

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

13.3 Lid-driven flow and heat transfer

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

Known:

An infinite long solid plate with uniform temperature $T_{w1} = 80^{\circ}\text{C}$ is moving at the top of a square cavity with velocity $u=0.1\text{m/s}$. 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.

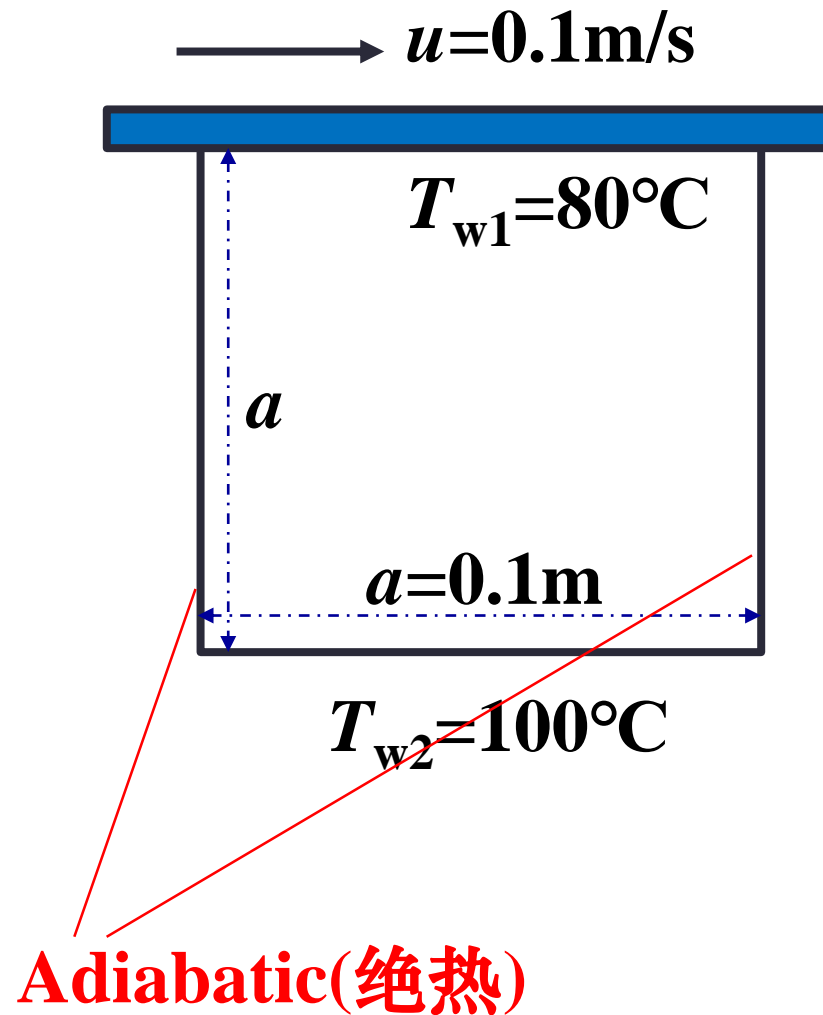


Fig.1 Computational domain

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:

