Numerical Heat Transfer

Chapter 13 Application examples of Fluent for flow and heat transfer problem

Instructor Li Chen, WenQuan Tao

CFD-NHT-EHT Center
Key Laboratory of Thermo-Fluid Science & Engineering
Xi'an Jiaotong University

Xi'an, 2018-Dec.-12

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：陈黎，陶文铨

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
2018年12月12日，西安

Class A

13. A1 Flow and heat transfer in microchannels with secondary channels (具有二次通道的微通道中流动换热)

13. A2 Flow and heat transfer in porous media (多孔介质流动换热)

13. A3 Multiphase flow using Volume of Fraction method (多相流VOF方法模拟)

For each example, the general content of the lecture is as follows:

1：Using slides to explain in detail the general 10 steps for Fluent simulation!（PPT讲解）

1. Read mesh **2. scale domain**
3. Choose model **4.define material**
5. define zone condition **6. define boundary condition**
7. Solution **8. Initialization**
9. Run the simulation **10. Post-processing**

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

3：Drawing inferences for each example (举一反三）

13_A1: Flow and heat transfer in microchannels with secondary channels

Background:

Because of the integration(集成化) of electron component (电子元件), the heat flux of a EC greatly increases, even reaches $MW \cdot m^{-2}$ order of magnitude.

Traditional cooling techniques cannot meet the cooling demand of such high heat flux.

Microchannel is proposed for this purpose.

What is "Microscale" ?

1. The continuum assumption (连续介质假设) does not stand.

Depending on Kn number, the flow may be in continuum, slip , transition or even free molecular flow region.

The NS equation should be modified or even is not applicable.

2. The relative importance of affecting factors changes.

Fluid flow is controlled by body forces and surface forces

body forces: ~m^3 surface forces: ~m^2

surface forces/body forces: ~m^{-1}; surface force becomes stronger as length scale decreases.

Known

Steady single phase fluid flow and heat transfer of water in a copper microchannel. There are secondary channels in the domain, as displayed in Fig. 1.

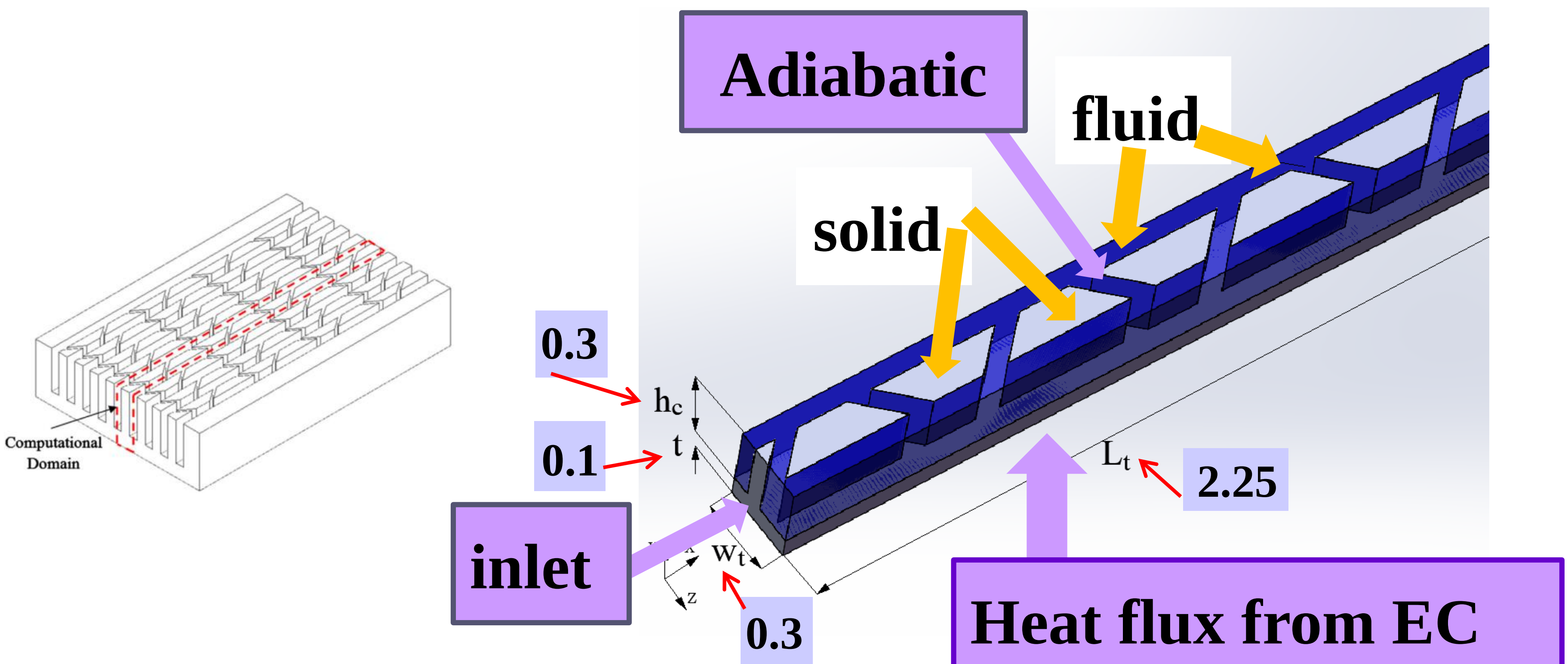
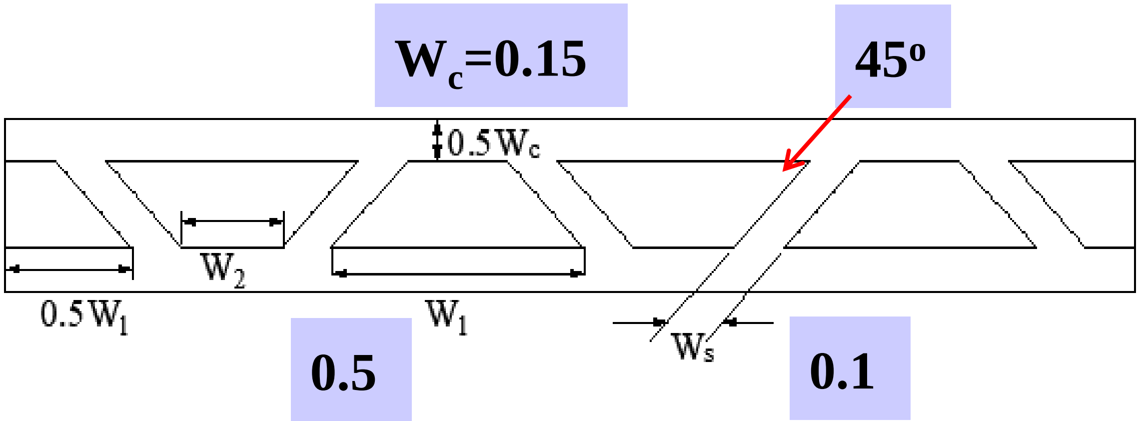


Fig.1 Computational domain

Top view



Boundary conditions

	Heat transfer	Fluid flow
Inlet	F:300K; S: adiabatic	F:Velocity inlet; S:wall
Outlet	adiabatic	F:1atm; S:wall
Bottom	Heat flux($1\times10^{6}\ W\cdot m^{-2}$)	Wall
Up	Adiabatic	Wall
Side	Symmetry	Symmetry

- **Find:** average Nusselt number (Nu_{ave}), average temperature of bottom surface (T_b) and resistance factor (f) at different *Re* numbers (100, 200, 300, 400, 500).
- **Assumptions:**

(1) When *Kn* is less than 10^{-3}, N-S Eqs still can be used;

(2) laminar, incompressible, Newtonian fluid;

(3) Physical parameters are constant;

(4) The gravity and viscous dissipation can be ignored;

(5) The thermal radiation can be ignored.

Remark: construct the reasonable physical model and write down the right governing equation, BC and IC is the first and most important step before using Fluent.

Fluent is just a tool for solving above problem !

Background of NHT helps you better use the tool.

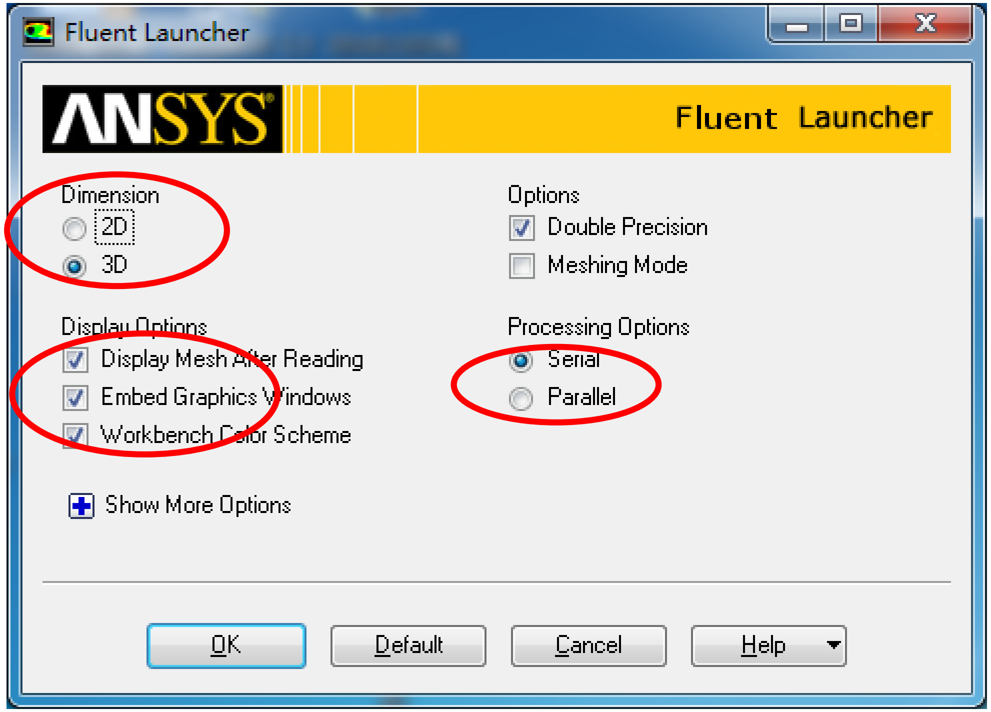
- **Governing equations:**

Continuum equation $\nabla \boldsymbol{u} = 0$

Momentum equation $\nabla\left(\rho \boldsymbol{u}\boldsymbol{u}\right) = -\nabla p + \eta \nabla^2 \boldsymbol{u}$

Energy equation $\nabla(\rho c_p \boldsymbol{u} T_{\mathrm{f}}) = \nabla \lambda_{\mathrm{f}} \nabla T_{\mathrm{f}} \quad 0 = \nabla \lambda_{\mathrm{s}} \nabla T_s$

Start the Fluent software



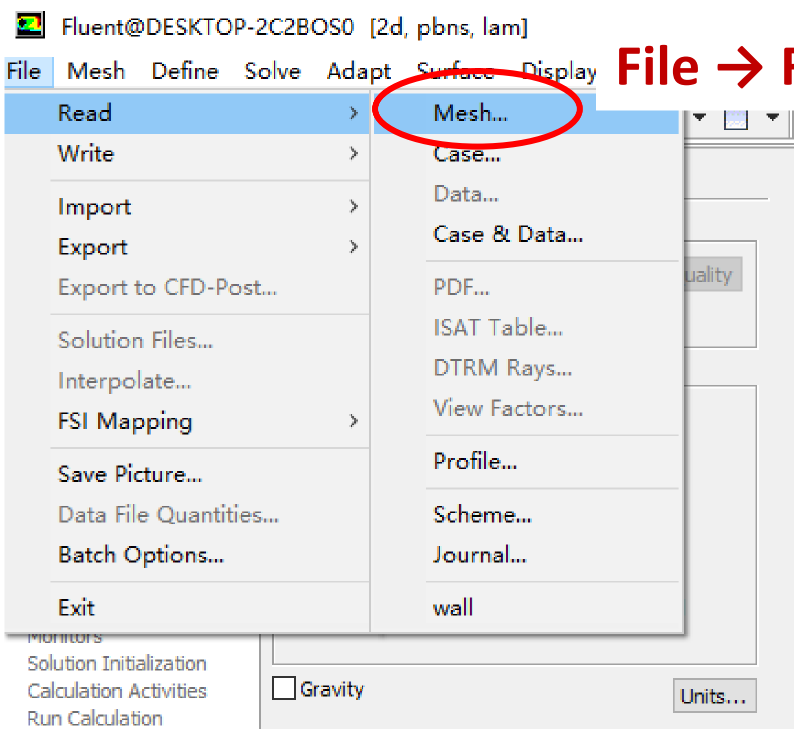
1. Choose 3-Dimension
2. Choose display options
3. Choose Serial processing option or parallel to choose different number of processes

Note: Double precision or Single precision

For most cases the single precision version of Fluent is sufficient. For example, 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) “.msh”
- This step is similar to the Grid subroutine (UGRID, Setup1) in our general teaching code.



File → Read→Mesh

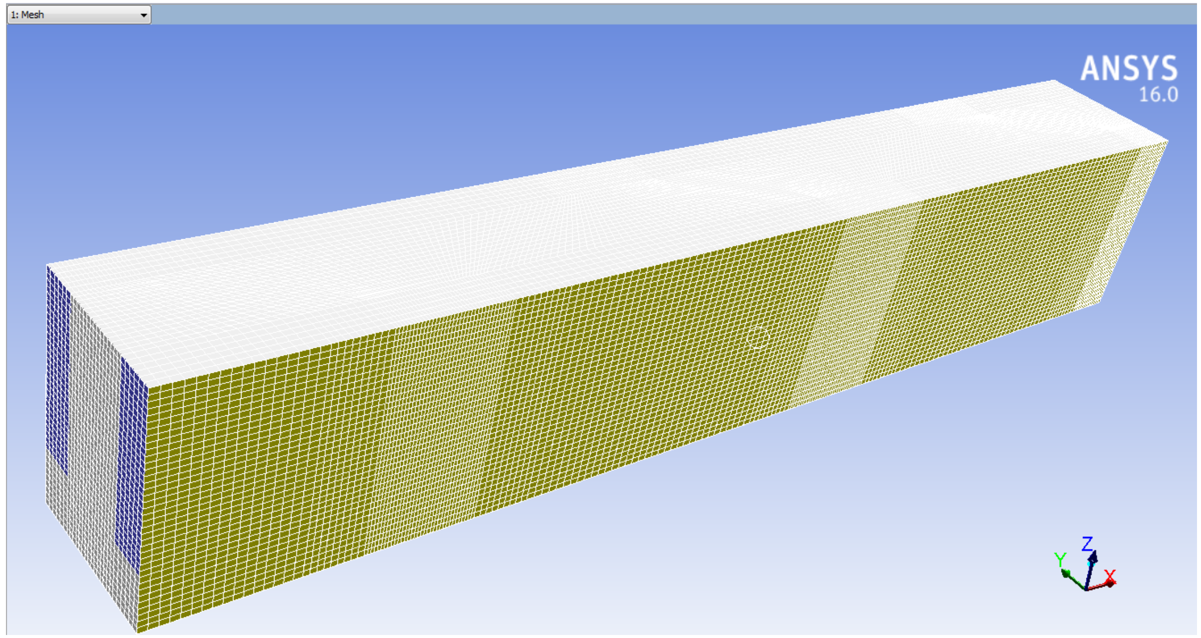
```
Building...
     mesh
Slitting wall zone 29 into a coupled wall.
     materials,
     interface,
     domains,
     zones,
        inter_surface_sf-shadow
        down_wall_s
        up_wall_s
        up_wall_f
        inter_surface_sf
        wall_left_f
        wall_left_s
        wall_right_f
        wall_right_s
        out_s
        out_f
        in_s
        in_f
        int_fluid
        int_solid
        fluid
        solid
Done.

Preparing mesh for display...
Done.
```

Step 1: Read and check the mesh

Mesh→Check

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



- **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

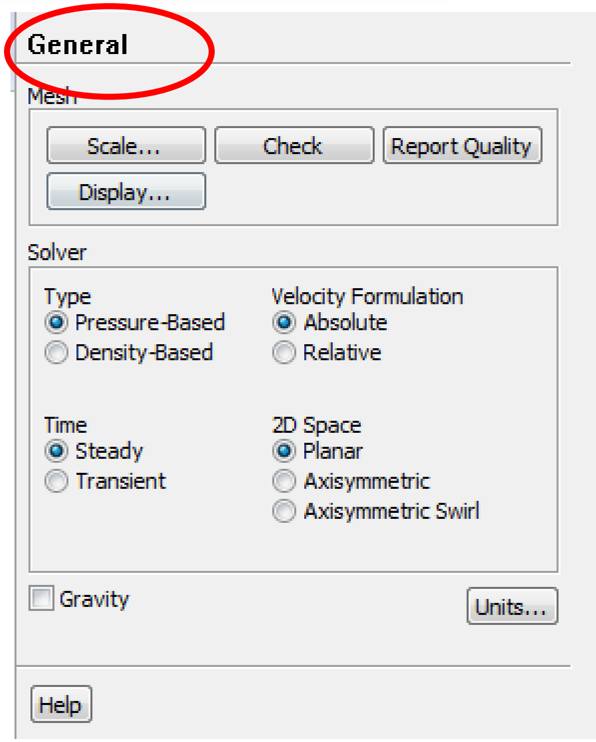
WARNING: Mesh check failed.

