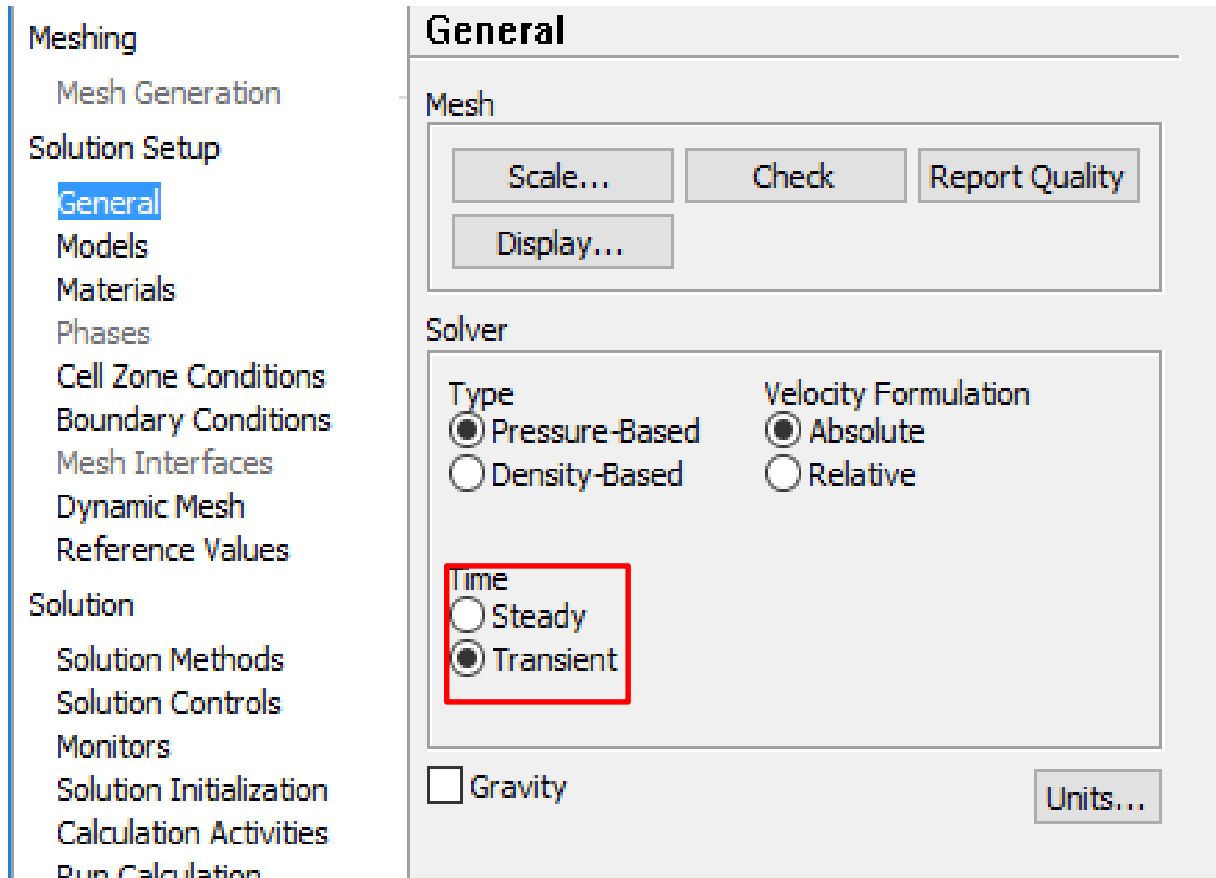
Scale Mesh

Domain Extents

Xmin (m)	-0.02499196	Xmax (m)	0.02497915
Ymin (m)	-0.025	Ymax (m)	0.025
Zmin (m)	-0.02498061	Zmax (m)	0.02496219

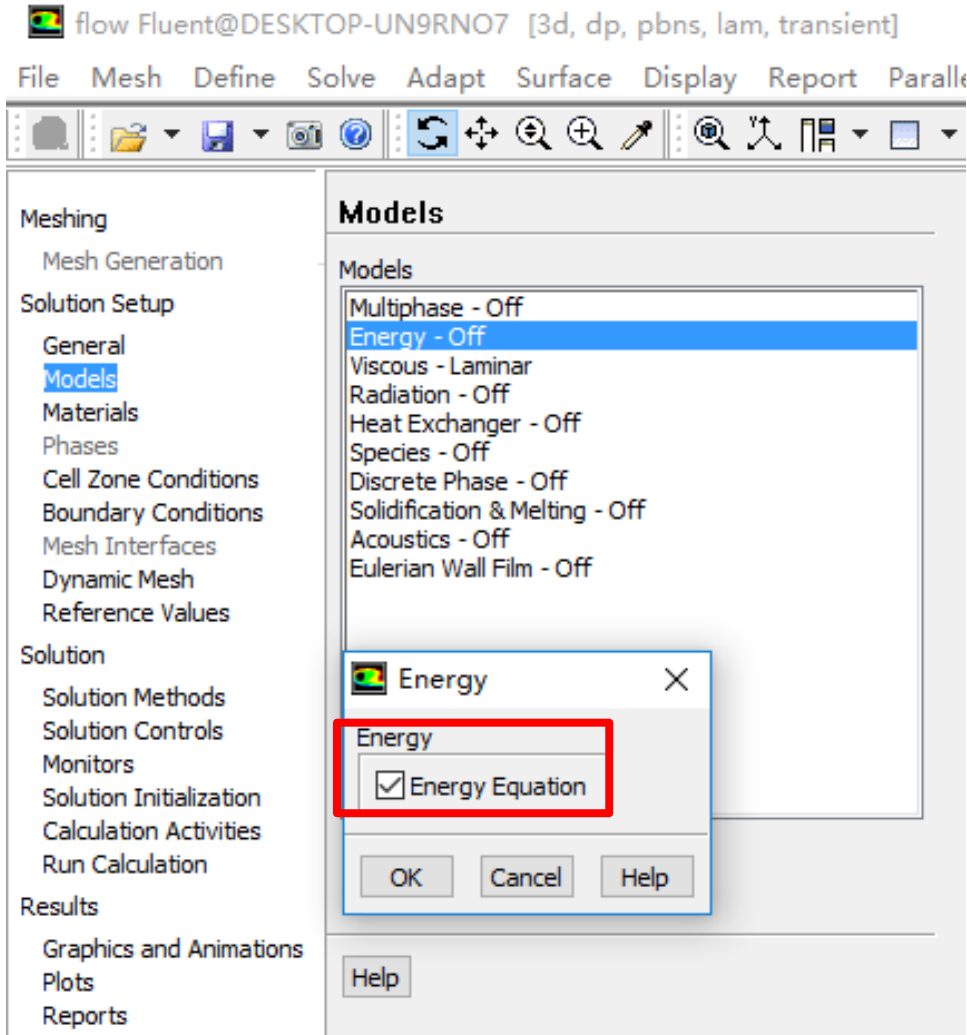
View Length Unit In: m

Close Help



Choose the “transient” for a unsteady problem!