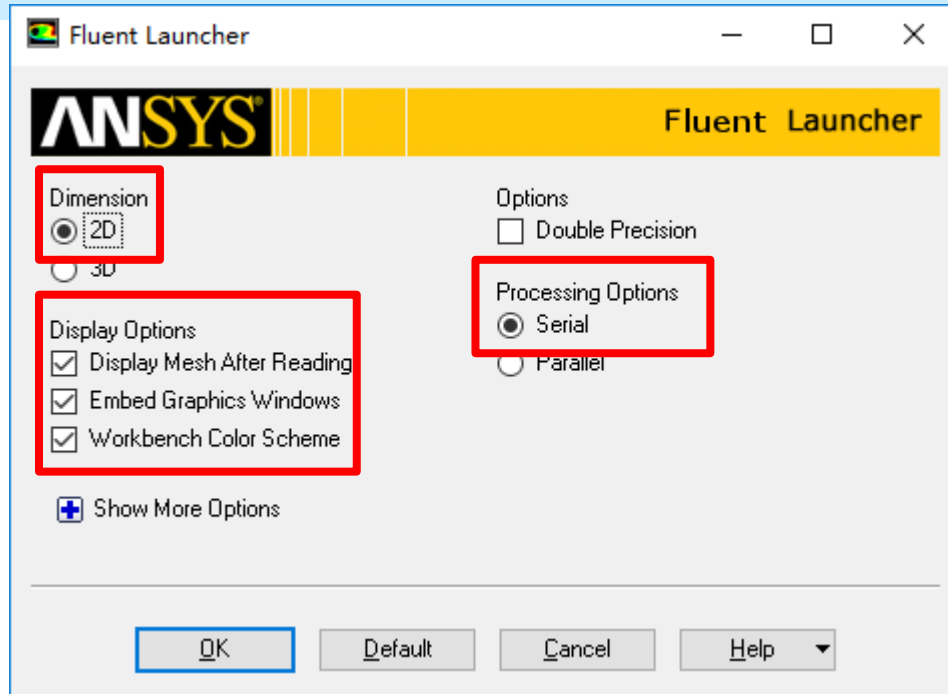
$$u_{max} = 0.1 \text{ m/s}, l = 0.1 \text{ m}, \nu = 1.46 \times 10^{-6} \text{ m}^2/\text{s}$$

$$Re = \frac{ul}{\nu} = 684$$

Laminar flow

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

Start the Fluent software



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

Note: Double precision or Single precision

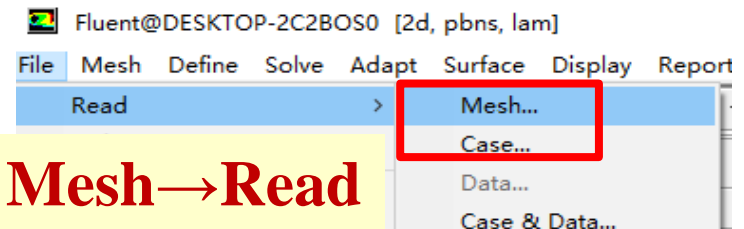
For most cases the single precision version of Fluent is sufficient. For heat transfer problem, **if the thermal conductivity between different components is high,** Double precision version is better.

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”

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



```
> 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
```