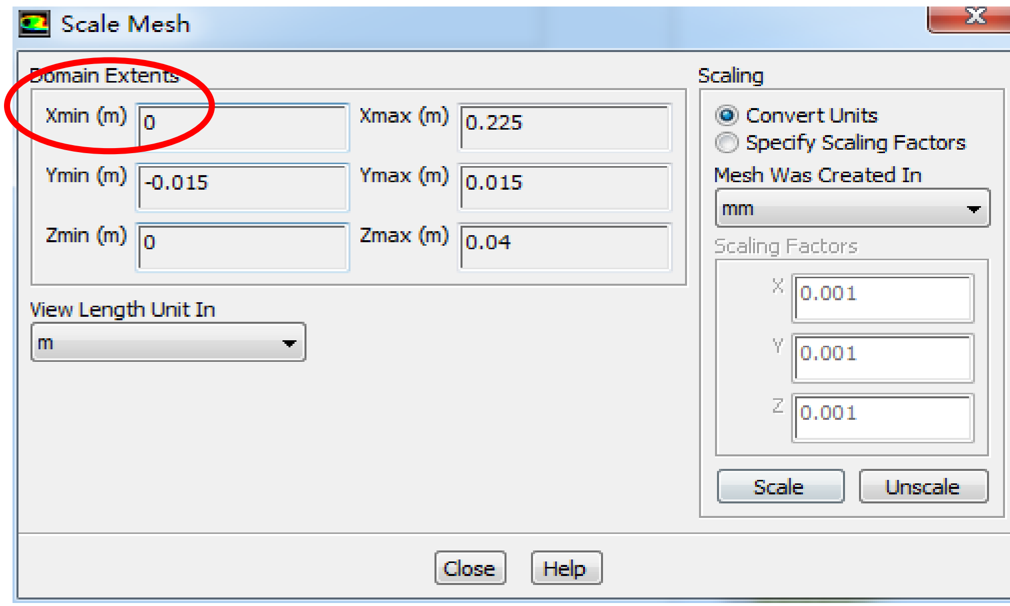
WARNING: The mesh contains high aspect ratio quadrilateral, hexahedral, or polyhedral cells.

Step 2: Scale the domain size (缩放)

General→Scale

Make sure the unit is right.

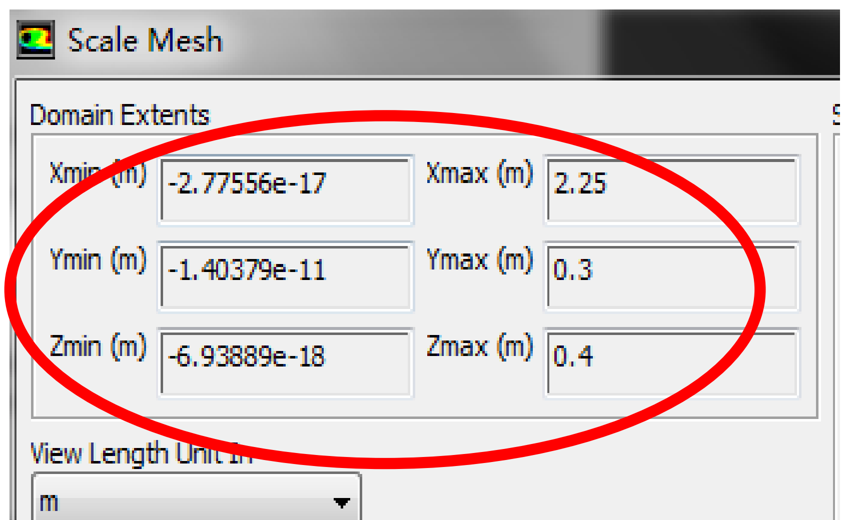


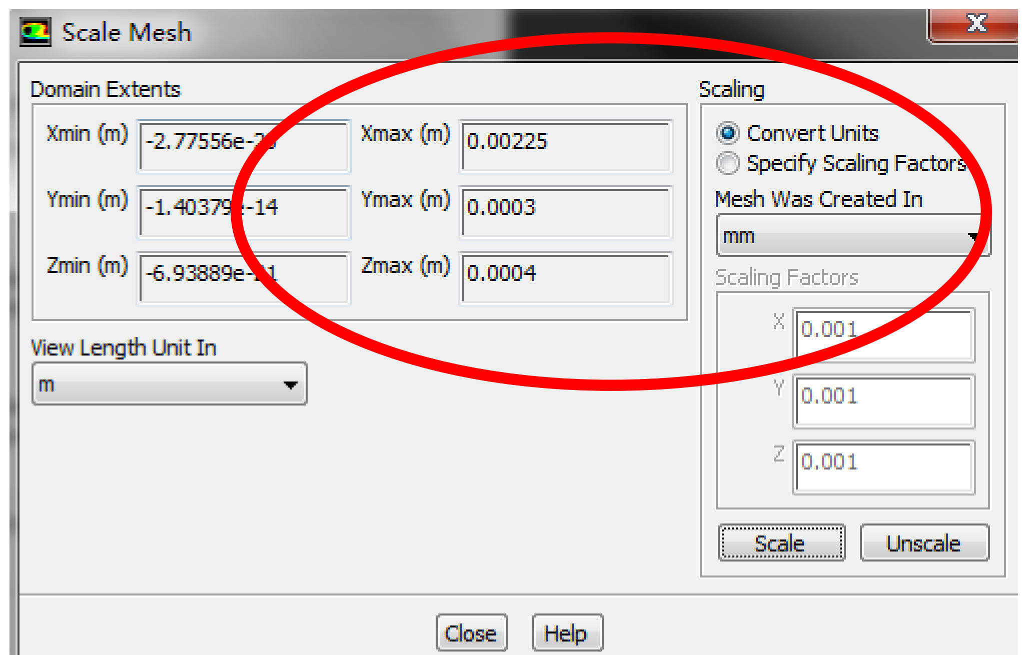


- **You can scale the domain size use "Convert Units" or " Specify Scaling Factors" command.**

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 **scale the mesh into the right size**. The values will be multiplied by the Scaling Factor.

ICEM: 1 mm -> Fluent: 1m -> Scale: cm, factor: 1/1000





Step 3: Choose the physicochemical model

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

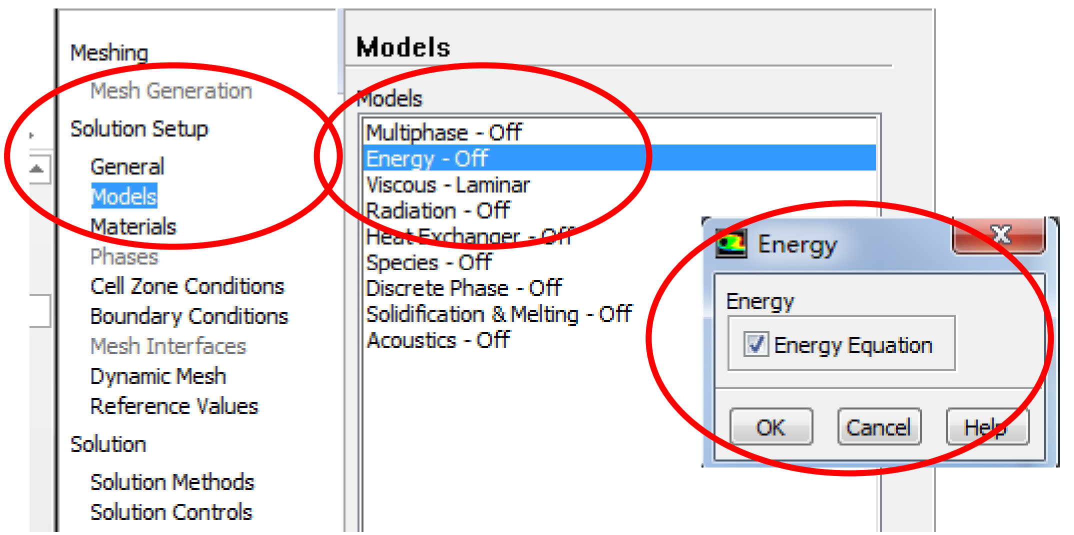
Remark: Understand the problem you are going to solve, and write down the right governing equations is the first and most important step for numerical simulation. Without background of "Fluid mechanics", "Heat Transfer" and "Numerical heat transfer", it is hard to complete this step for fluid flow and heat transfer problem.

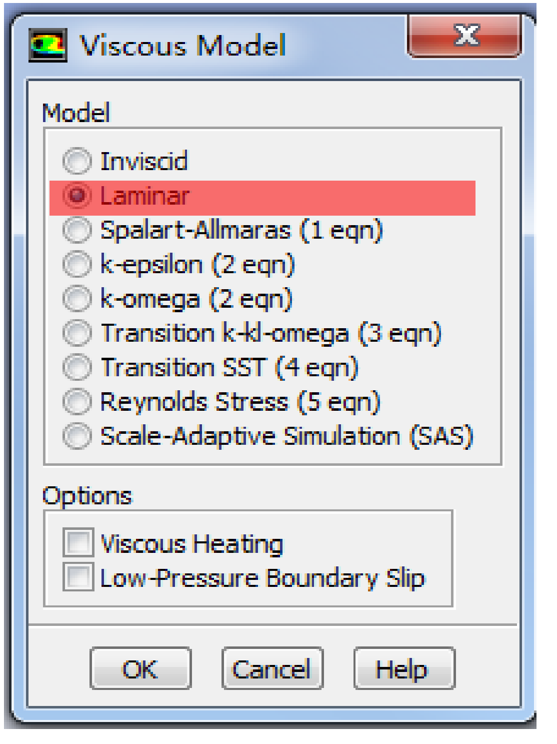
Fluent is just a tool!

Step 3: Choose the physicochemical model

To select the model, the command is as follows:

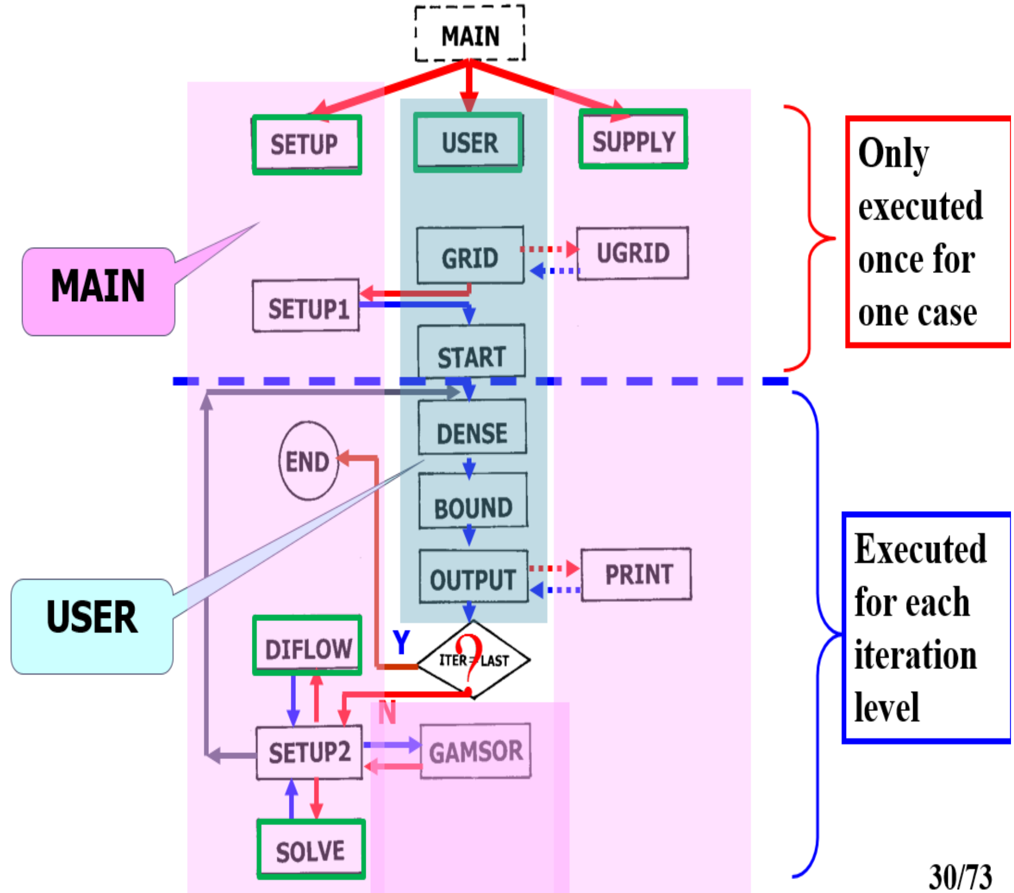
Solution Setup→Model





Remark: In our general teaching code

In SETUP2, Visit NF from 1 to NFMAX in order; **If LSOLVE(NF)=.T. , this variable is solved; Similarly in PRINT SUBROUTINE NF is visited form 1 to NFX4(=14) in order , as long as LPRINT(NF)=.T., the variable is printed out.**