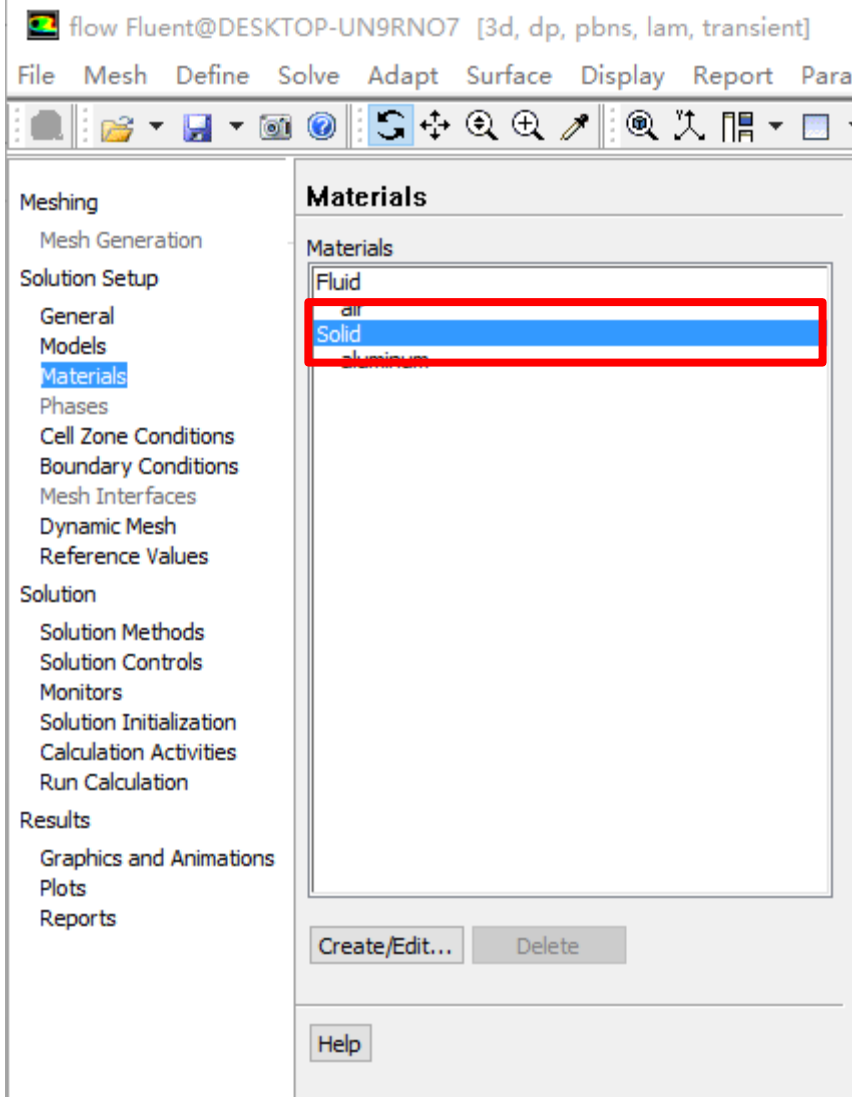
Step 3: Choose the physicochemical model



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

The energy equation is activated.

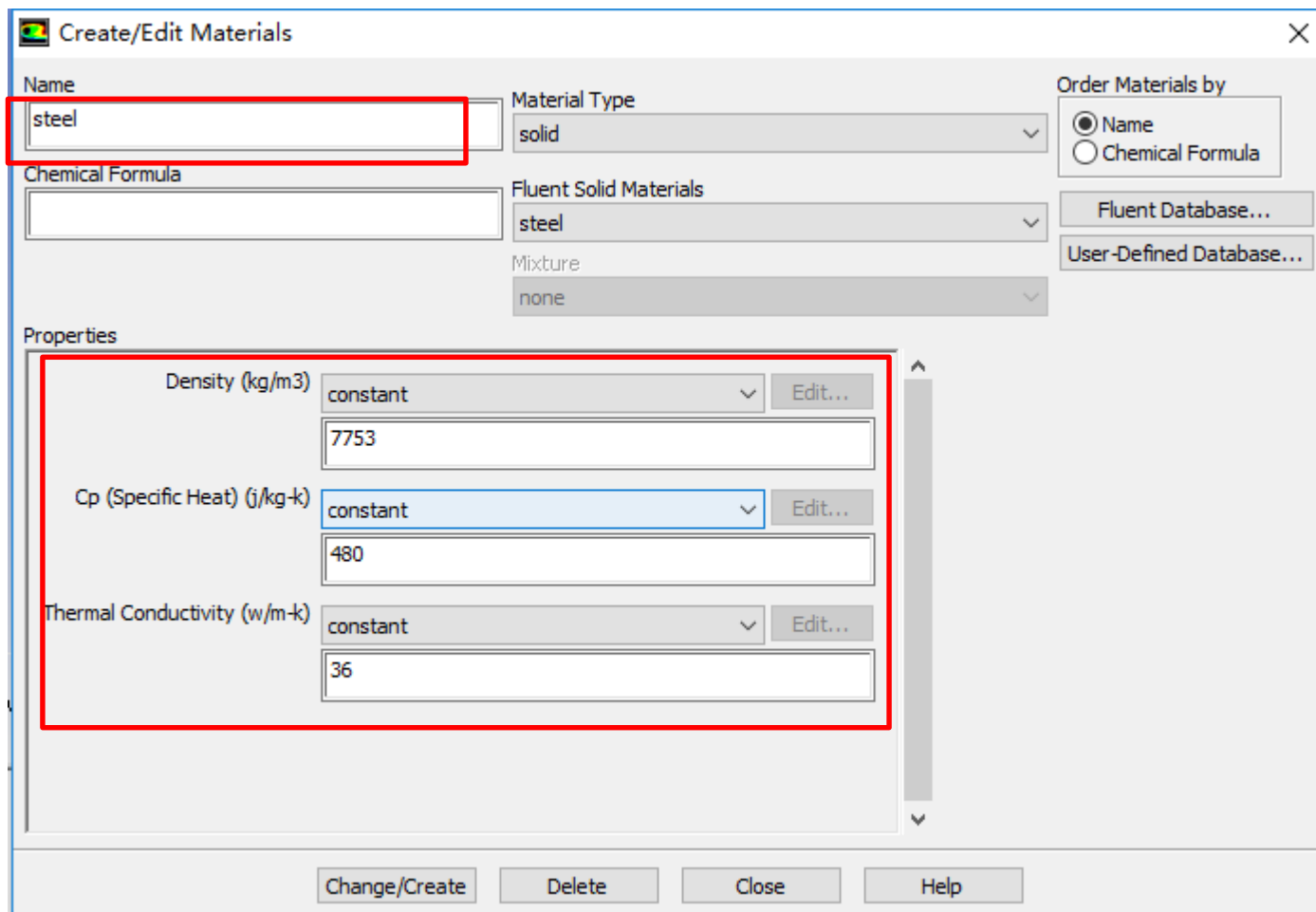
Step 4: Define the material properties



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

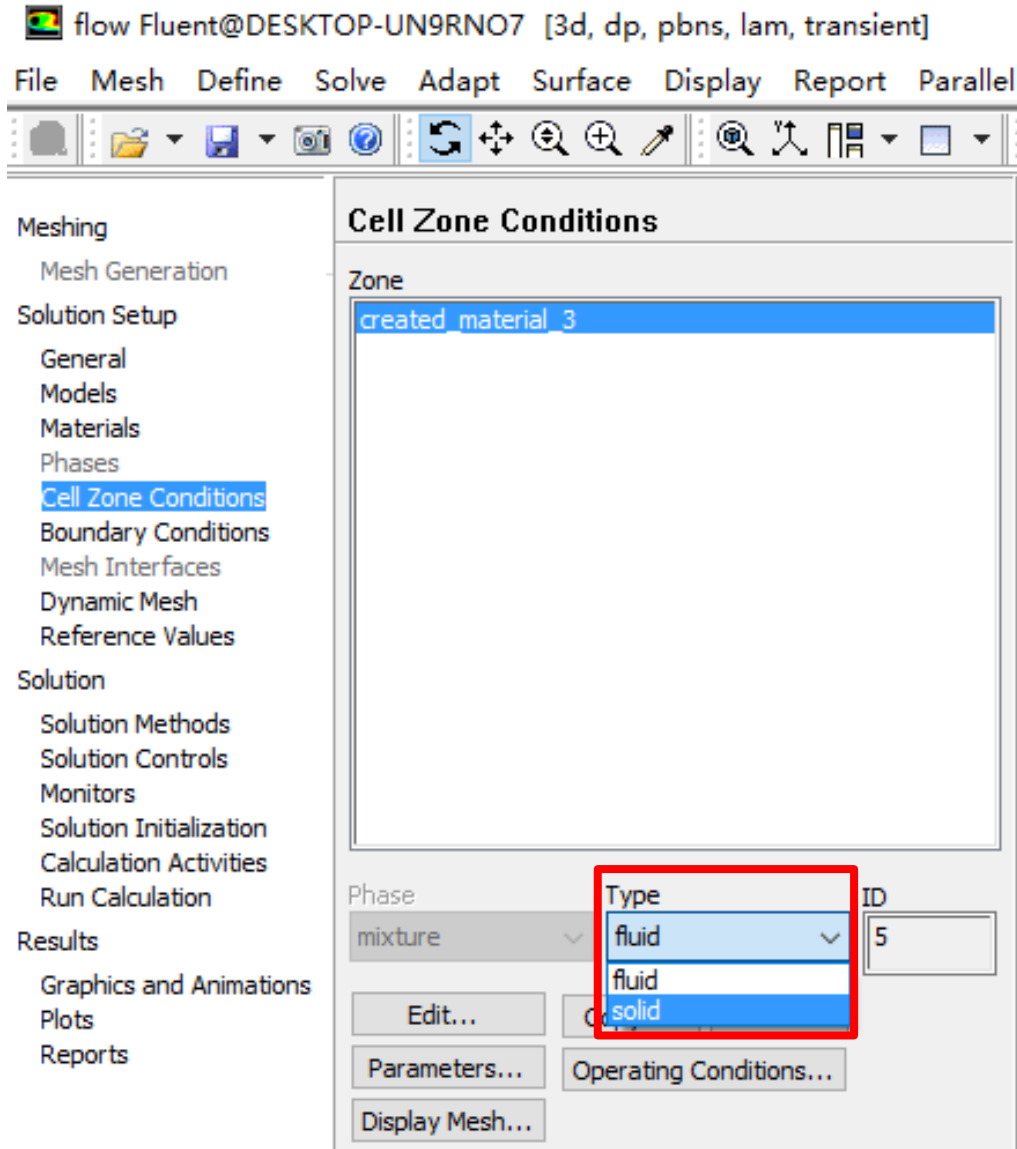
**The default solid in Fluent
is Aluminum.**