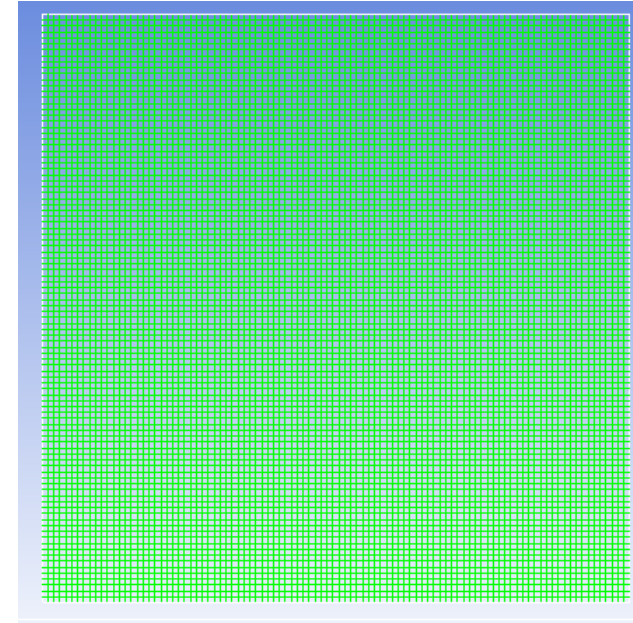
```
Done.
```

1st step: Read and **check** the mesh

Mesh→Check/Report quality

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

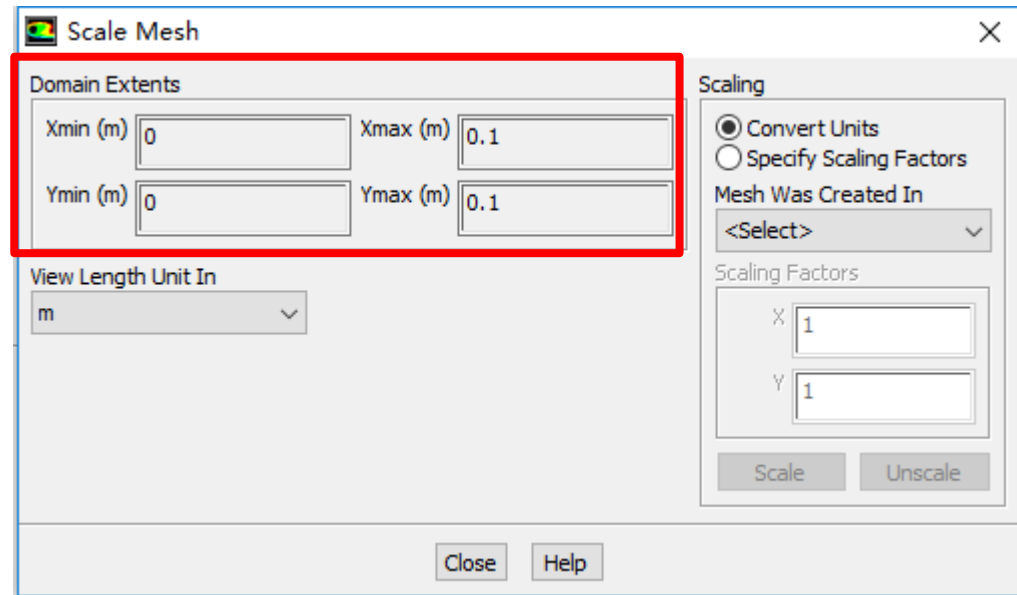
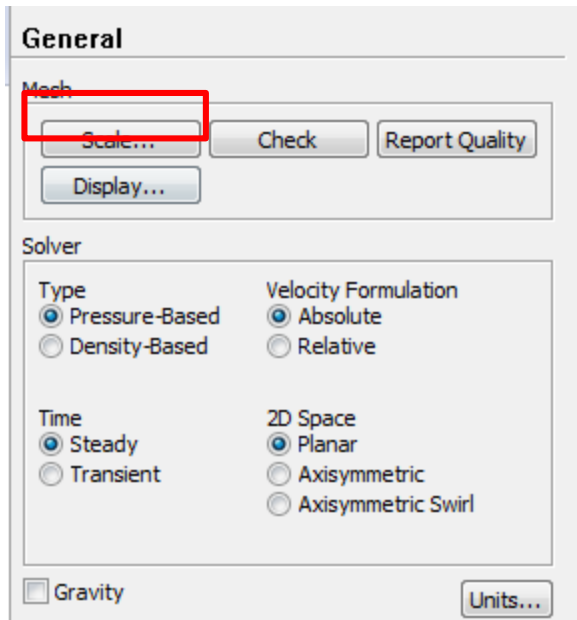
```
Domain Extents:
  x-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
  y-coordinate: min (m) = 0.000000e+00, max (m) = 1.000000e-01
Volume statistics:
  minimum volume (m3): 1.020304e-06
  maximum volume (m3): 1.020304e-06
  total volume (m3): 1.000000e-02
Face area statistics:
  minimum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

2nd step: Scale the domain size

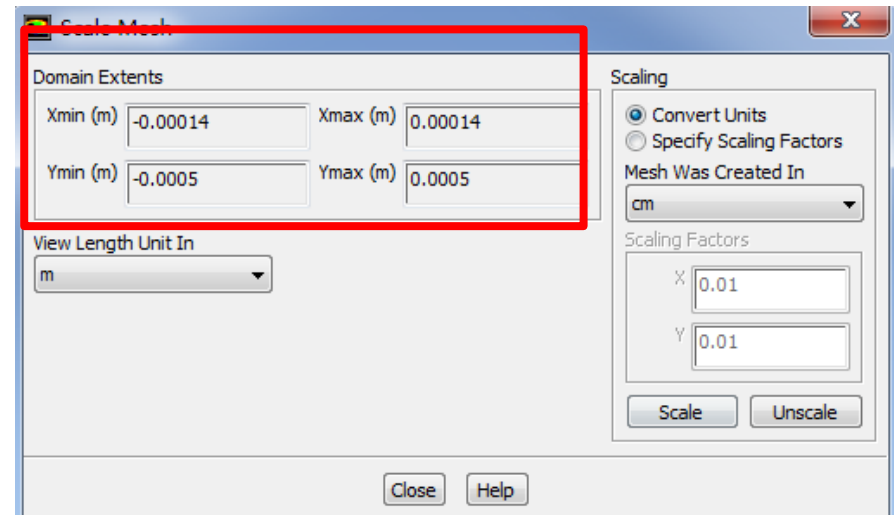
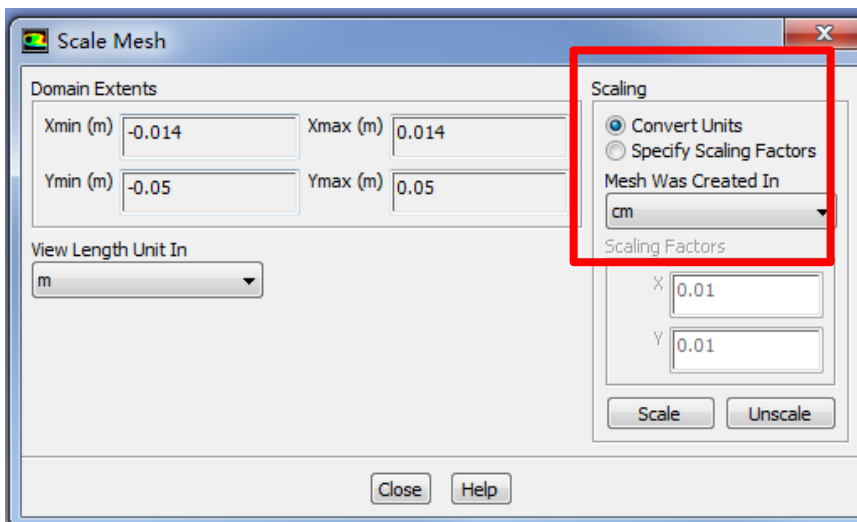