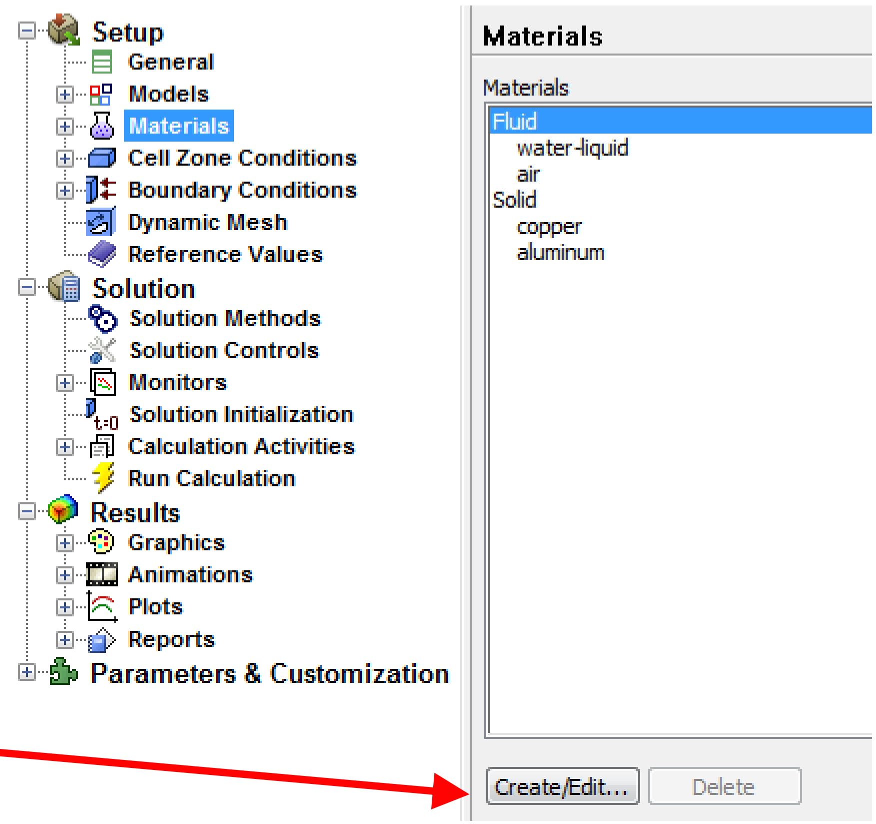
Step 4: Define the material properties

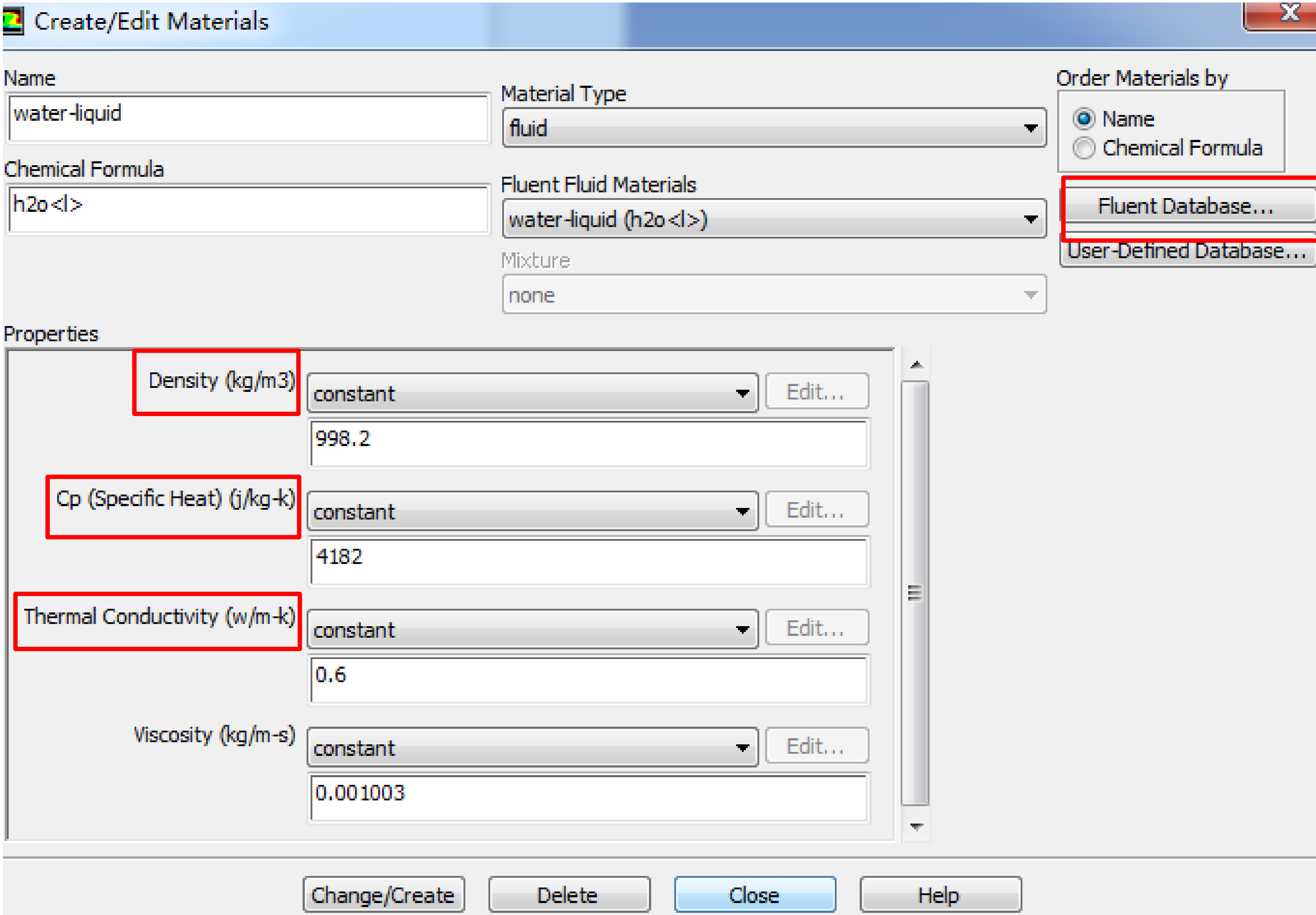
Define the properties required for modeling! For fluid flow and heat transfer problem studied here, ρ, c_p and λ should be defined.

Solution Setup→Materials

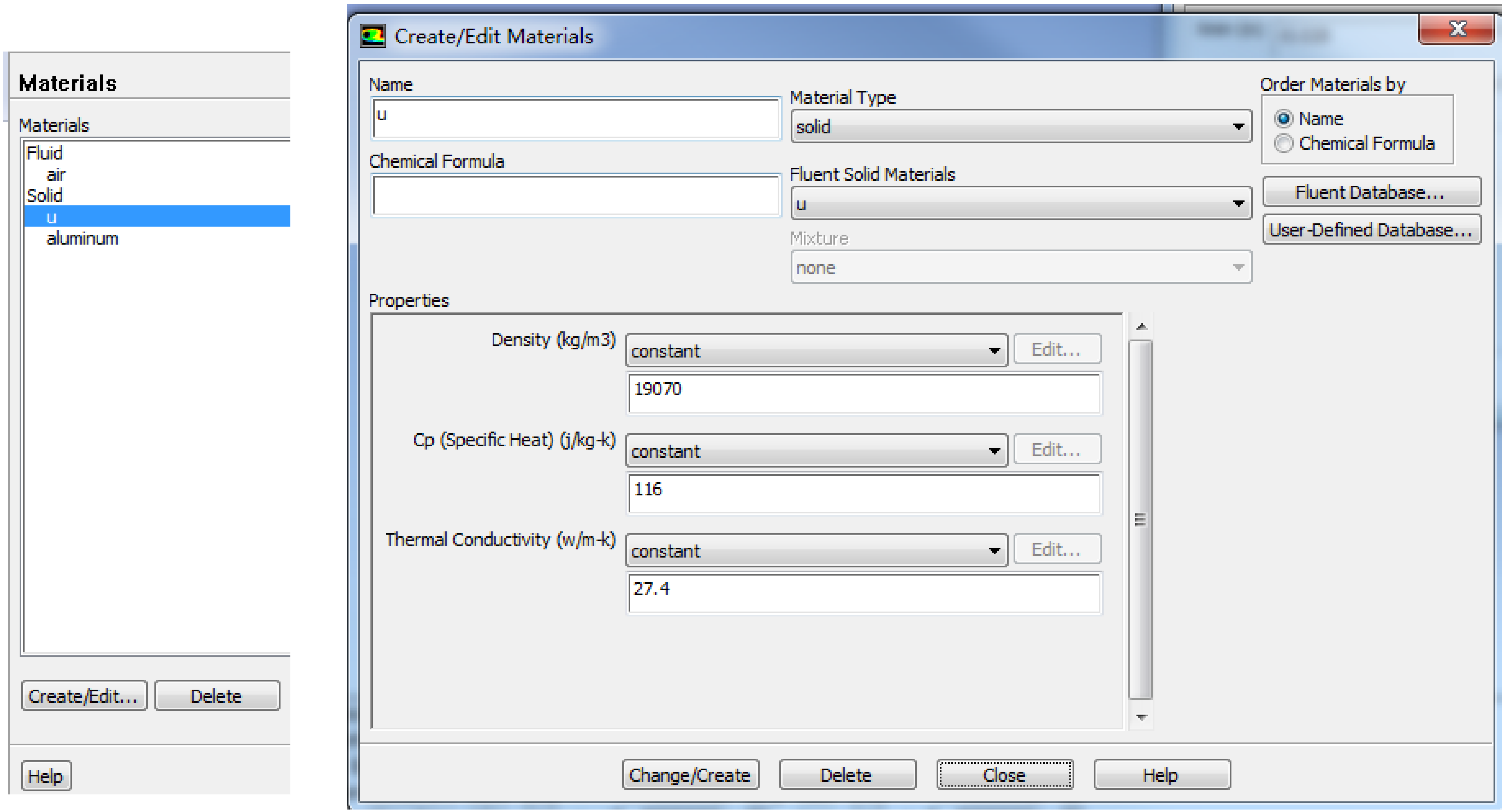
In Fluent, the default fluid is air and the default solid is Al.

Click the Create/Edit button to add copper and liquid water in our case.




Create/Edit Materials
Name
water-liquid
Chemical Formula
h2o<l>
Material Type
fluid
Fluent Fluid Materials
water-liquid (h2o<l>)
Mixture
none
Order Materials by
Name
Chemical Formula
Fluent Database...
User-Defined Database...
Properties
Density (kg/m3)
constant
Edit...
998.2
Cp (Specific Heat) (j/kg-k)
constant
Edit...
4182
Thermal Conductivity (w/m-k)
constant
Edit...
0.6
Viscosity (kg/m-s)
constant
Edit...
0.001003
Change/Create
Delete
Close
Help

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

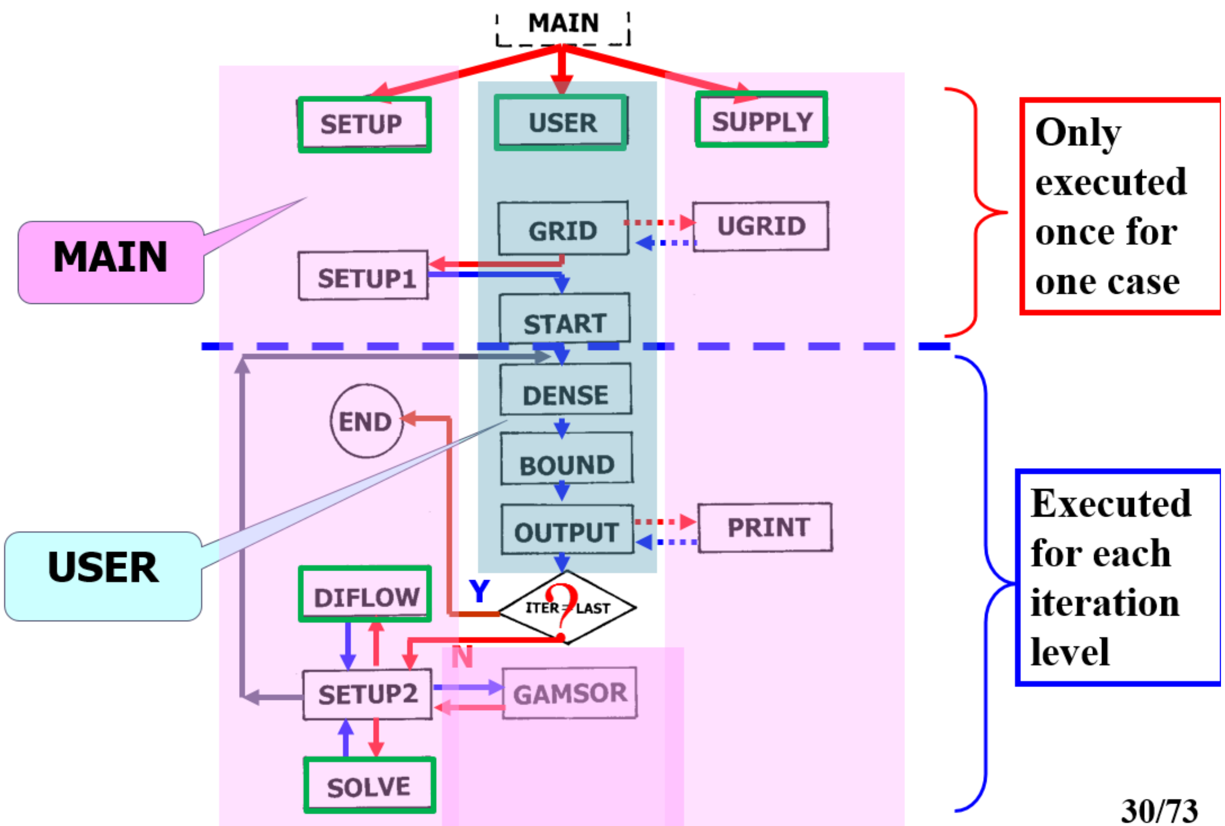


Our general Code:

12. GAMSOR

(1) Determine Γ_{ϕ} for different variables:

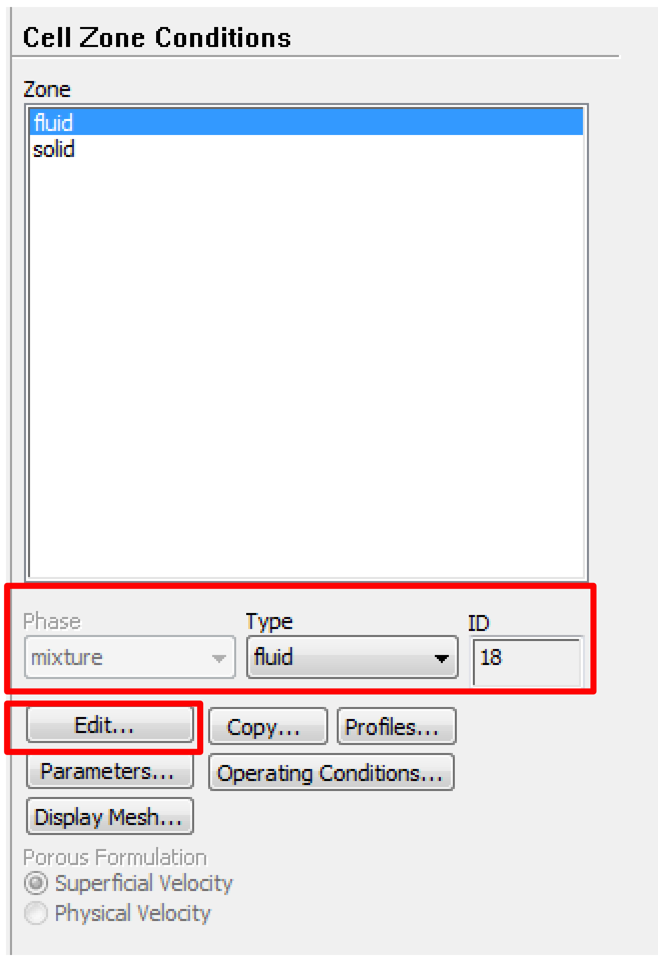
$$u\ ,v\ -\eta \quad ;T-\lambda$$



30/73

Step 5: Define zone condition

Solution Setup→Cell Zone Condition



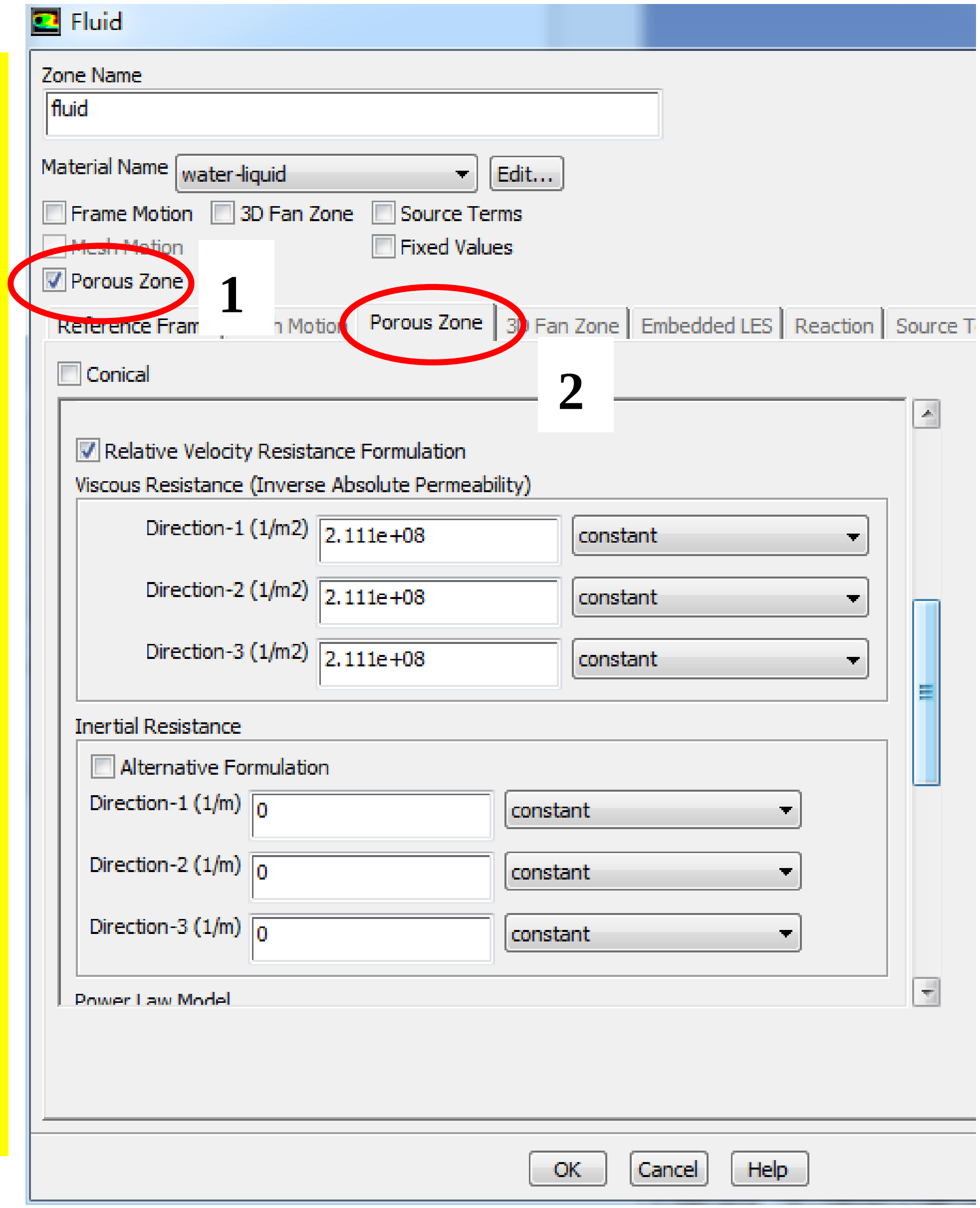
Each zone has its ID.

Each zone should be assigned a type, either fluid or solid.

Phase is not activated here. It can be edited under other cases, for example multiphase（多相流）flow model is activated. See Example A3.

Click Edit to define the zone condition of each zone.

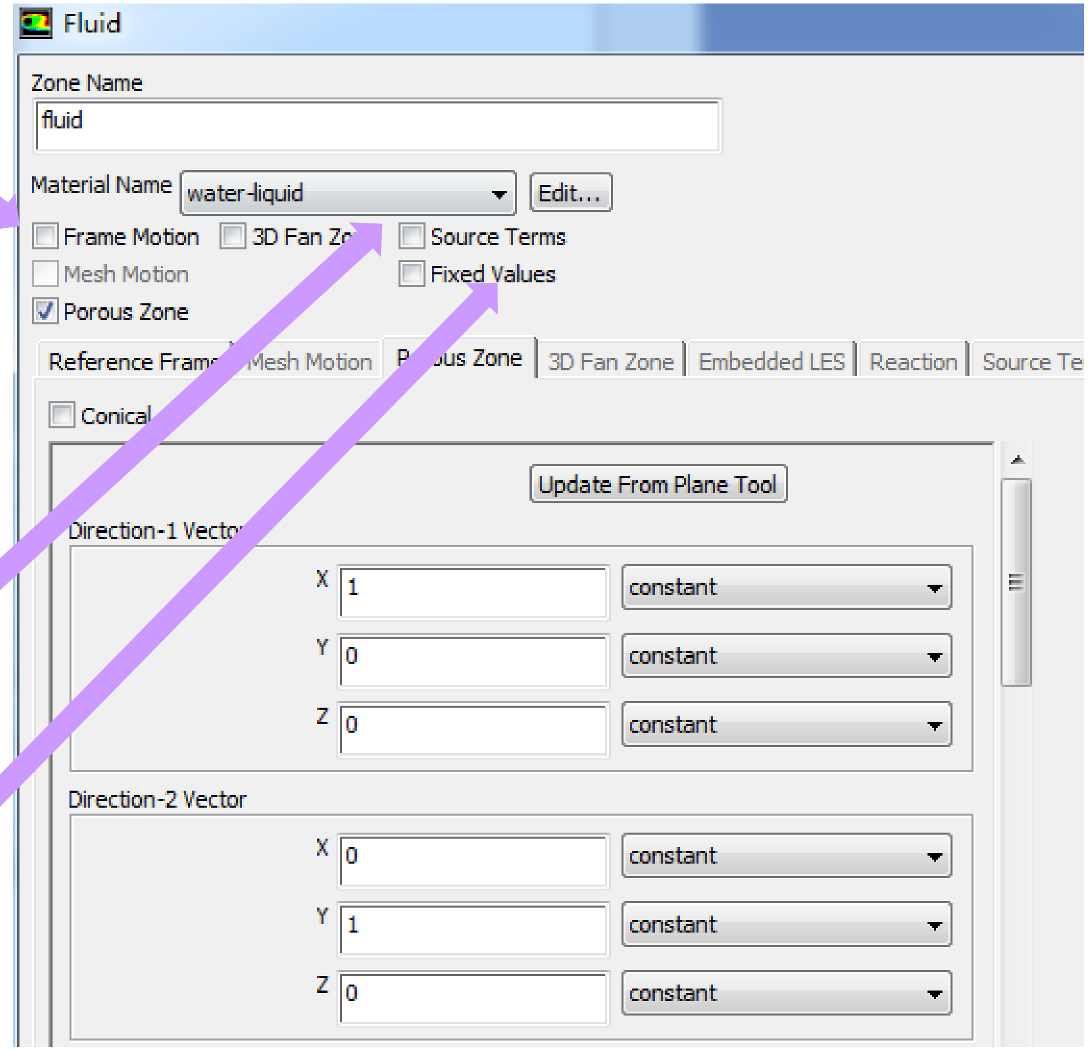
Porous media is treated as a type of fluid zone, in which parameters related to porous media should be given such as porosity, permeability (渗透率）, etc. We will discuss it in Example A2.



Frame motion and Mesh motion is used if the solid or the frame is moving.

Added in need as a constant value or by user defined with .c file compiled if you need.

If T of the zone is fixed, you can select the Fixed value button.



Step 6: Define the boundary condition

Boundary condition definition is one of the most important and difficult step during Fluent simulation. General boundary conditions in Fluent can be divided into two kinds:

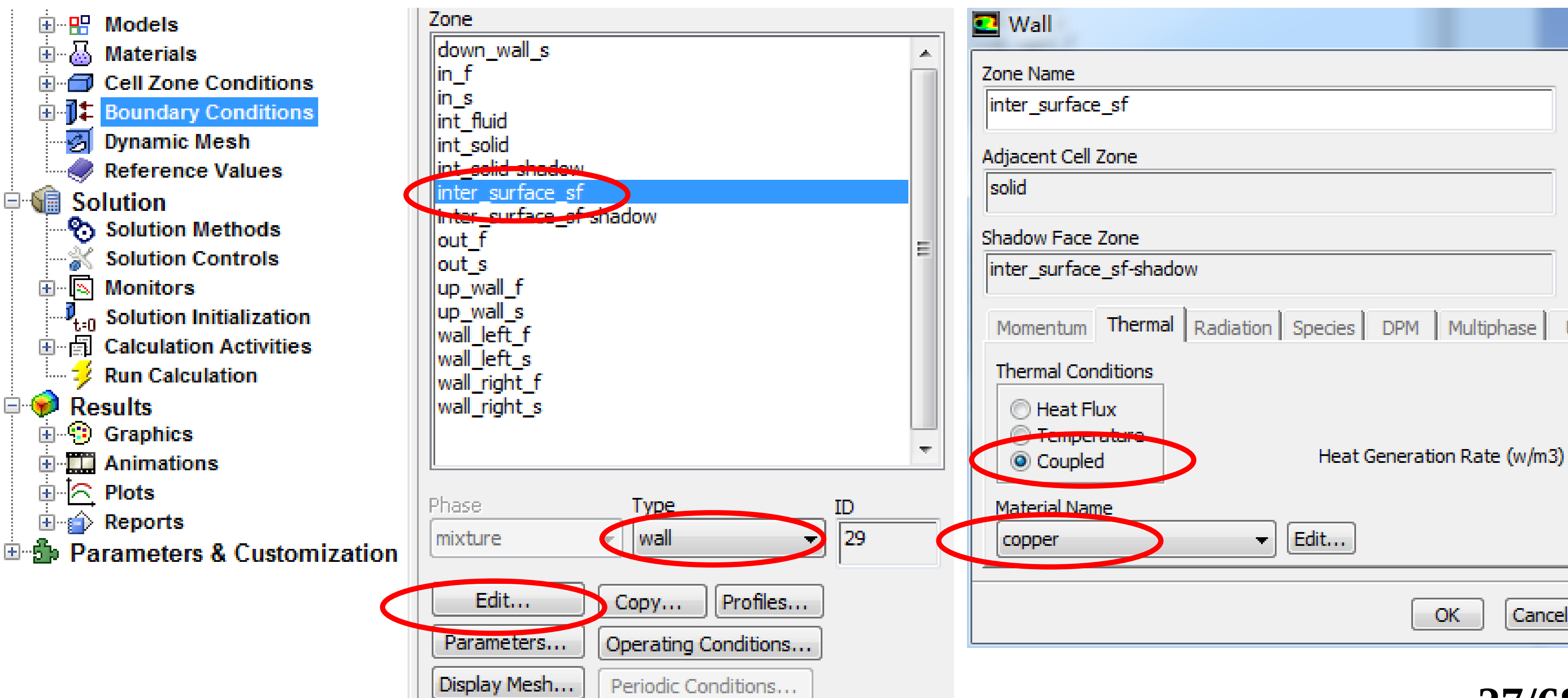
1. BC at inlet and outlet: pressure, velocity, mass flow rate, outflow…

2. BC at wall: wall, periodic, symmetric…

Remark: **Interior cell zone and interior interface will also shown in the BC Window.**

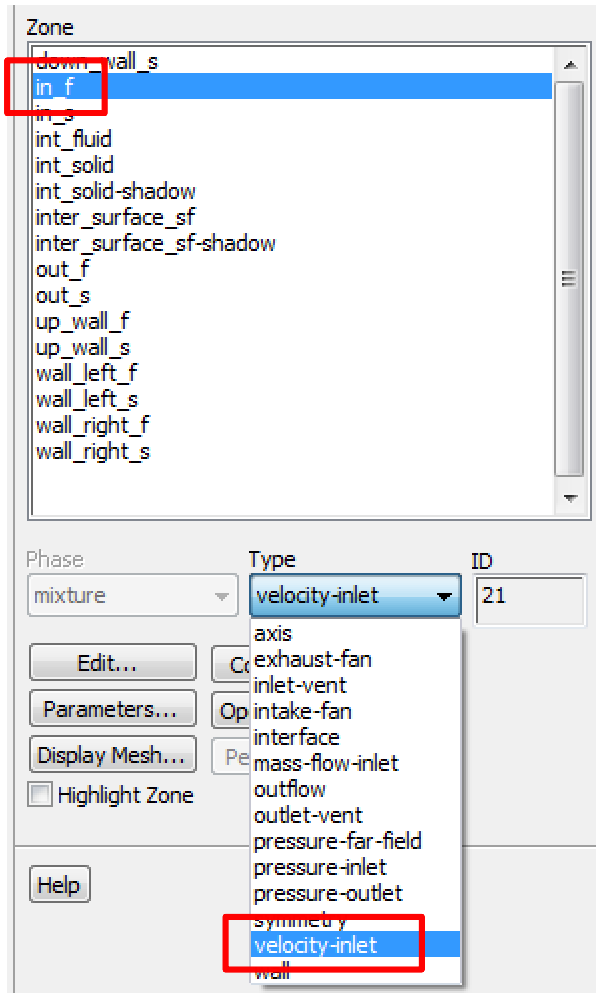
For example, inter_surface_sf: 29 is listed here. It is the interface between fluid and solid zones.

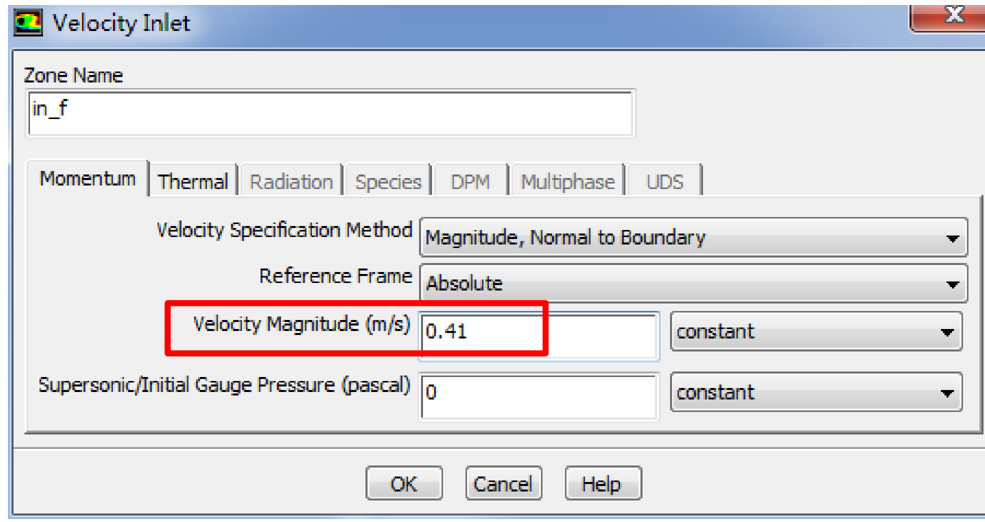
It is treated as coupled, conjugate condition (流固耦合)

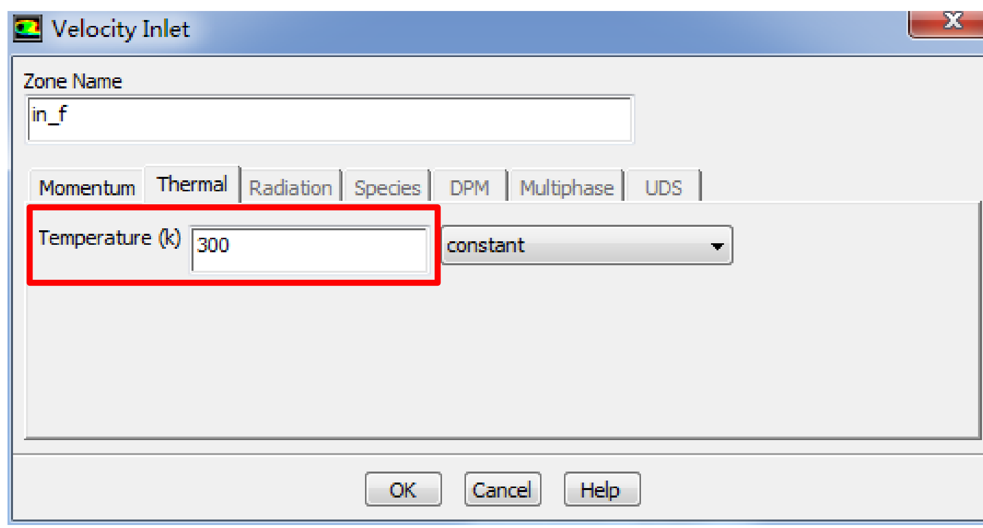


Other BCs are as follows:

For fluid inlet: velocity inlet

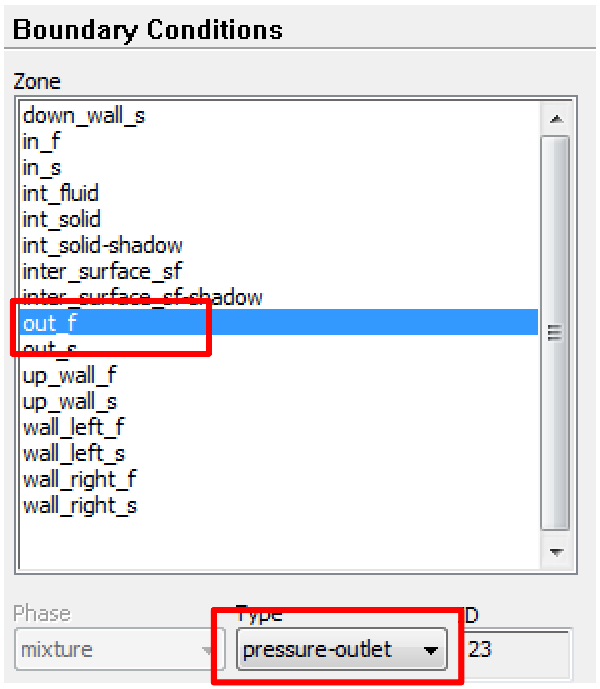


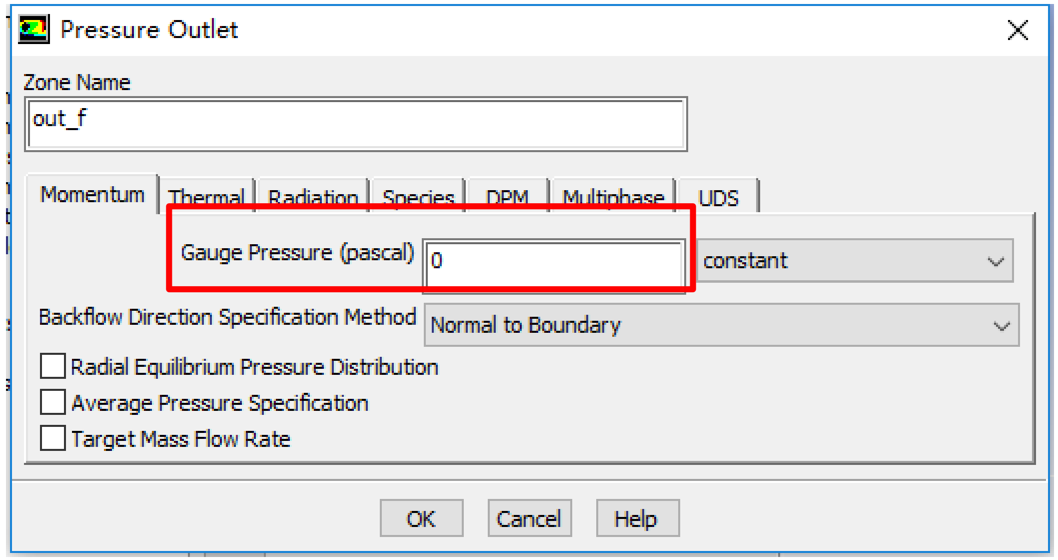


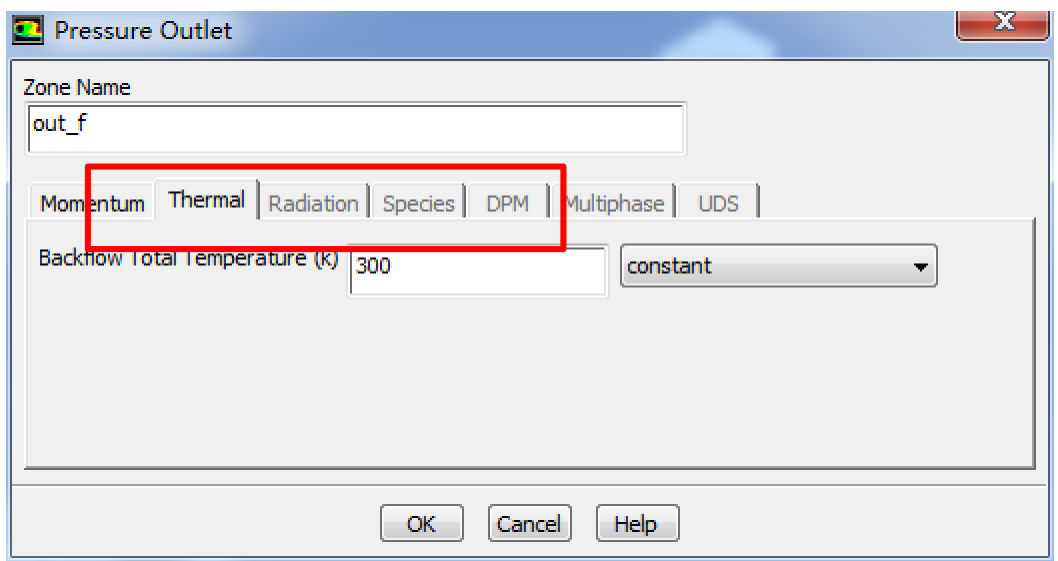


Other BCs are as follows:

For fluid outlet: pressure outlet







Seven kinds of Pressure in Fluent

1. Atmospheric pressure (大气压)

2. Gauge pressure (表压): the difference between the true pressure and the Atmospheric pressure.

3. Absolute pressure (真实压力): the true pressure

= Atmospheric pressure + Gauge pressure

4. Operating pressure (操作压力）： the same as the reference pressure （参考压力）in our teaching code

Pressure in Fluent

Absolute pressure (真实压力): the true pressure

= Reference Pressure + Relative Pressure

5. Static pressure (静压): the difference between true pressure and operating pressure.

The same as relative pressure.

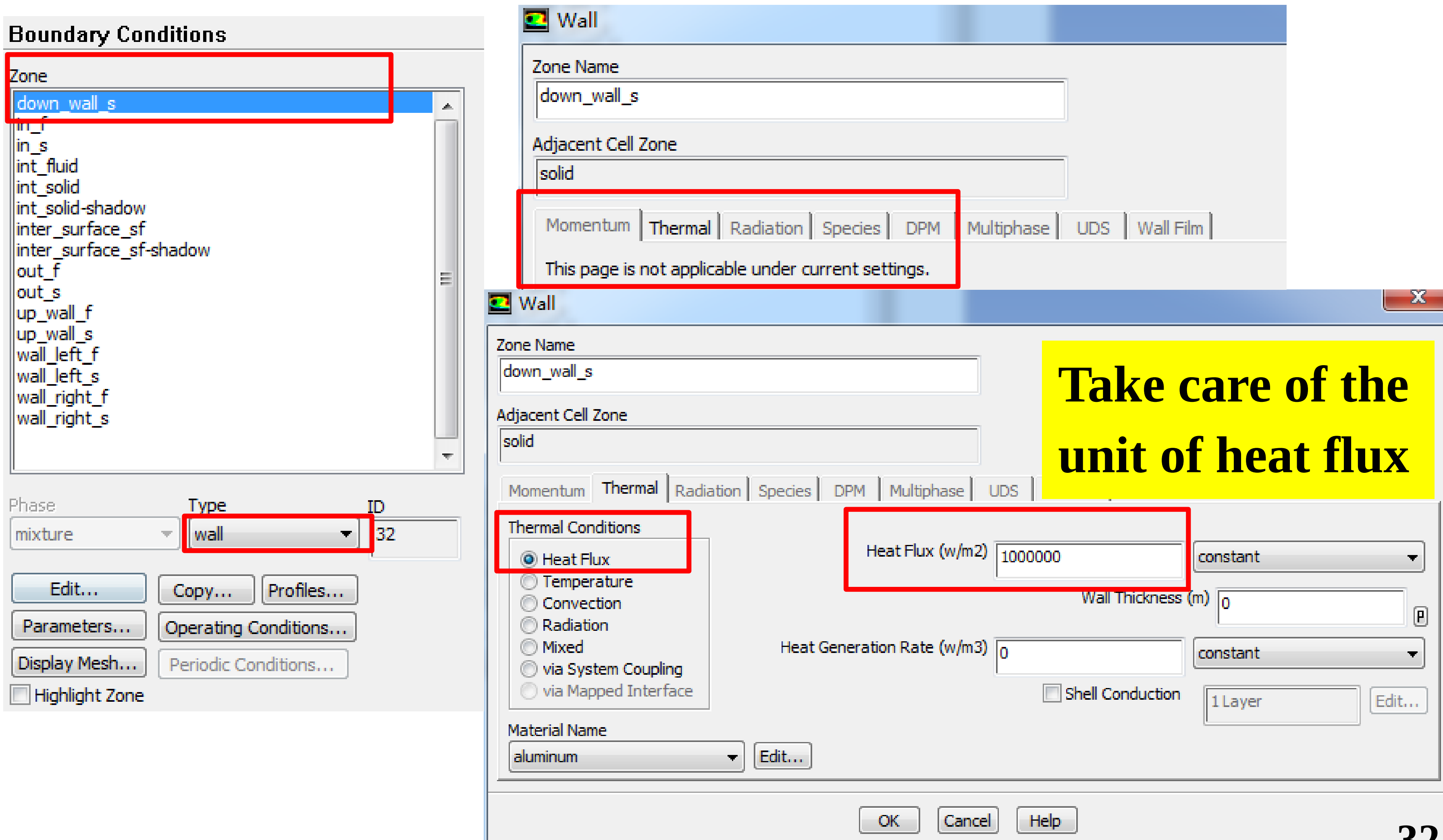
6. Dynamic pressure (动压): calculated by 0.5ρU^2 is related to the velocity.

7. Total pressure (动压):

= Static pressure + dynamic pressure

Other BCs are as follows:

For bottom surface: constant heat flux

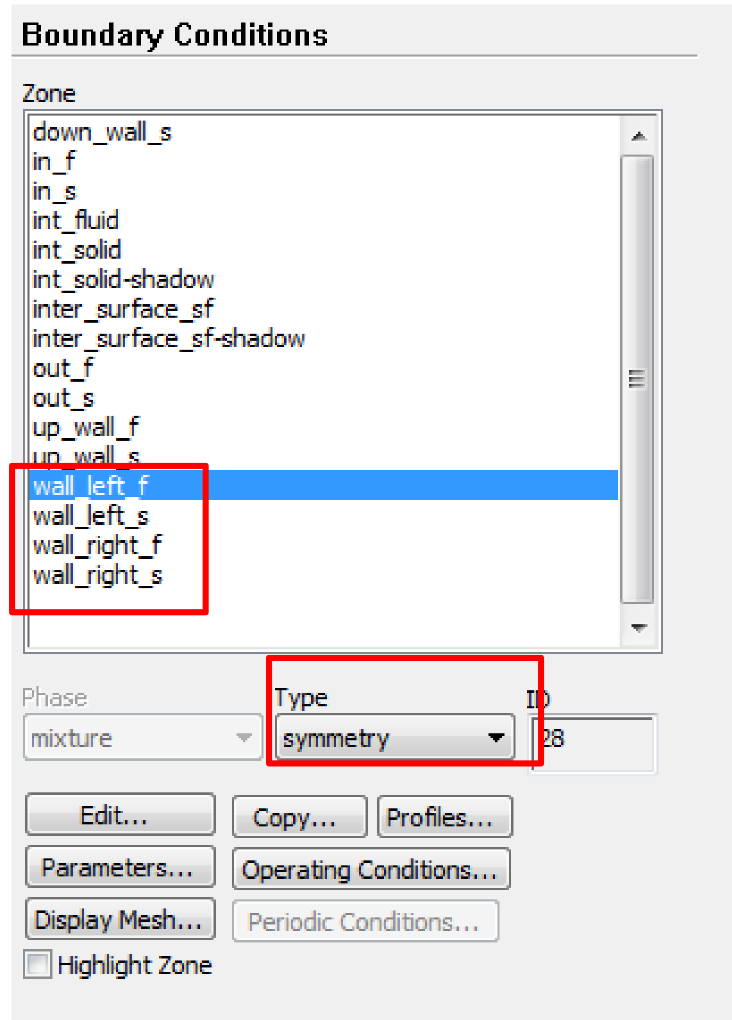


Take care of the unit of heat flux

Other BCs are as follows:

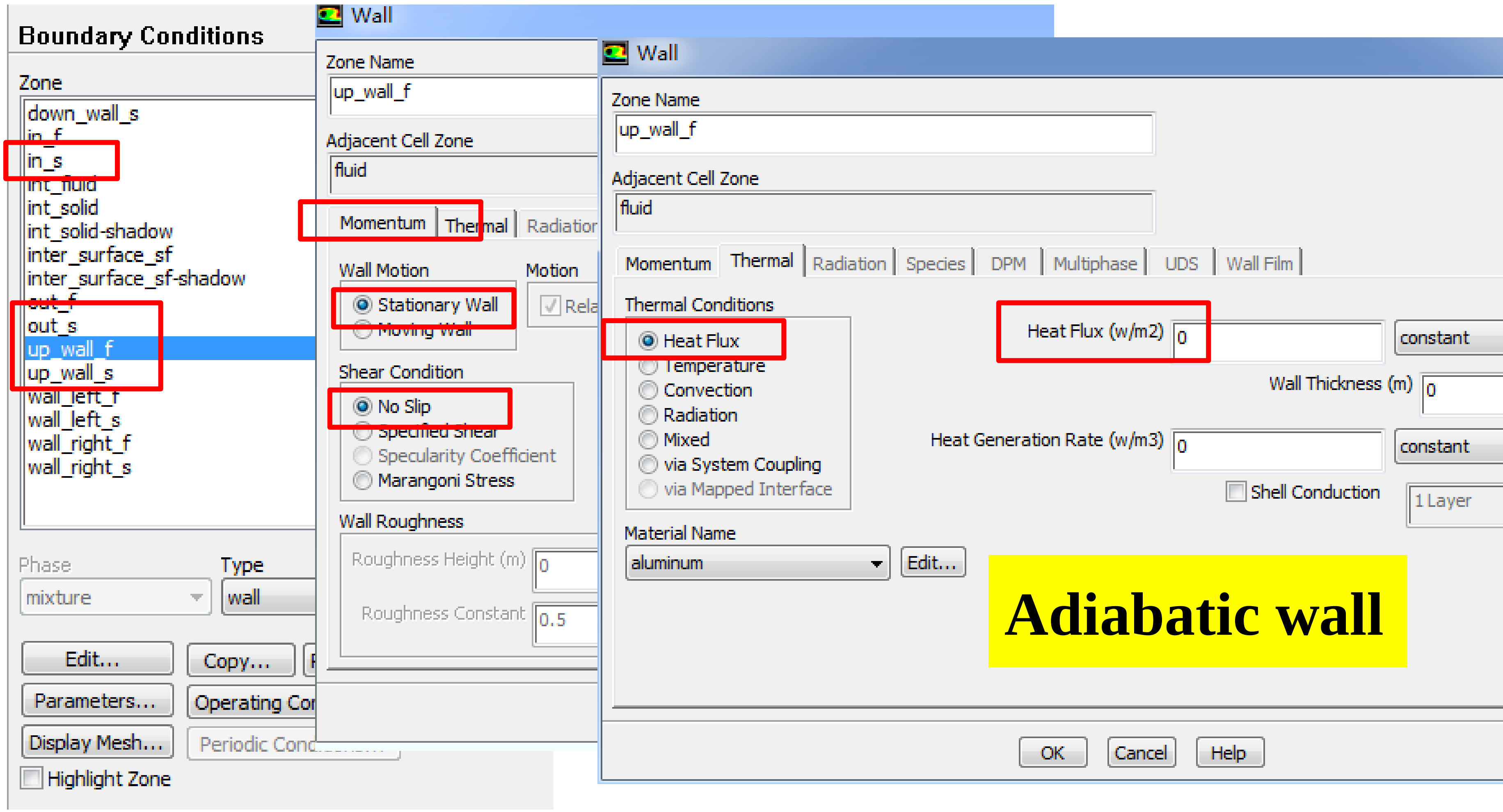
For left and right fluid surfaces: symmetry

The left and right boundary for solid and fluid are set as symmetry. Because the calculation domain is a typical part extracted from the total district, which can represent the heat transfer and fluid flow characteristics.