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

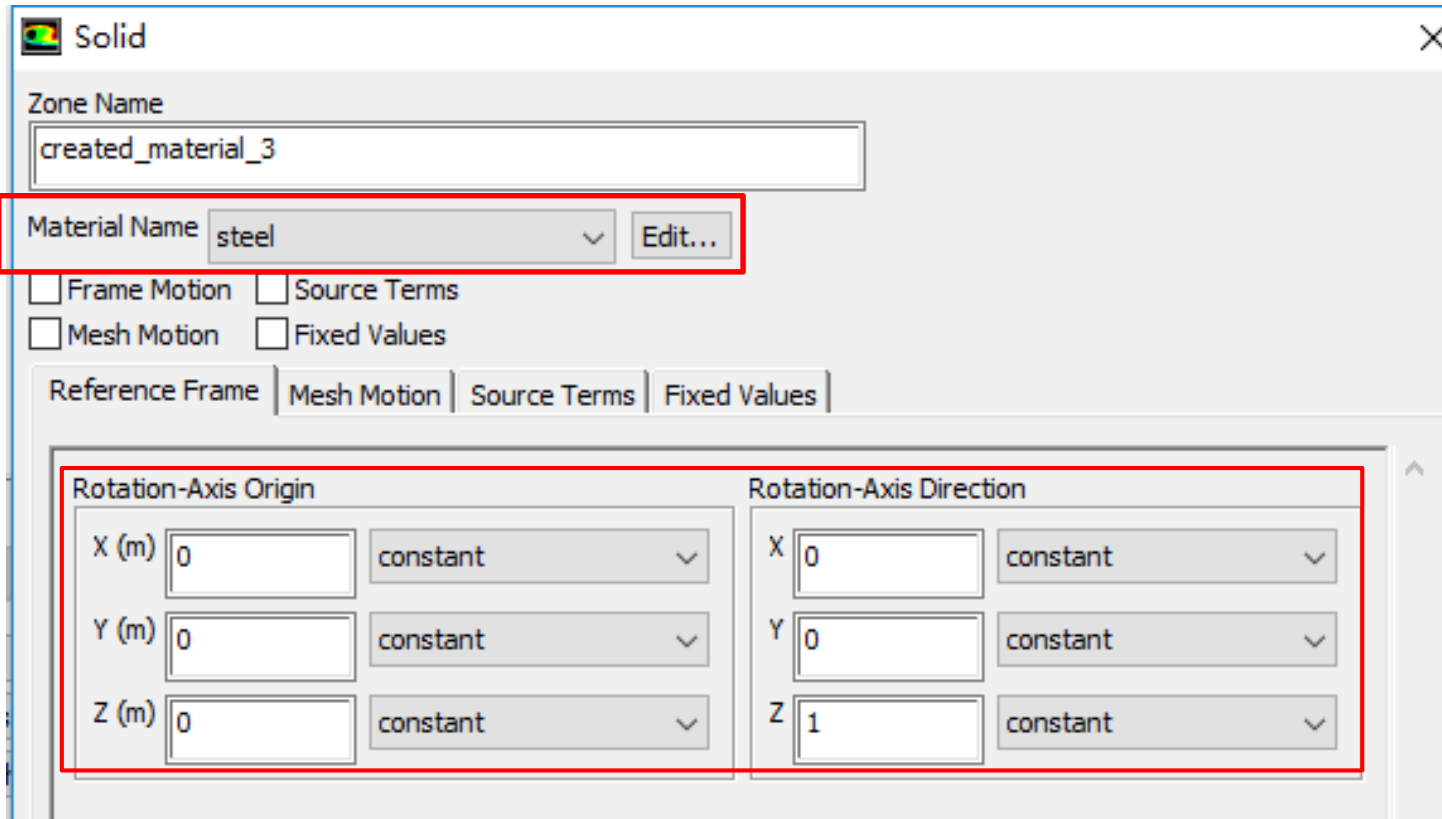


The properties of steel are manually inputted.

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

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

Heat Transfer Coefficient (w/m²-k)

constant

Free Stream Temperature (k)

constant

Material Name

Heat Generation Rate (w/m³)

constant

Wall Thickness (m)

P

Shell Conduction
 Define...

In this problem, the BC is third kind of boundary condition, so we select “Convection” and input 240 for “Heat Transfer Coefficient”, and 303K for the “Free Stream Temperature”.

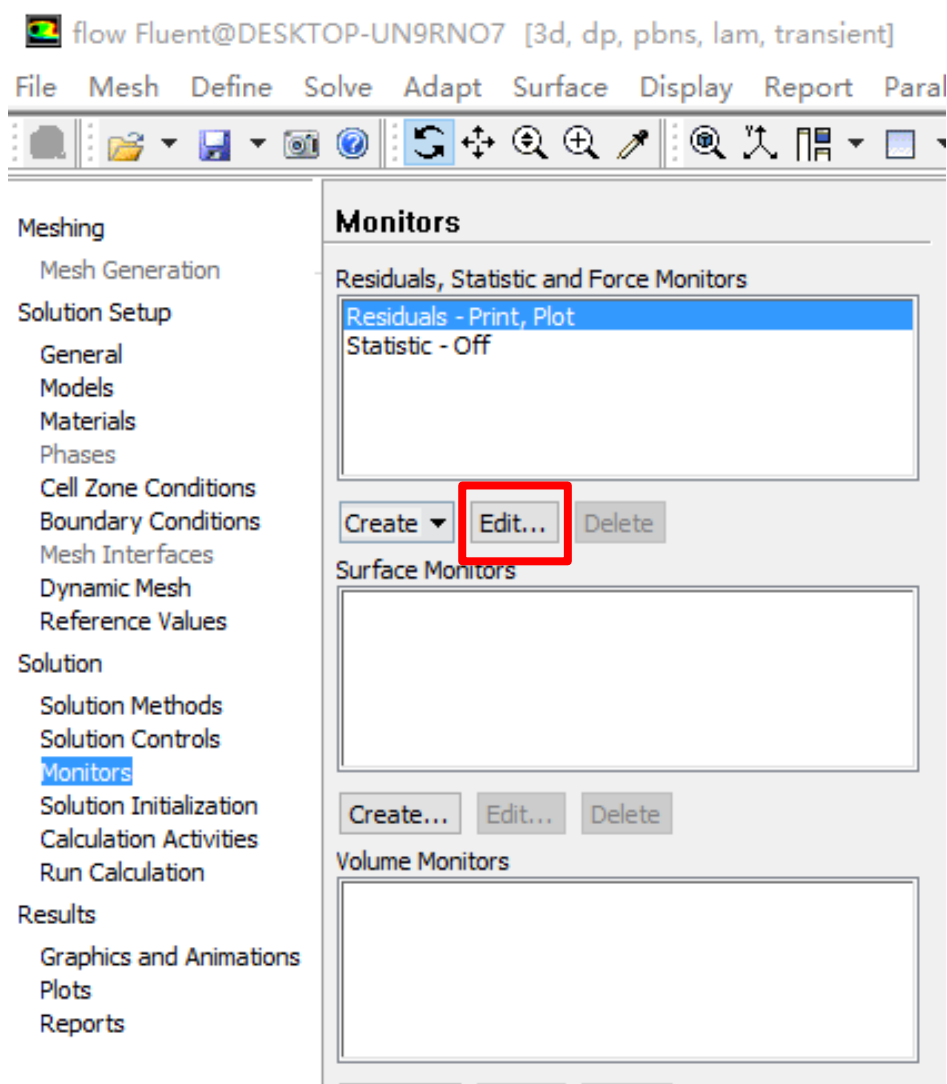
Step 7: Solution setup: algorithm and scheme