General→Scale



- Fluent stores the mesh in units as “m”, SI unit. You can show it in different units such as cm, mm, in, or ft.
- This time ,we don’t need to scale the mesh.

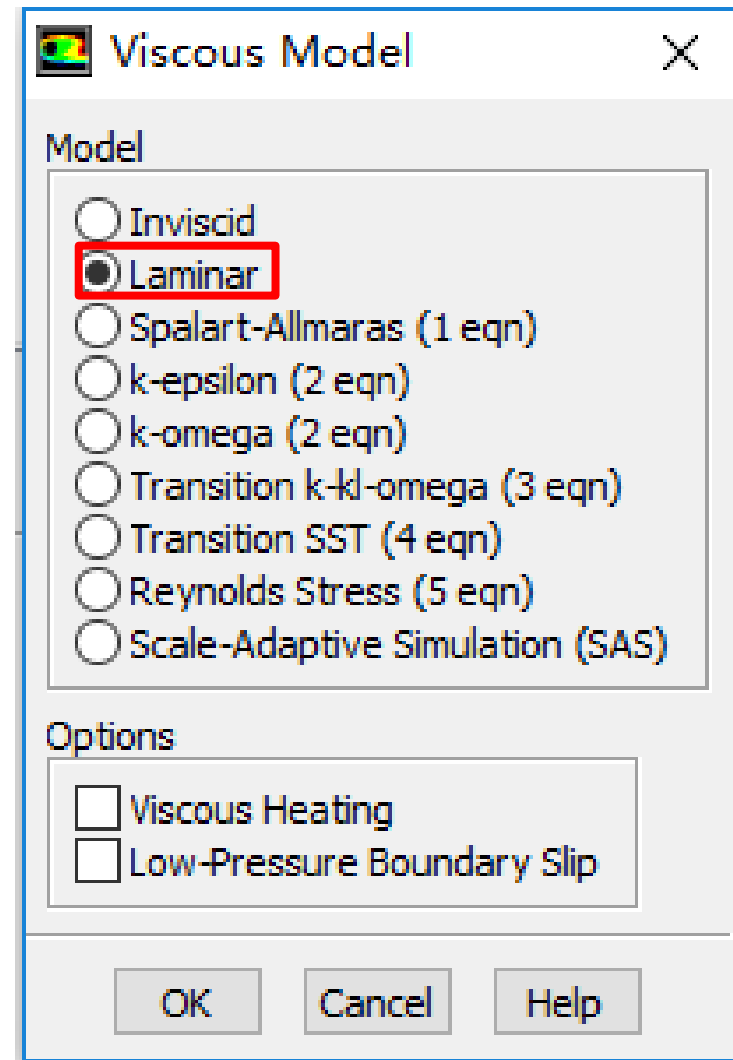
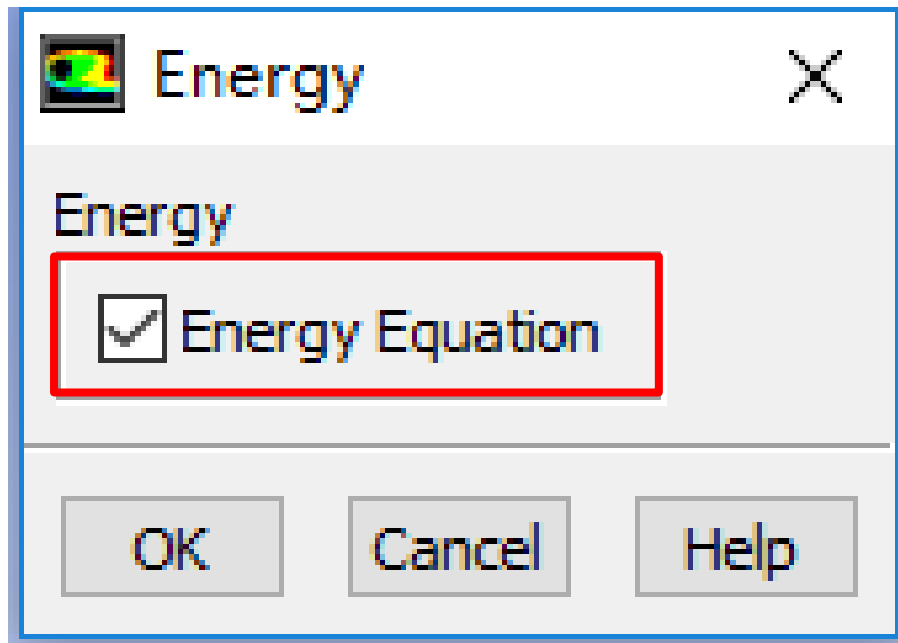
- You also can scale the domain size use “Convert Units” or “Specify Scaling Factors” command.

Remark: Fluent thought you create the mesh in units of m. However, if your mesh is created in a different unit, such as cm, you **must** use **Convert Units** Command to change the mesh into the right size. The values will be multiplied by the Scaling Factor.

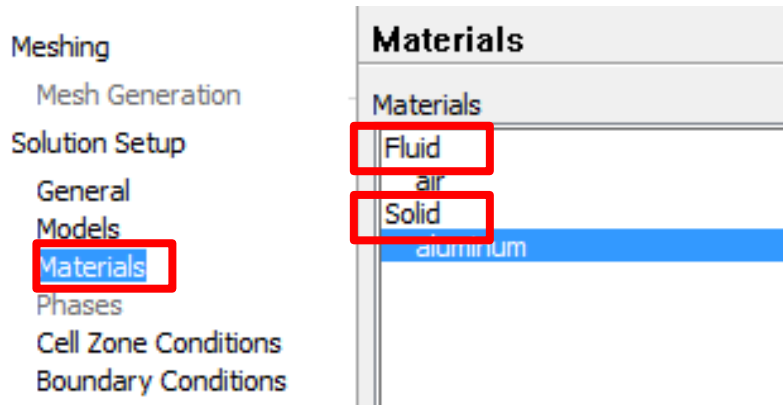


Step 3: Choose the physicochemical model