Other BCs are as follows:

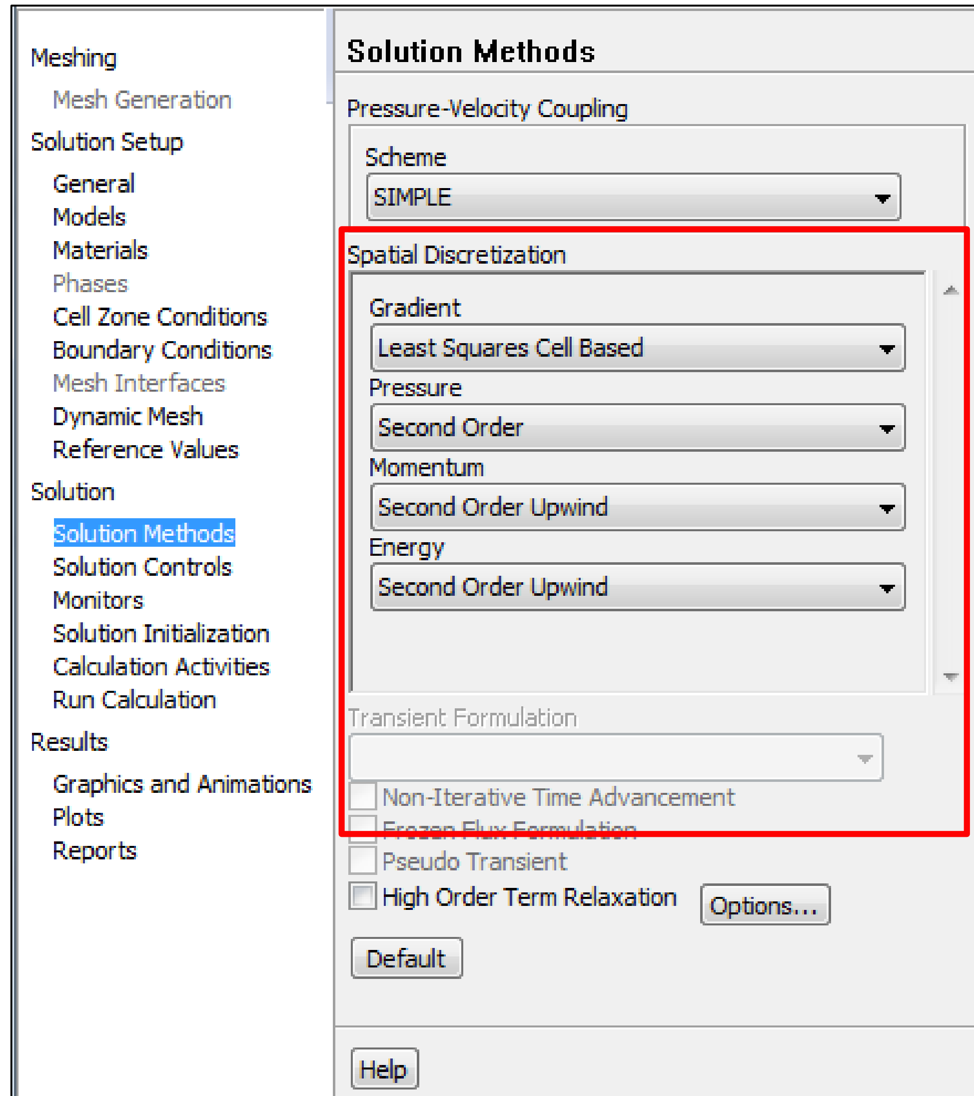
For top surface, solid in and out surfaces: adiabatic and non-slipping wall



Step 7: Solution setup: algorithm and scheme

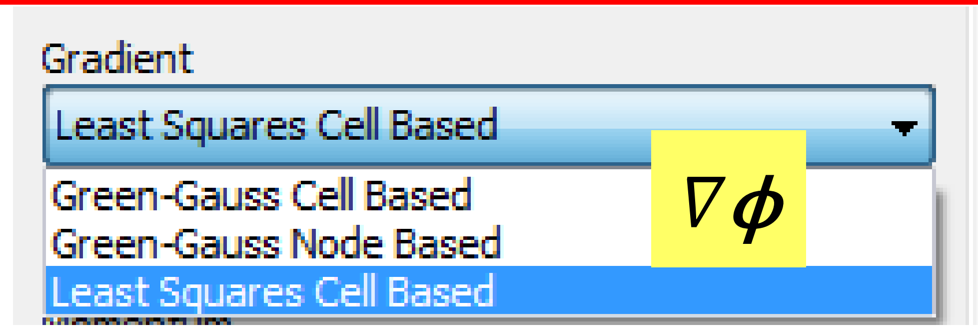
Remark: In Fluent, for the SIMPLE series algorithms, only SIMPLE and SIMPLEC are included.

Review: What is the difference between SIMPLE, SIMPLEC and SIMPLER?



Gradient calculation,

There are three schemes.



1. **Green-Gauss Cell-Based (格林-高斯基于单元法)**
2. **Green-Gauss Node-Based (格林-高斯基于节点法)**
3. **Least-Squares Cell Based 基于单元体的最小二乘法**

   It is the default scheme for gradient calculation.

Green-Gauss Theory:

The averaged gradient over a control domain is:

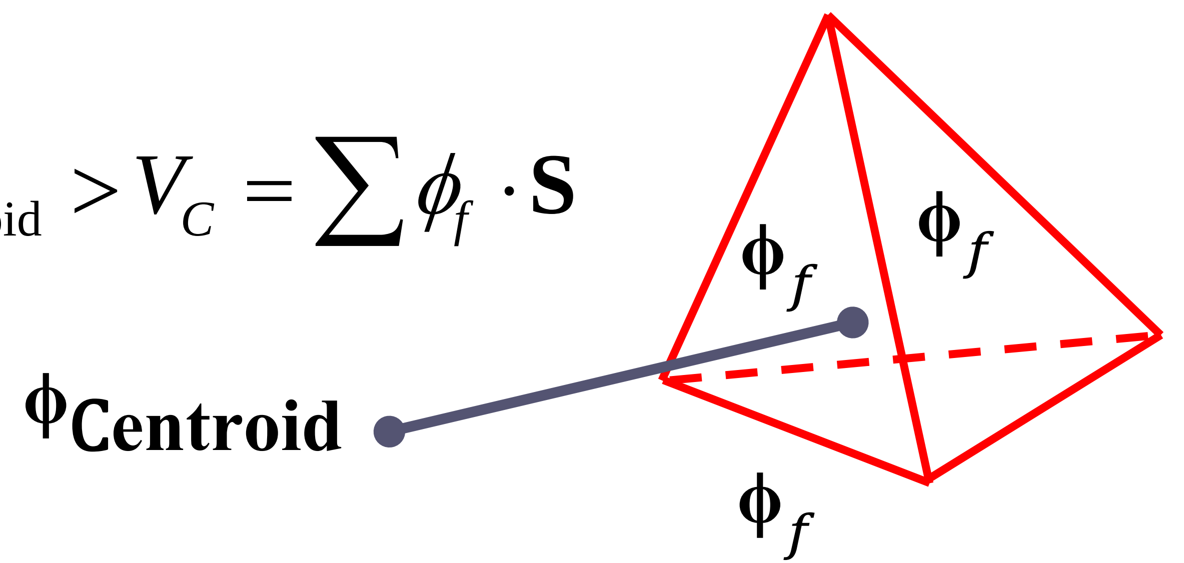
$$<\nabla\phi>=\frac{1}{V_C}\int_{V_C}\nabla\phi dV$$

Using the Gauss integration theory （高斯定理）, the volume integral (体积分）is transformed into a surface integral（面积分）:

$$< \nabla \phi > = \frac{1}{V_C} \int_{V_C} \nabla \phi dV = \frac{1}{V_C} \oint \phi \cdot \mathbf{n} dS$$

In the presence of discrete faces, the above equation can be written as:

$$< \nabla \phi_{\text{centroid}} > V_C = \sum \phi_f \cdot \mathbf{S}$$



$$\nabla \phi_{\text{centroid}} V_C = \sum \phi_f \cdot \mathbf{n} S$$

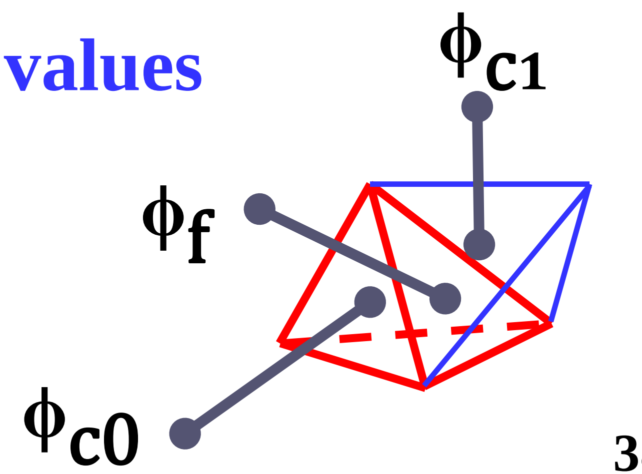
The problem of calculating gradient is transferred into the following equation:

How to determine ϕ_f at the face?

1. Green-Gauss Cell-Based (格林-高斯基于单元法)

Calculate ϕ_f using cell centroid values (网格中心点).

$$\phi_f = \frac{\phi_{C0} + \phi_{C1}}{2}$$

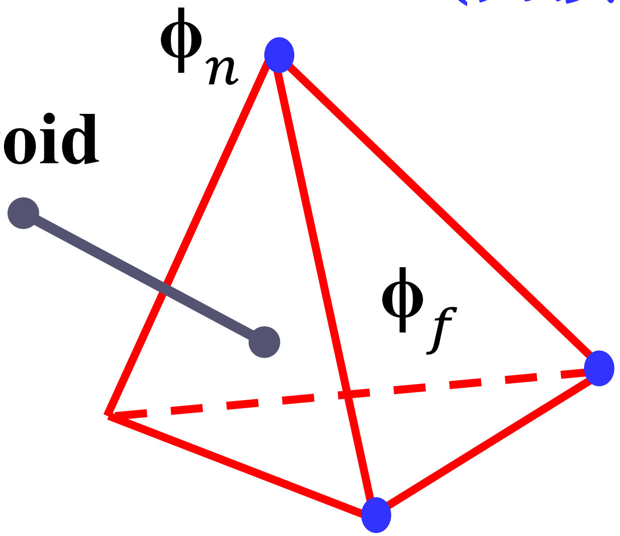


2. Green-Gauss Node-Based (格林-高斯基于节点法)

Calculate ϕ_f by the average of the node values. (面顶点的代数平均值)

$$\boxed{\phi_f = \frac{1}{N_f}\sum \phi_n}$$

$$\phi_n = \sum_{i}^{N_{\text{cells}}(n)} \phi_{c_i} w_{c_i,n}$$



Nf： number of nodes on the face, ϕ_n: node value.

ϕ_n， is calculated by weighted average of the cell values surrounding the nodes ϕ_{Ci} .

Review: **the node-based method is more accurate than the cell-based method.**

3. Least-Squares Cell Based 基于单元体的最小二乘法

It is the default scheme for gradient calculation.

The basic idea is as follows. Consider two cell centroid C_0 and C_i, and their distance vector as δr. Then, the following equation

$$\phi_{Ci} = \phi_{C0} + (\nabla\phi)\cdot(\mathbf{r}_{Ci} - \mathbf{r}_{C0})$$

is exact only when the solution field is linear! In other words, there is no second-order term for Taylor expansion of ϕ!

For a cell centroid C_0 with N neighboring nodes C_i,

$$\Phi_{Ci} = \phi_{Ci} - \underbrace{\left[\phi_{C0} + (\nabla\phi)\cdot(\mathbf{r}_{Ci} - \mathbf{r}_{C0})\right]}_{\text{Calculated value}}$$

True value **Calculated value**

Making summation of all these Φ_{Ci} with a weighting factor w_i

$$\xi = \sum_{i=1}^{N} w_i \Phi_{Ci} = \sum_{i=1}^{N} \left\{ w_i \left(\phi_{Ci} - \left[\phi_{C0} + (\nabla\phi)\cdot(\mathbf{r}_{Ci} - \mathbf{r}_{C0})\right] \right)^2 \right\}$$

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

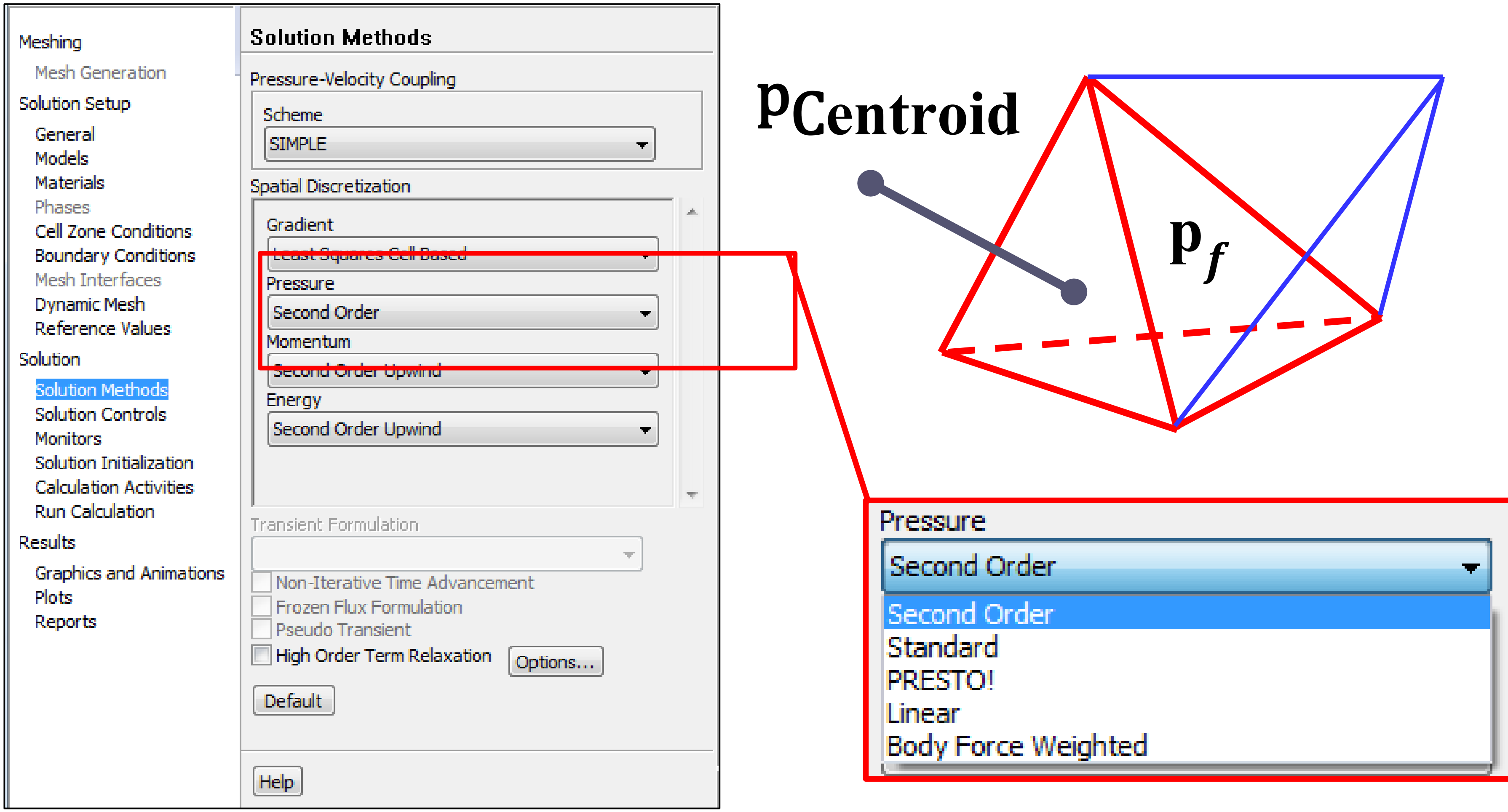
Therefore, to calculate the gradient $\nabla\phi$ is to find the one leading to the minimum ξ!