The screenshot displays the ANSYS Fluent software interface. The top menu bar includes File, Mesh, Define, Solve, Adapt, Surface, Display, Report, and Paralle. The main window is divided into several panels:

- Mesging Panel:** Contains 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), and Results (Graphics and Animations, Plots, Reports).
- Solution Methods Panel:**
 - Pressure-Velocity Coupling: Scheme is set to 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.

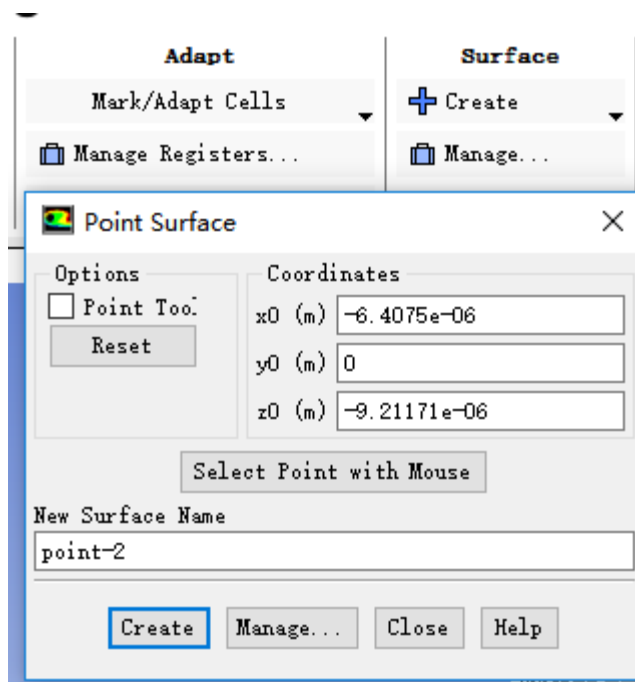
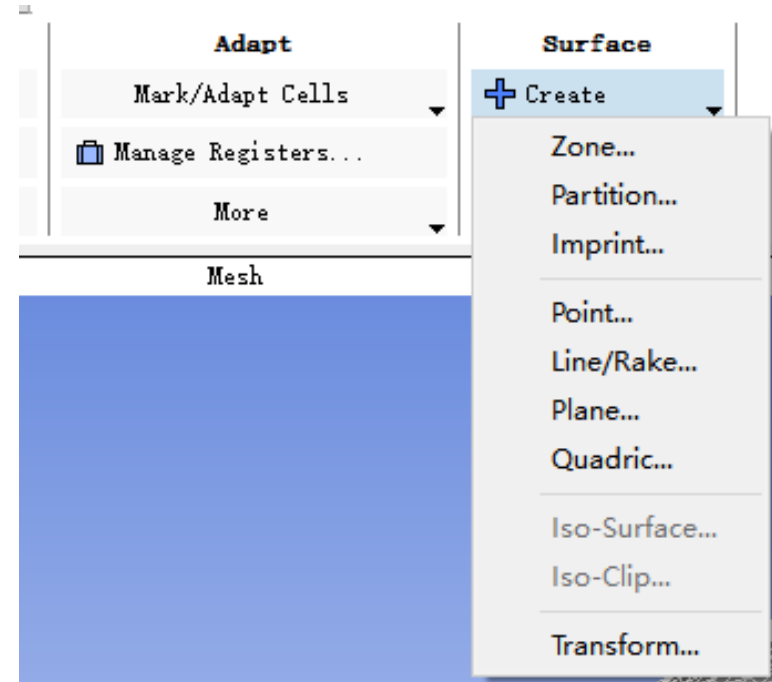
Step 7: Solution setup: monitors