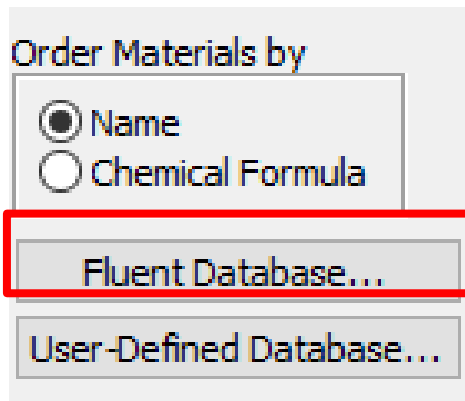
Based on the governing equations you are going to solve, select the related model in Fluent.



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

The image shows the 'Cell Zone Conditions' dialog box in ANSYS Fluent. The 'Zone' list contains 'inner'. The 'Phase' is set to 'mixture' and the 'Type' is set to 'fluid'. A red box highlights the 'Type' dropdown menu. A blue box highlights the 'Edit...' button. A callout box shows a zoomed-in view of the 'Type' dropdown menu, which is set to 'fluid'. Another callout box shows a zoomed-in view of the 'Fluid' properties dialog box, which is open for the 'inner' zone. The 'Zone Name' is 'inner' and the 'Material Name' is 'air'. The 