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

This is the idea of Least-Squares method.

Remark: On irregular (不规则) unstructured meshes, the accuracy of the least-squares gradient method is comparable to that of the node-based gradient. However, it is more computational efficient compared with the node-based gradient.

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



1. Linear scheme

Computes the face pressure use the average of the pressure values in the adjacent cells.

$$P_f = \frac{P_{C0} + P_{C1}}{2}$$

2. Standard scheme

Interpolate the pressure using momentum equation coefficient.

$$P_f = \frac{\dfrac{P_{c0}}{a_{P,c0}} + \dfrac{P_{c1}}{a_{P,c1}}}{\dfrac{1}{a_{P,c0}} + \dfrac{1}{a_{P,c1}}}$$

3. Second Order

Calculate the pressure value using a central difference scheme

$$P_f \approx \frac{P_{C0} + \nabla P_{C0}\mathbf{r}_{C0} + P_{C1} + \nabla P_{C1}\mathbf{r}_{C1}}{2}$$

4. Body Force Weighted scheme

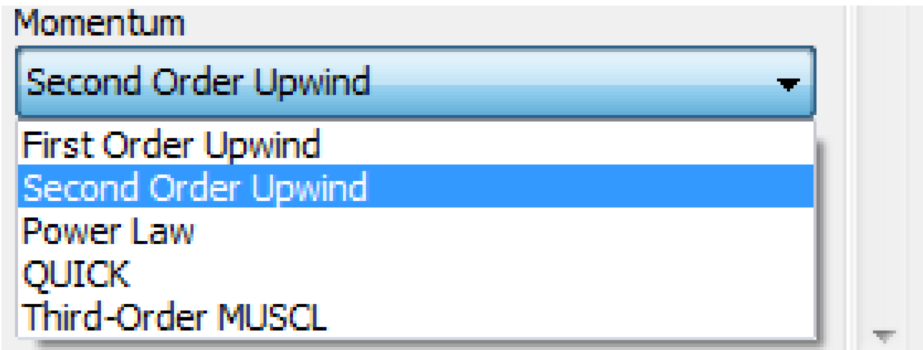
Calculate the pressure according to the body force.

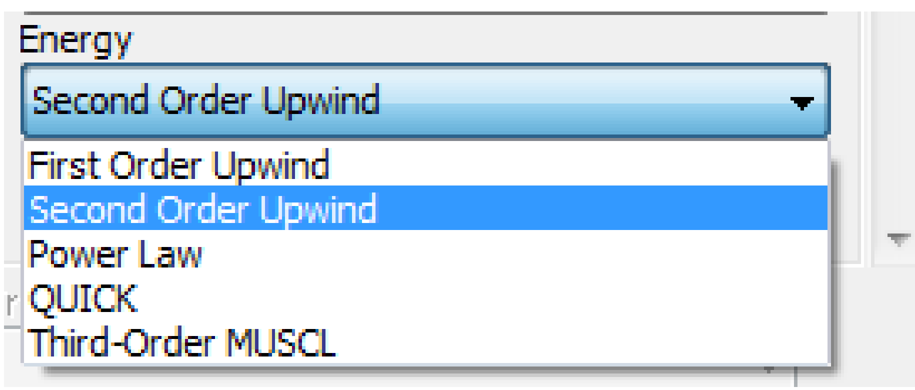
- ✓ **Multiphase flow such as VOF (Volume of Fluid，体积函数法) or LS (Level Set, 水平集): recommended.**
- ✓ **For porous media: not recommended!**

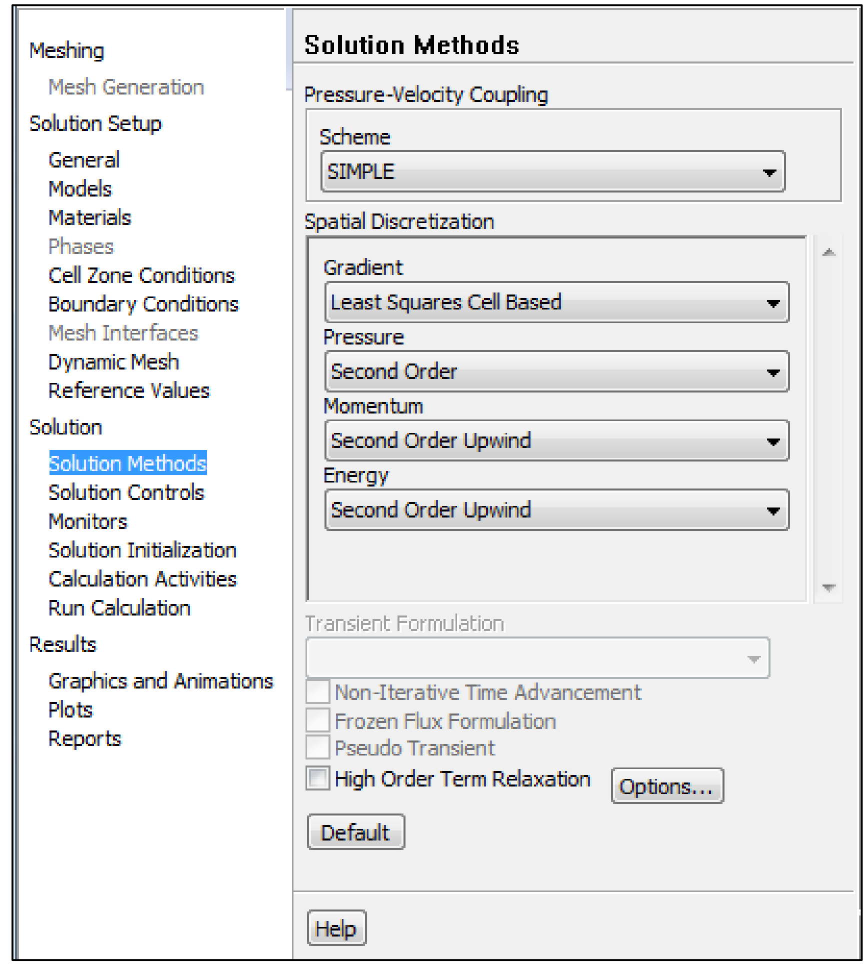
5. PRESTO! (Pressure Staggering Option) scheme

For problem with high pressure gradient.

For convective term scheme, we are very familiar!

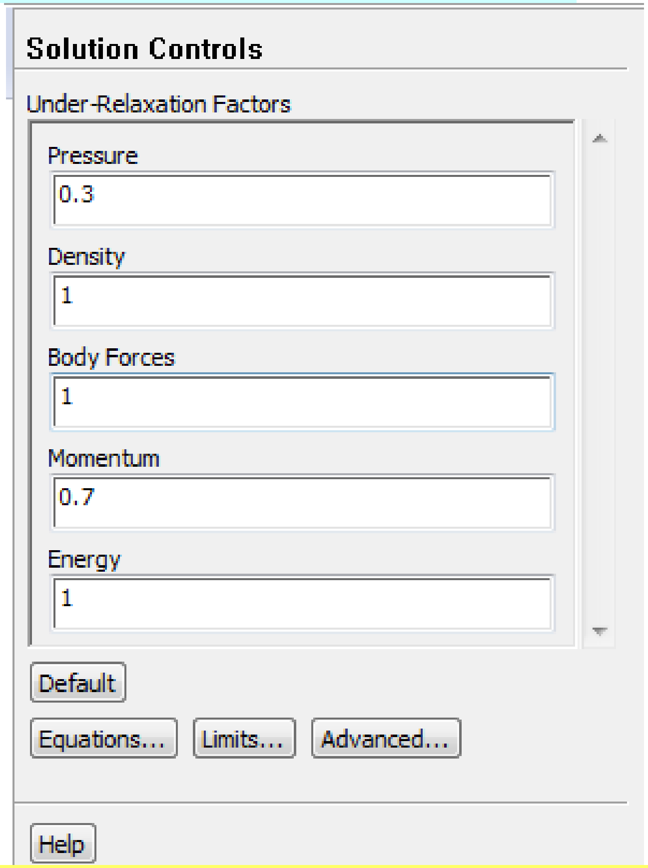






Step 7: Solution setup: relaxation

Under-relaxation is adopted to control the change rate of simulated variables in subsequent iterations.



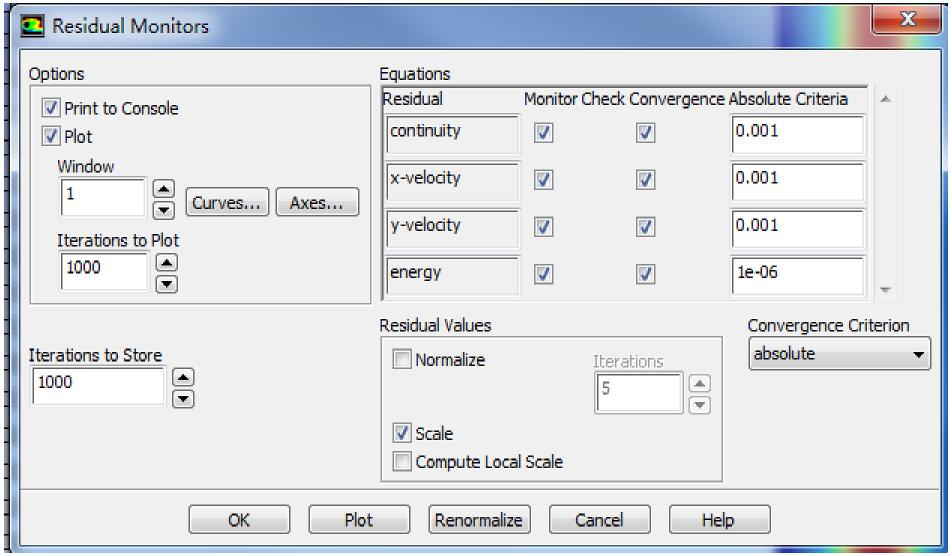
The relaxation factor α for each variable has been optimized for the largest possible.

In some cases, if your simulation is not converged, and you are sure there is no problem with other setting, you can try to reduce α!

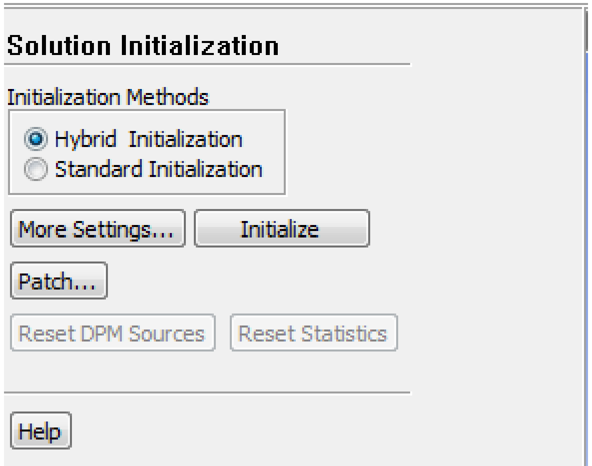
Step 7: Solution setup: monitors

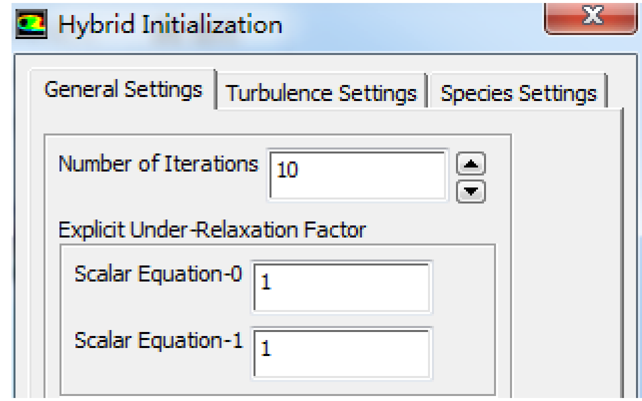
Similar to "Print" function in our teaching code, you can use Monitors in Fluent to setup a certain number of variables to monitor the iteration process of the simulation.

The Residuals are the most important values to be monitored. You can set the related values.



Step 8: Initialization



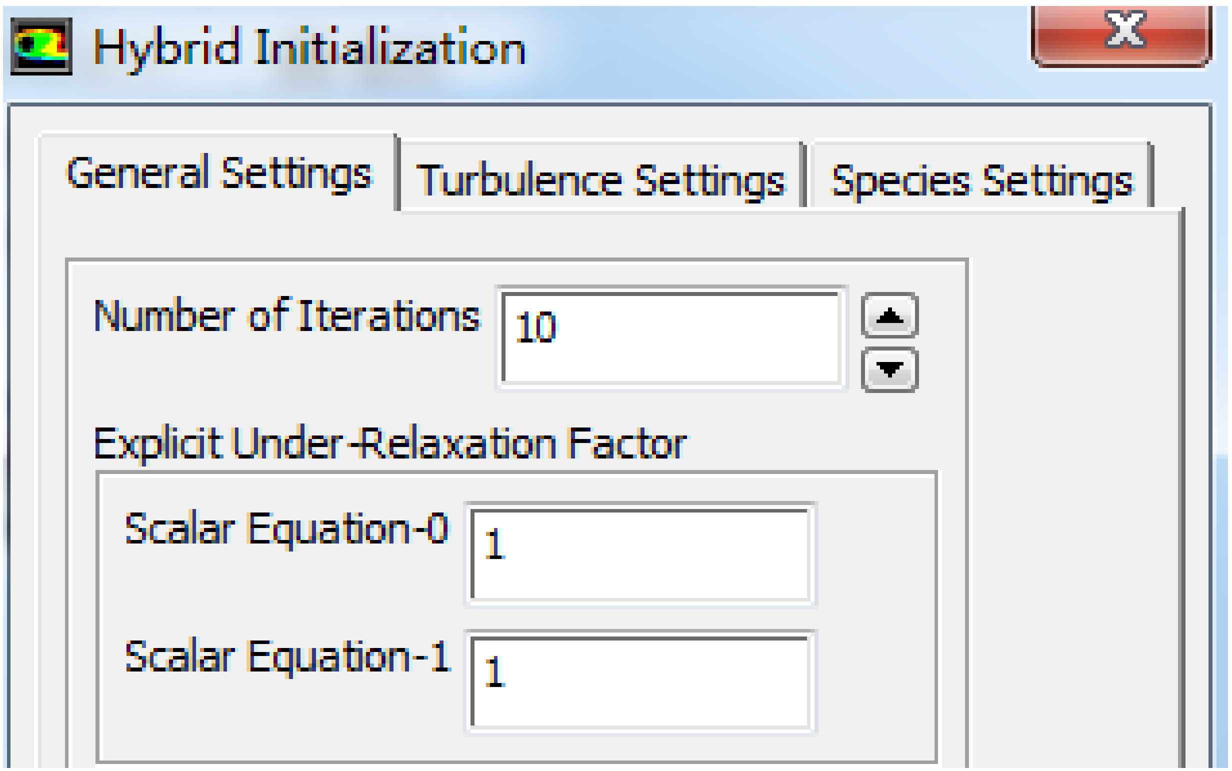


The default selection is Hybrid initialization (混合初始化).