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

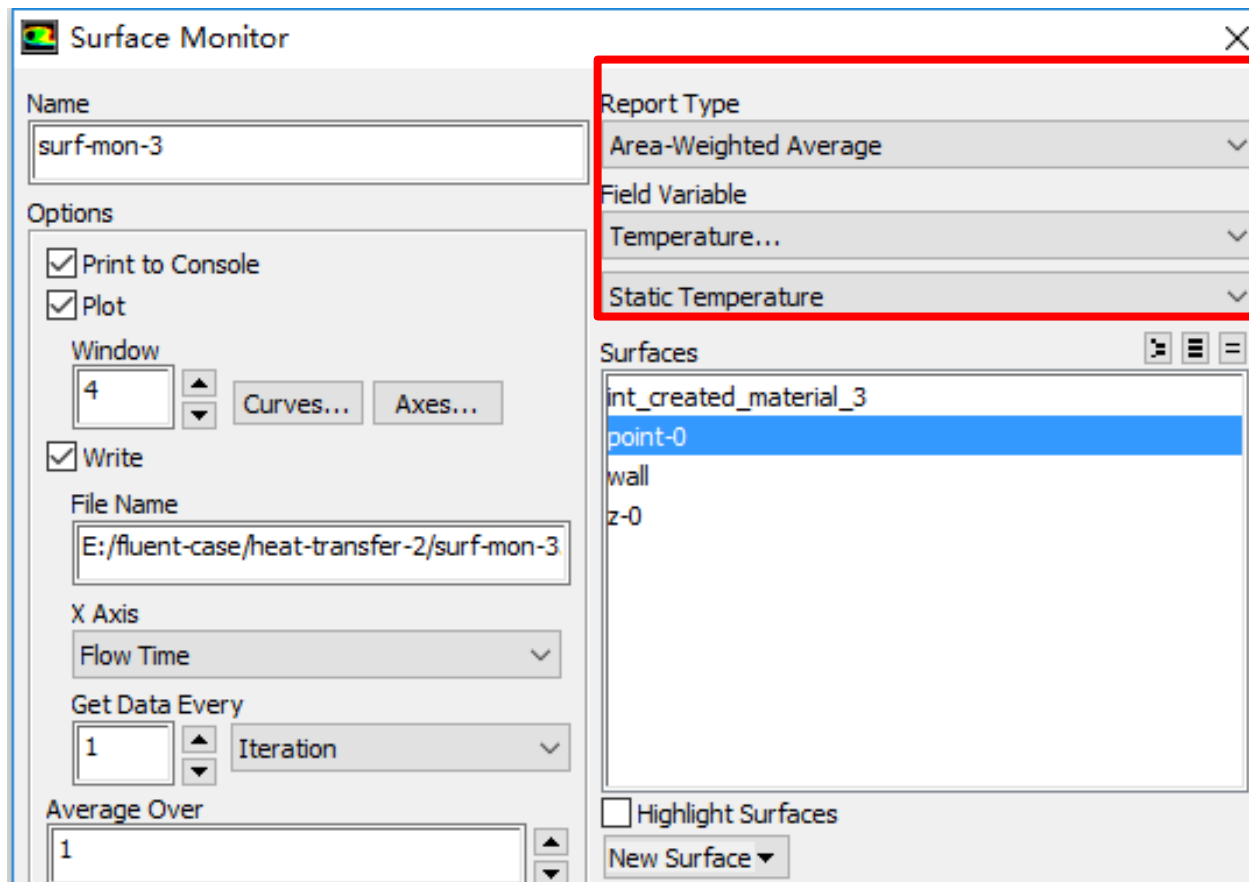
z-0

Create Manage... Close Help

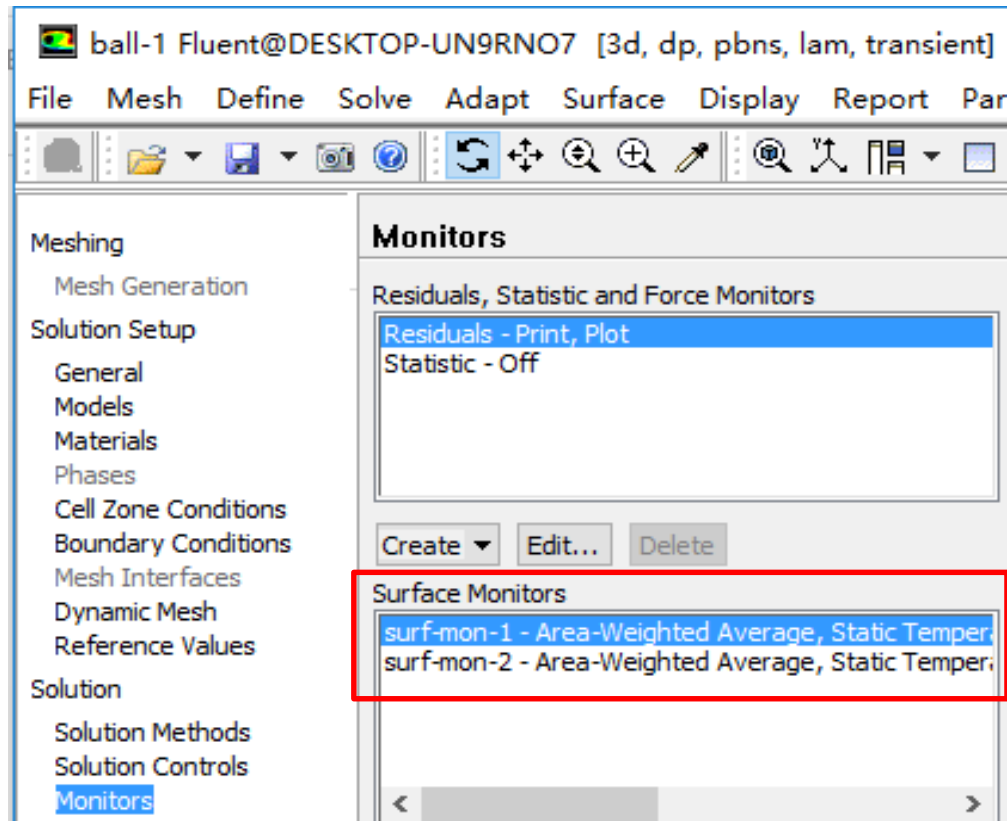
OP-UN9RNO7 [3d, dp, pbns, lam, transient]

Surface

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



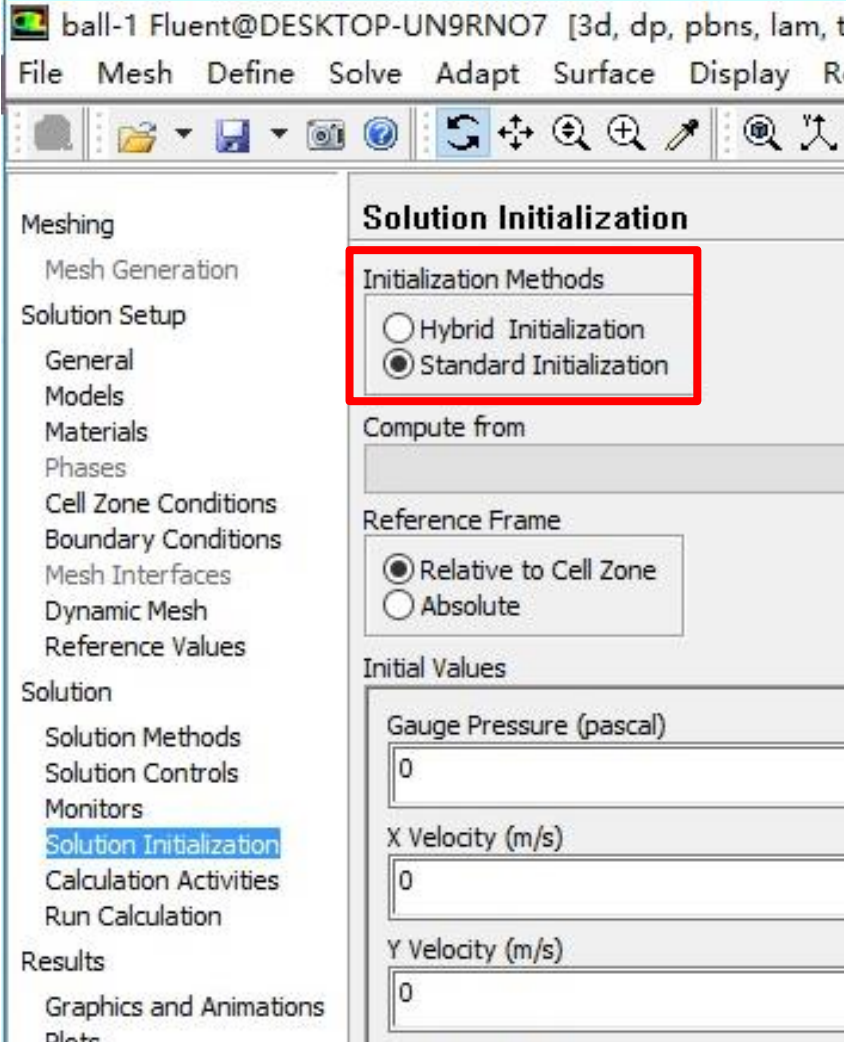
Next, you can create the monitors in the “Monitors” dialog. Select the “Report type”, the variable you want to monitor, and the position you want to monitor.



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



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.

The image shows the ANSYS Fluent interface with the **Patch** dialog box open. The **Patch** dialog has the following settings:

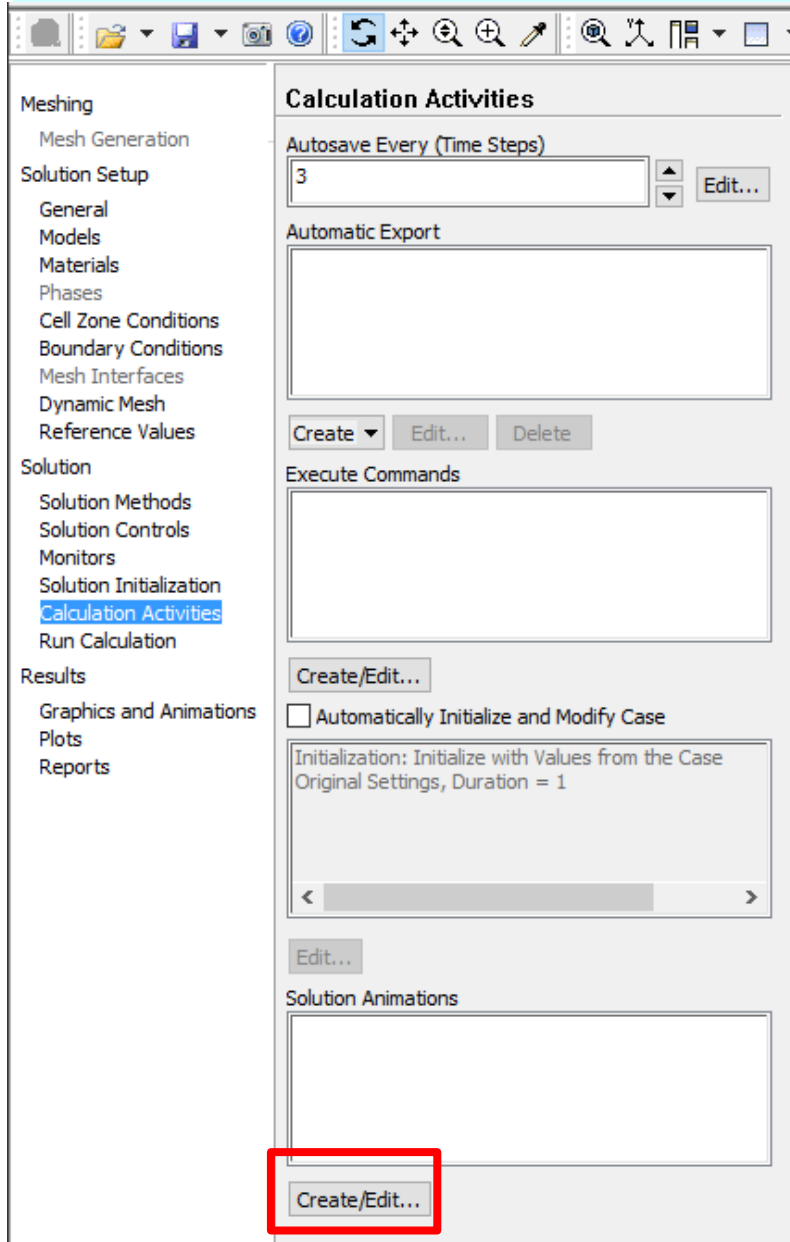
- Reference Frame:** Relative to Cell Zone, Absolute
- Value (k):** 723
- Use Field Function:**
- Variable:** Temperature
- Zones to Patch:** created_material_3
- Registers to Patch:** (empty)
- Buttons:** Patch, Close, Help