'Edit...' button is also highlighted in this callout. The 'Fluid' properties dialog box includes checkboxes for 'Frame Motion', 'Source Terms', 'Mesh Motion', 'Fixed Values', and 'Porous Zone'. The 'Reference Frame' tab is selected, and the message 'This page is not applicable under current settings.' is displayed.

Cell Zone Conditions

Zone

inner

Phase

mixture

Type

fluid

Edit...

Copy...

Profiles...

Parameters...

Operating Conditions...

Display Mesh...

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...

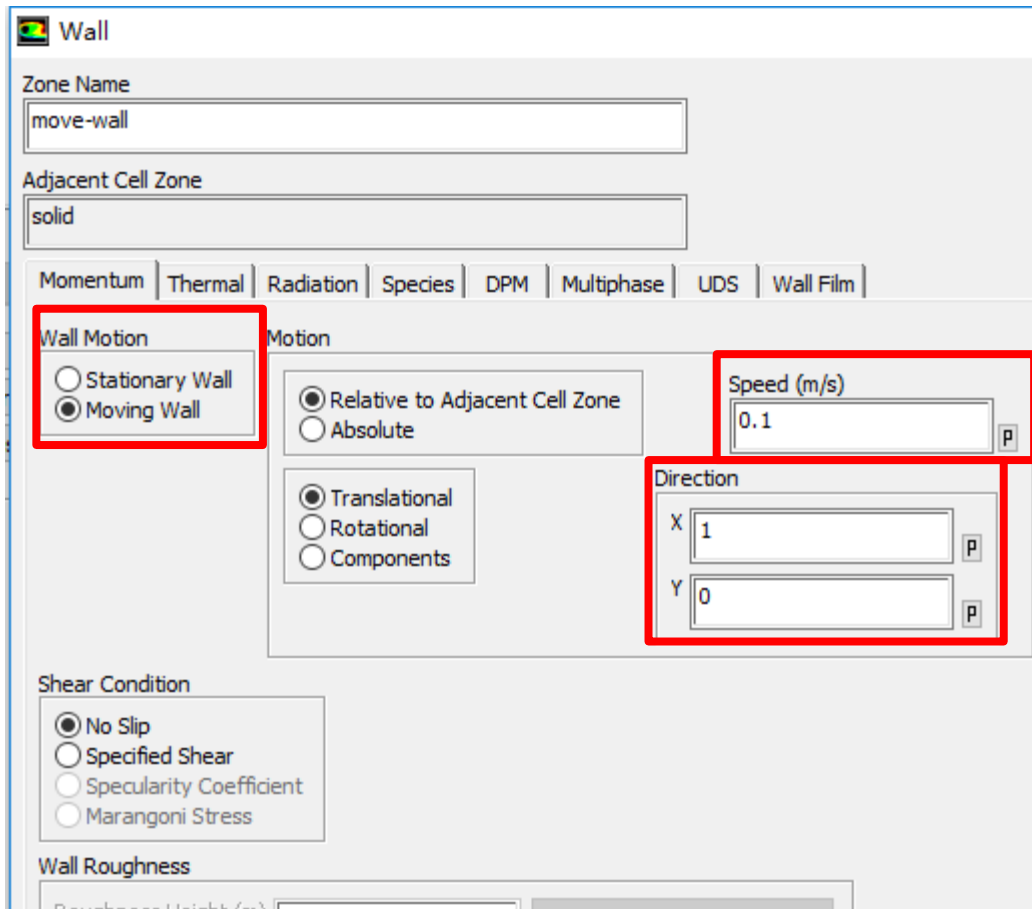
Periodic Conditions...

The bottom wall is not moving and its temperature is 100°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.