The **Solution Initialization** panel on the left shows the following settings:

- Initialization Methods:** Hybrid Initialization, Standard Initialization
- 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
- Buttons:** Initialize, Reset, Patch..., Reset DPM Sources, Reset Statistics, Help

The bottom right of the image shows a **Contours of Static Temper** plot with a color scale ranging from 7.23e+02 (blue) to 7.23e+02 (red). A large green circle is visible on the right side of the plot area.

9st step: set animations



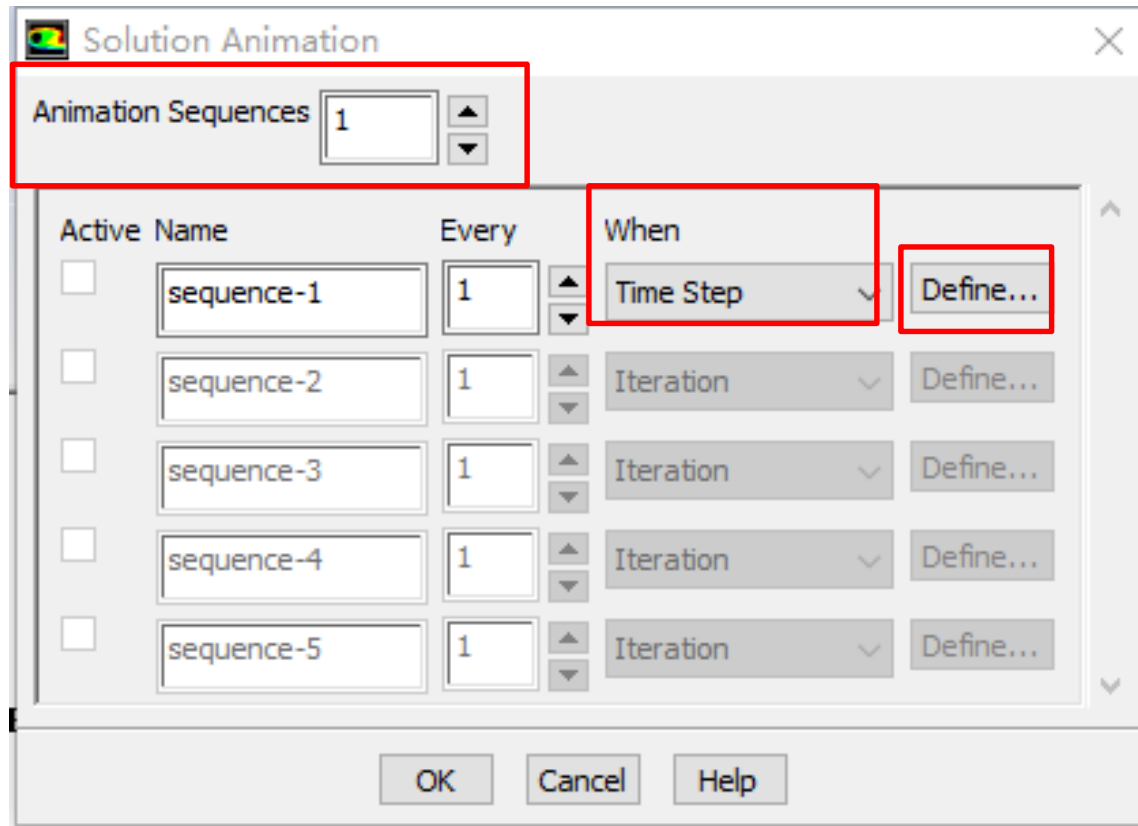
The screenshot displays the ANSYS Fluent software interface. The left sidebar shows the 'Calculation Activities' tab selected. The main window shows the 'Calculation Activities' dialog box with the following settings:

- Autosave Every (Time Steps):** 3
- Automatic Export:** (Empty text area)
- Execute Commands:** (Empty text area)
- Initialization:** Initialize with Values from the Case Original Settings, Duration = 1
- Solution Animations:** (Empty text area)

The 'Create/Edit...' button at the bottom of the dialog box is highlighted with a red rectangle.

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

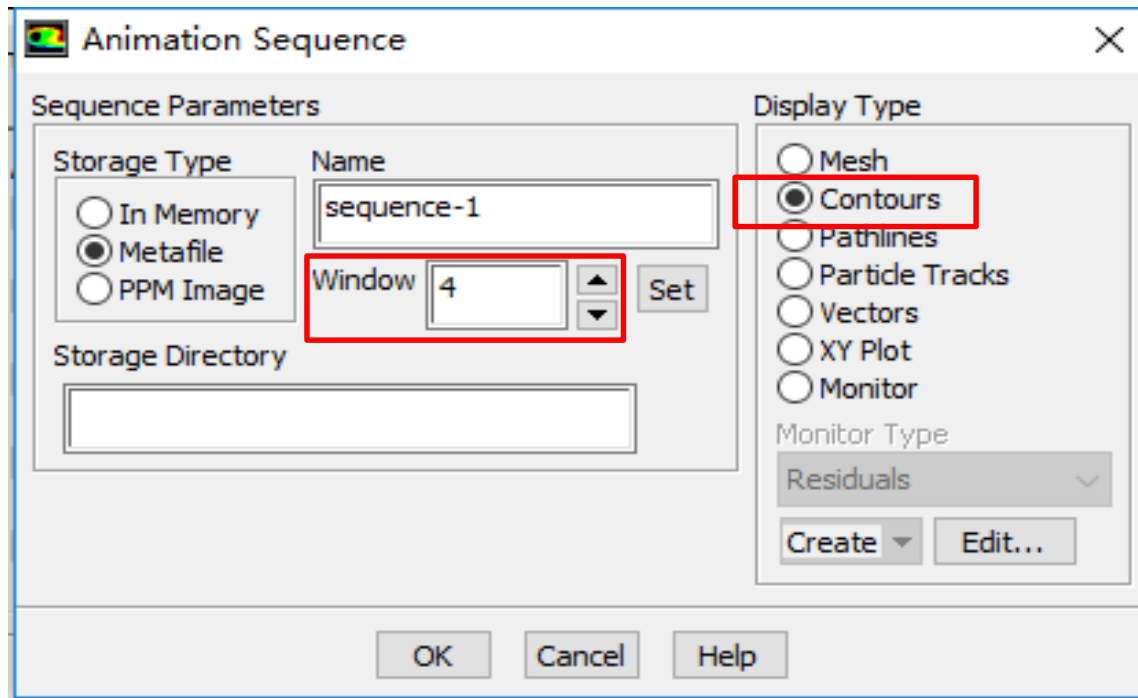
In the “Calculation Activities” dialog, click “Change/Create” in “Solution Animations”.



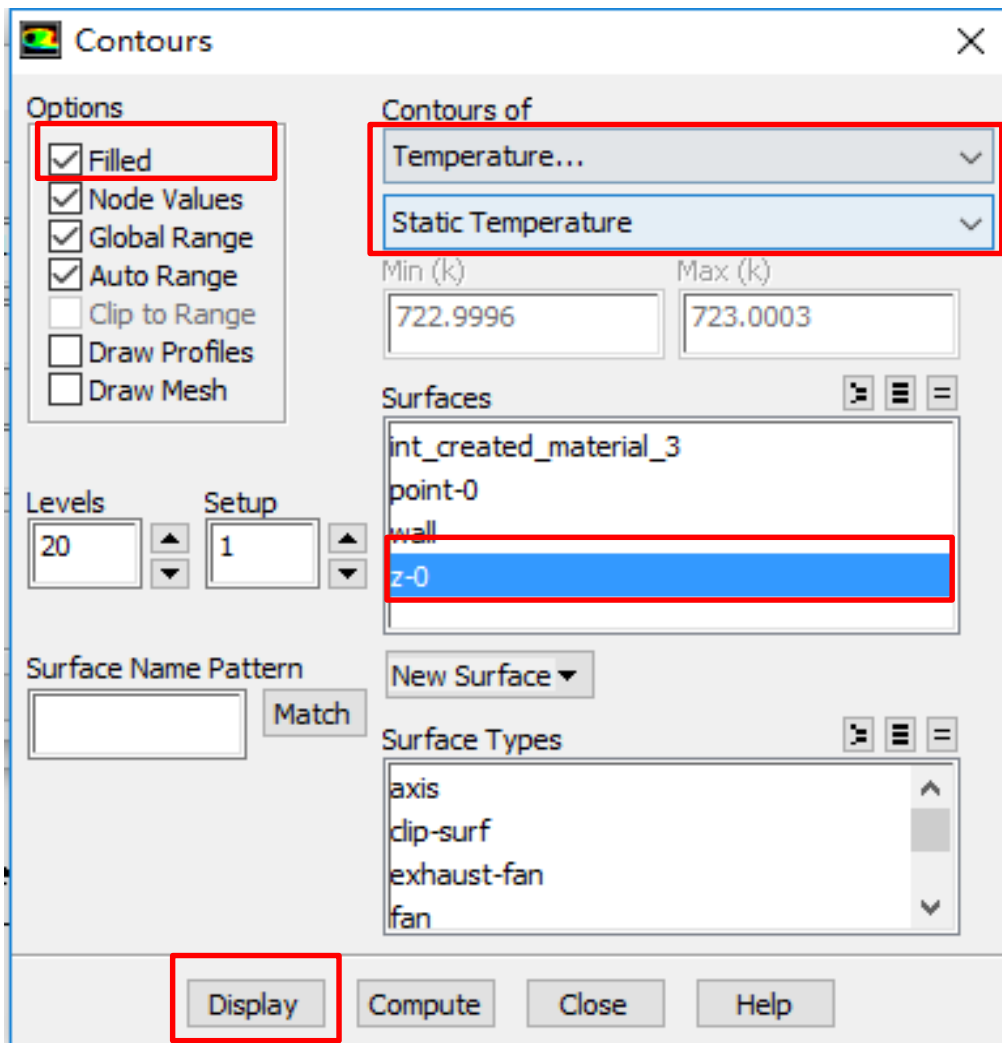
Set the “Animation Sequences” as 1.

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.**

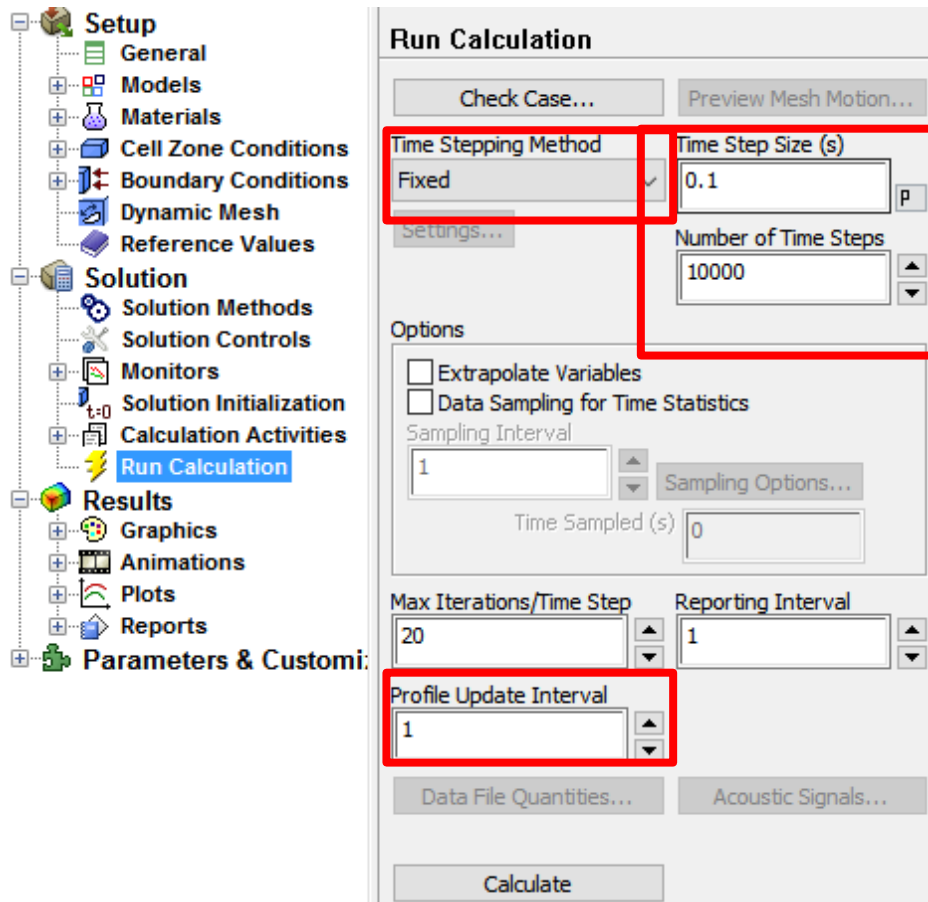


In “Contours” dialog, we choose “Temperature”, select “Filled”, and choose the surface: z-0.

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.



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



OUTER ITERATION

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 adopted in Fluent. Therefore, the value of Δt will not affect the stability. However, it will affect the accuracy.

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

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

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

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) :

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

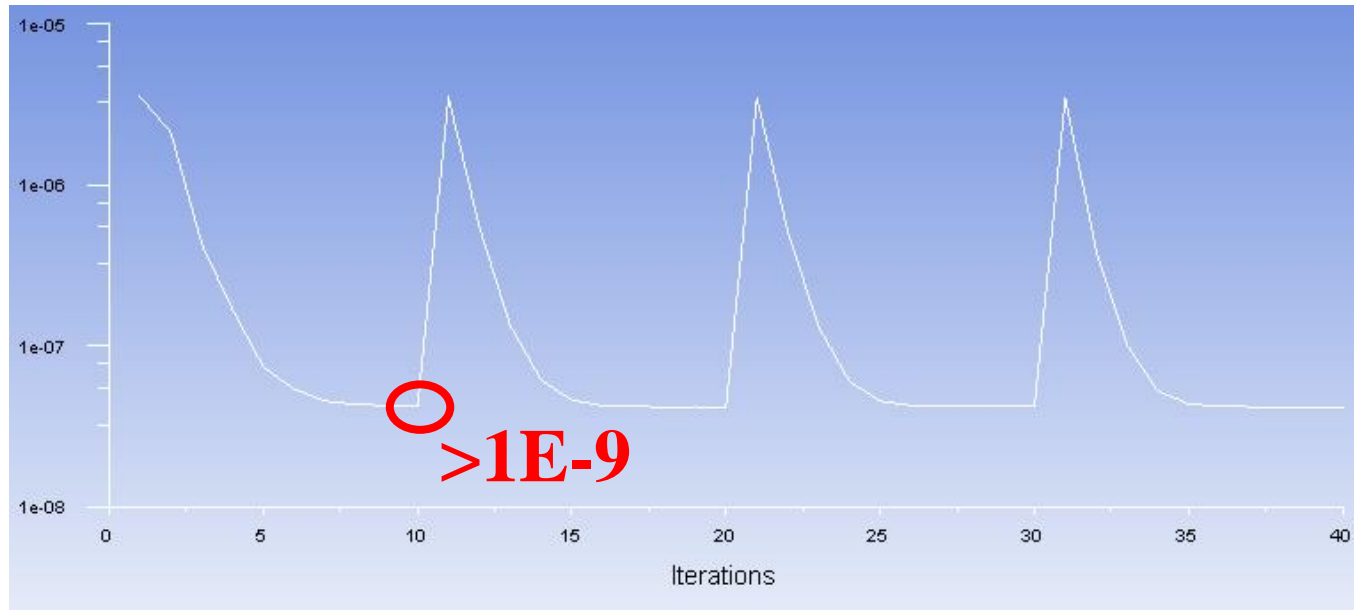
$$a_p = a_E + a_W + a_N + a_S + a_p^0 - S_p \Delta V$$

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

However, 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.