The image shows the ANSYS Fluent Wall boundary condition panel. The "Zone Name" is "move-wall" and the "Adjacent Cell Zone" is "solid". The "Momentum" tab is selected. In the "Wall Motion" section, the "Moving Wall" radio button is selected. In the "Motion" section, the "Relative to Adjacent Cell Zone" radio button is selected, and the "Speed (m/s)" is set to "0.1". In the "Direction" section, the "X" component is set to "1" and the "Y" component is set to "0". The "Shear Condition" section has the "No Slip" radio button selected. The "Wall Roughness" section is partially visible at the bottom.

Wall

Zone Name
move-wall

Adjacent Cell Zone
solid

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

Wall Motion

☐ Stationary Wall
☒ Moving Wall

Motion

☒ Relative to Adjacent Cell Zone
☐ Absolute

Speed (m/s)
0.1

Direction

X 1
Y 0

Shear Condition

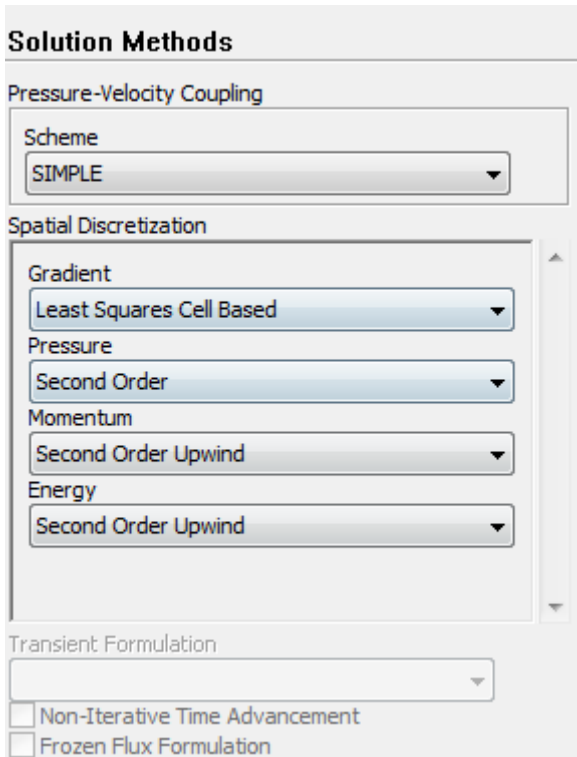
☒ No Slip
☐ Specified Shear
☐ Specularity Coefficient
☐ Marangoni Stress

Wall Roughness

Roughness Height (m)

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

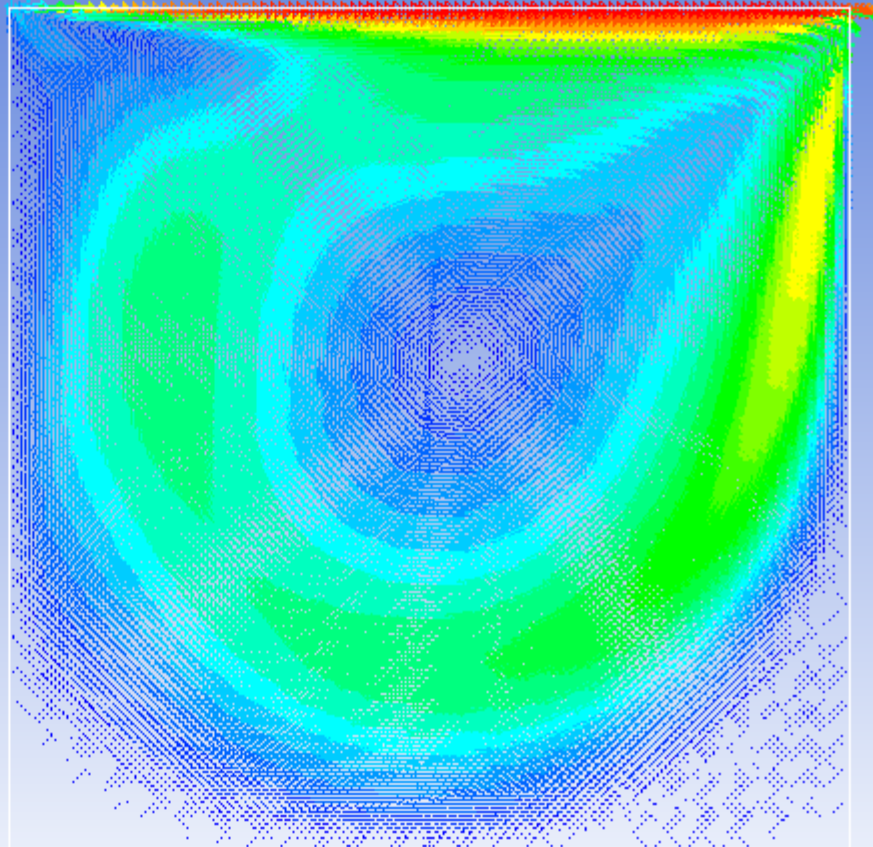
Step 10: Post-processing results

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



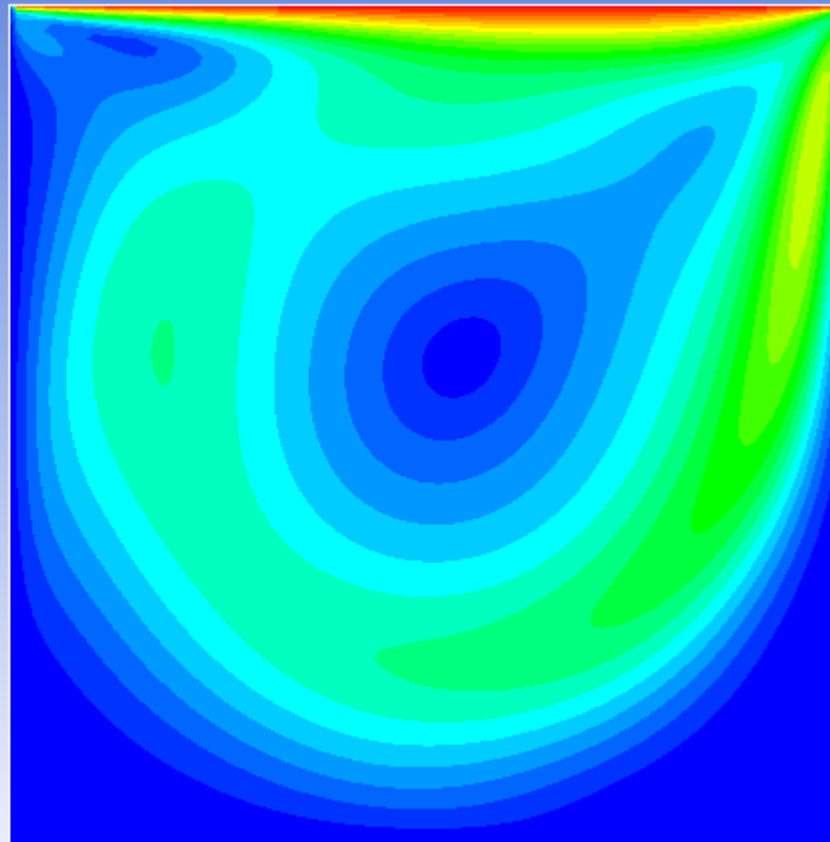
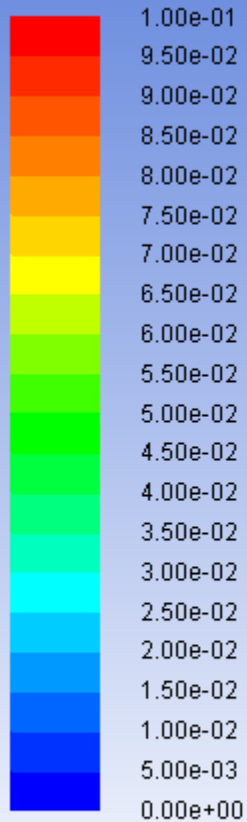
Velocity Vectors Colored By Velocity Magnitude (m/s)

ANSYS:

22/24

Velocity magnitude

1: Contours of Velocity Magn



Contours of Velocity Magnitude (m/s)

Temperature