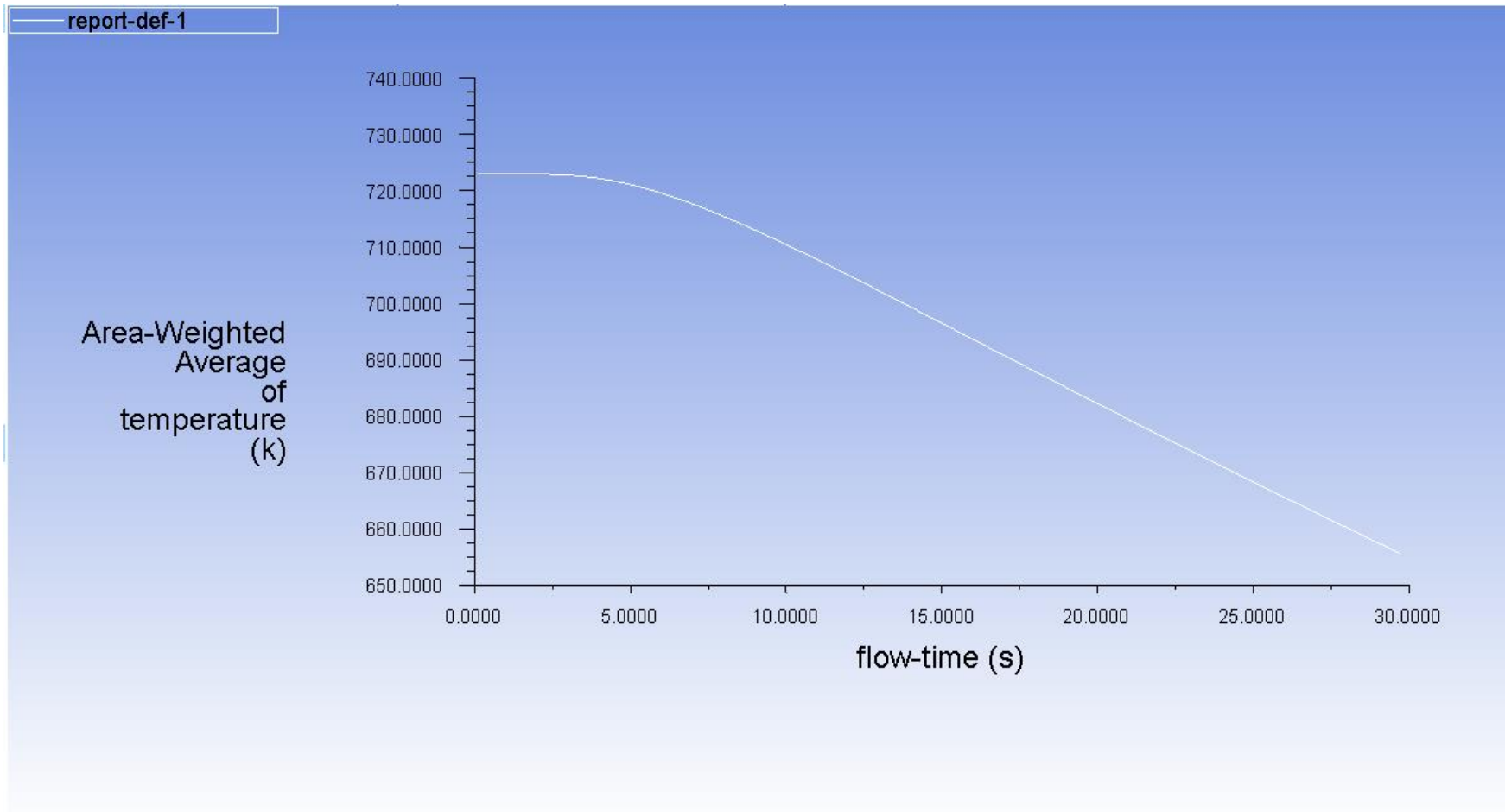
For some problem, it is reasonable to chose Adaptive method in which Δt is dynamically changed. For example, in multiphase flow simulation using VOF, you can use this function to update the phase interface more efficiently.

Run the simulation

The screenshot displays the ANSYS Fluent software interface during a simulation. The main window is divided into several panels:

- Tree:** Shows the simulation setup hierarchy, including Species (Off), Discrete Phase, Solidification, Acoustics (Off), Eulerian Wall Fil., Electric Potential, Materials (Fluid, Solid), Cell Zone Conditions, Boundary Conditions (int_created_mat, wall), Dynamic Mesh, Reference Values, and Solution (Methods, Controls, Report Definitions, Monitors, Calculation Activities).
- Task Page:** Contains the **Run Calculation** panel with settings for Time Stepping Method (Fixed), Time Step Size (0.1 s), and Number of Time Steps (1000). A **Calculate** dialog box is open, showing "Calculating the solution..." and options to stop at the end of the iteration or time step.
- Scaled Residuals:** A plot showing the convergence of residuals for velocity, pressure, and energy over 180 iterations. The residuals decrease significantly, indicating convergence.
- report-plot-0:** A plot showing the average temperature over time, with the temperature stabilizing around 725,000 K.
- Contours of Static Temperature (k):** A 3D visualization of the temperature distribution, showing a high-temperature region (red) in the center of the domain.

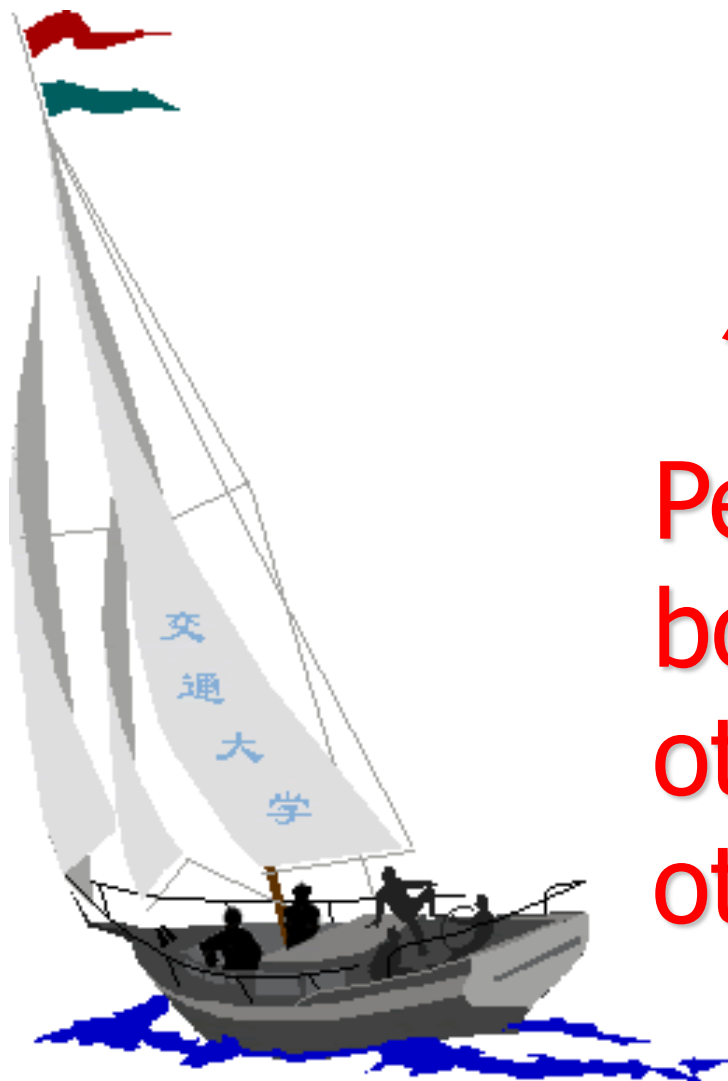
The average temperature on “point-0” change by time is as below:



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....