The initial pressure and velocity field you give usually are not consistent, in other words, not meet the NS equation.

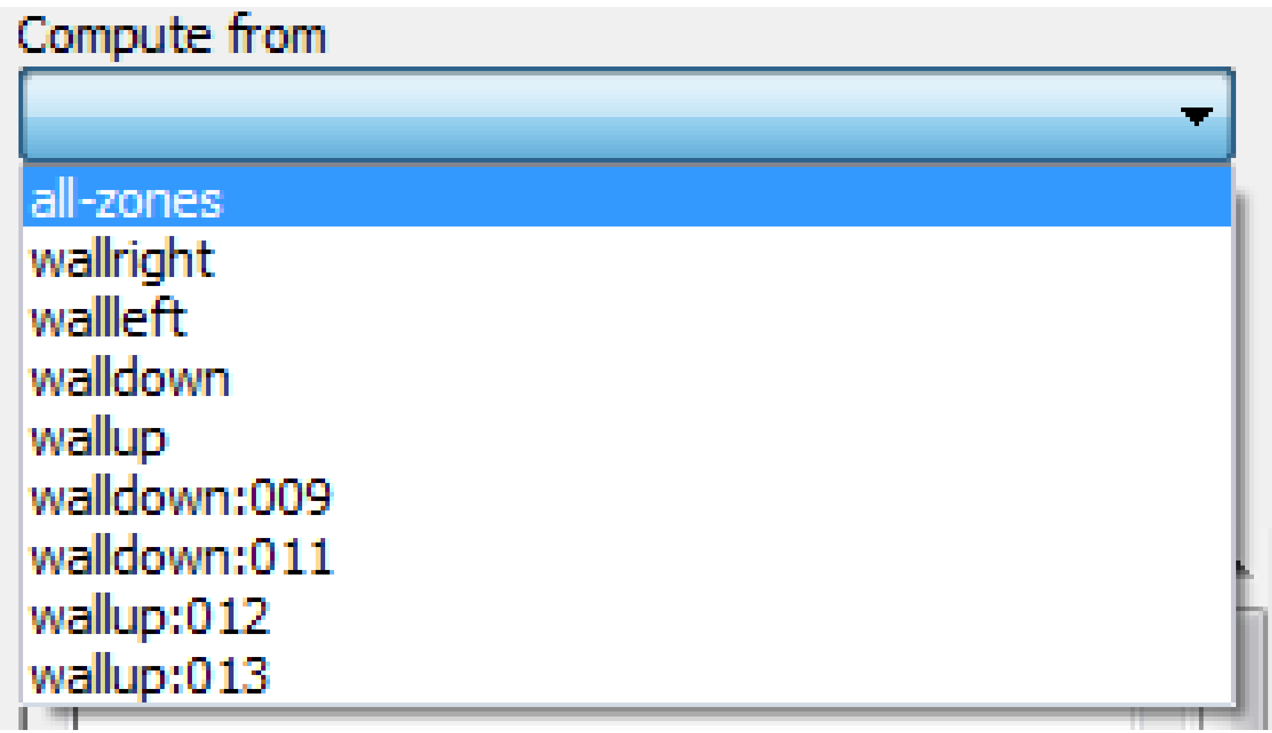
In SIMPLER algorithm, we solved an additional Poisson equation for pressure based on given velocity.

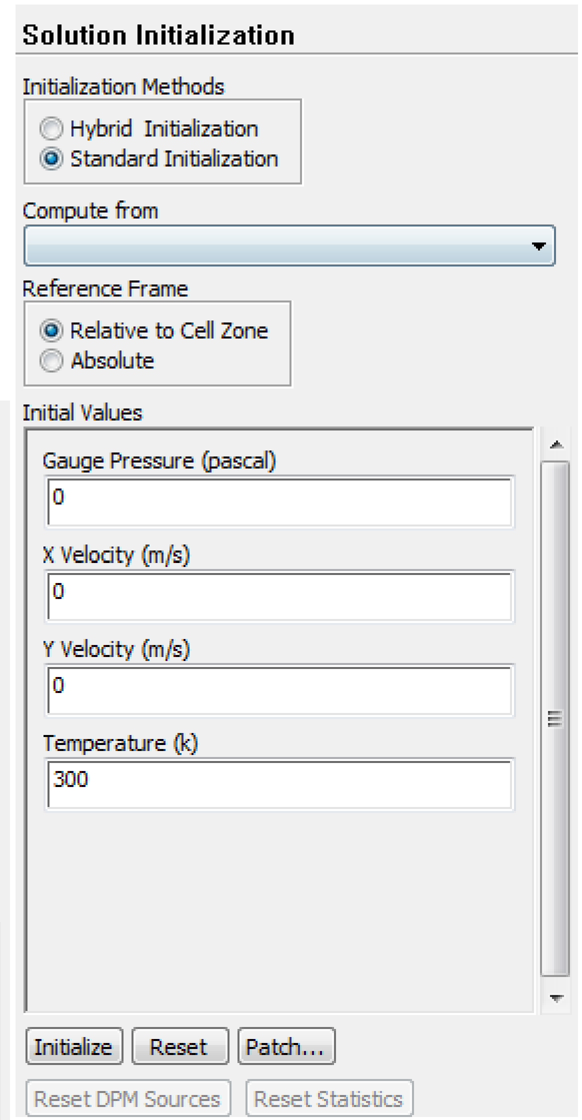
The Hybrid initialization method is similar that Poisson equation is solved to initialize the velocity and pressure equation. You can set the number of iterations to make sure the initial velocity and pressure are consistent.



Or you can simply chose Standard initialization method.

Click Compute from, the drop-down list will show, and you can select an region.



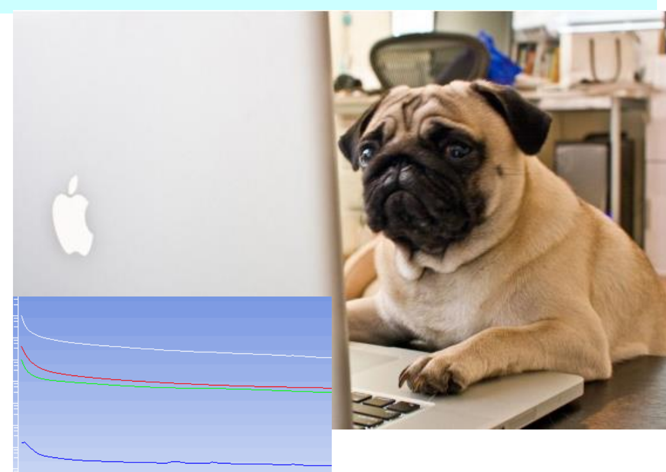


The eight steps for preparing a Fluent simulation have been completed!

1. Read mesh
2. scale domain
3. Choose model
4.define material
5. define zone condition
6. define boundary condition
7. Solution step
8. Initialization
9. Run the simulation.
10. Post-process

Step 9: Run the simulation

What should you do in this step? Just stare at the monitor to hope that the residual curves are going down for a steady problem.



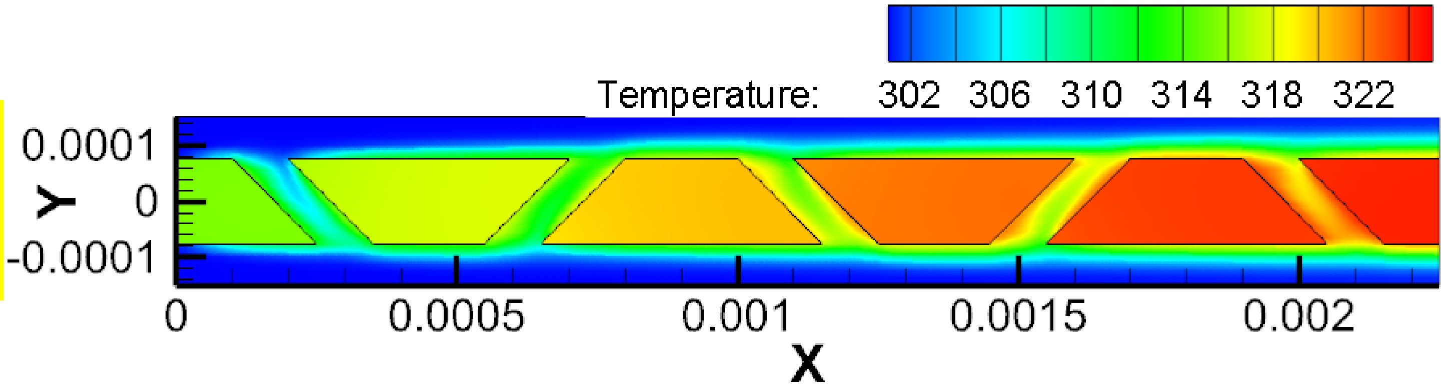
Diverged? Go back to Steps 1 to 8.

Review: The 10 steps for a Fluent simulation:

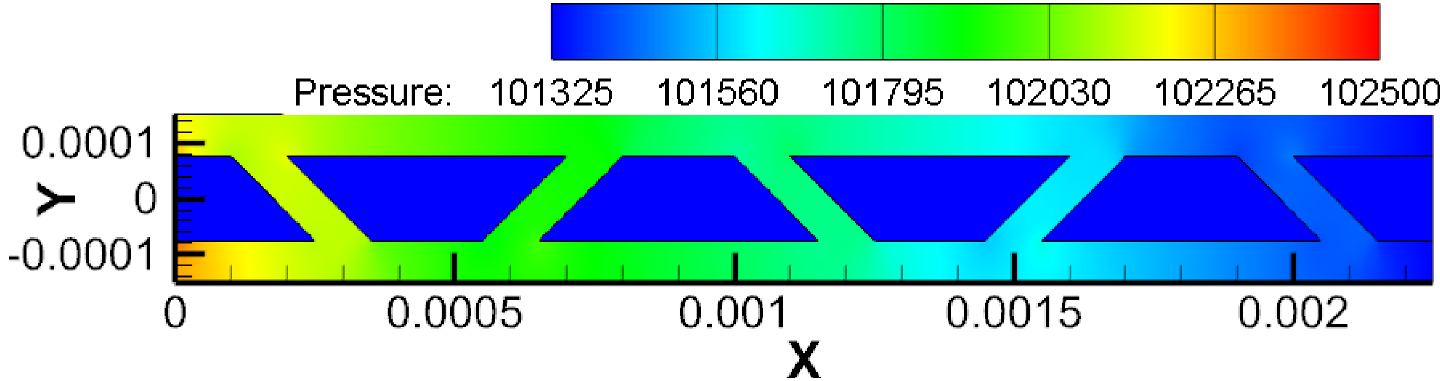
1. Read and check the mesh: mesh quality.
2. Scale domain: make sure the domain size is right.
3. Choose model: write down the right governing equation is very important.
4. Define material: the solid and fluid related to your problem.
5. Define zone condition: material of each zone and source term
6. 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.

Re=100

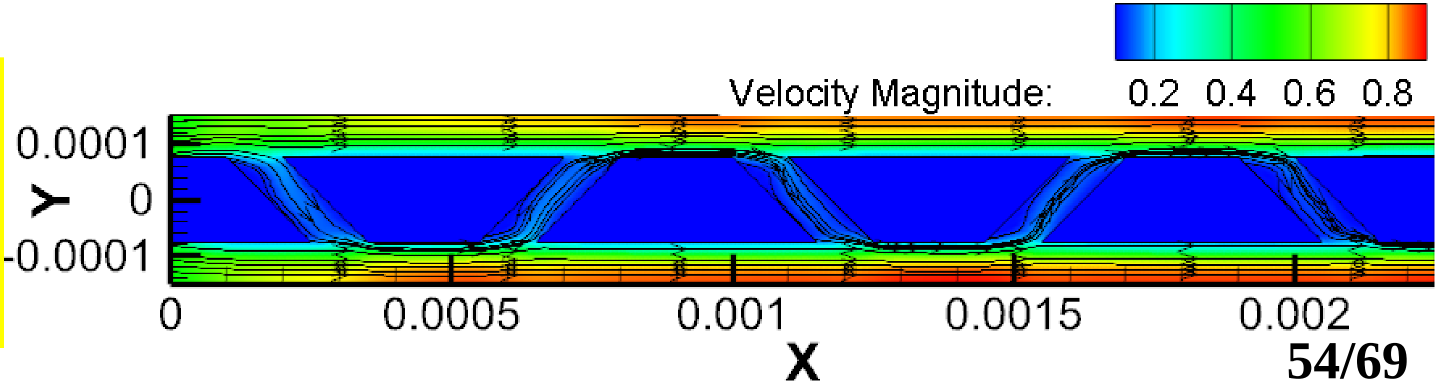
Temperature distribution



Pressure distribution



Streamline and velocity distribution



Step 10: Post-process: Data reduction

The Reynolds number (*Re*) is expressed as follow:

$$Re = \frac{\rho u_m D_h}{\mu}$$

$$D_h = \frac{2H_c W_c}{H_c + W_c}$$

u(m/s)	0.5	1	1.5	2	2.5
Re	100	200	300	400	500

Friction factor

$$f=\frac{2D_h\Delta P}{L_t\rho u_m^2}$$

Heat transfer coefficient

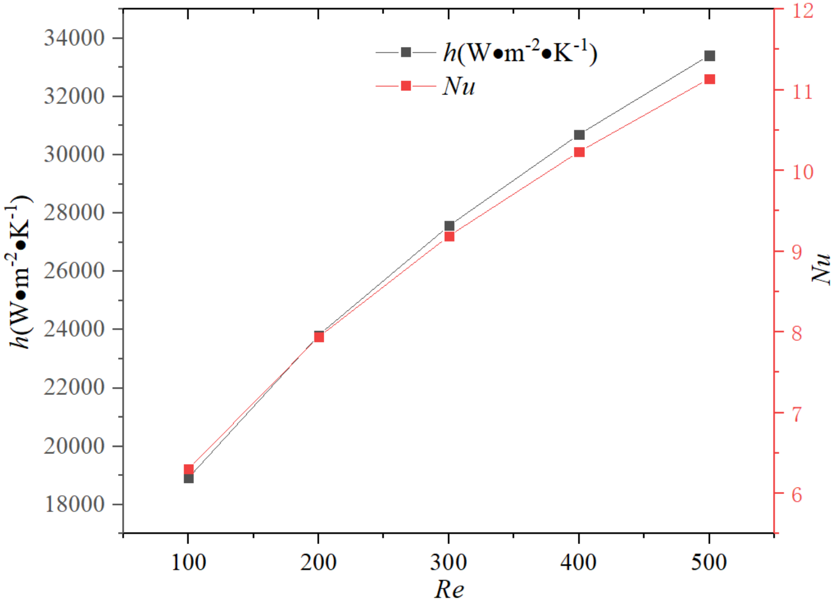
$$h_{ave}=\frac{q_w A_s}{A_{con}(T_{w,ave}-T_{f,ave})}$$

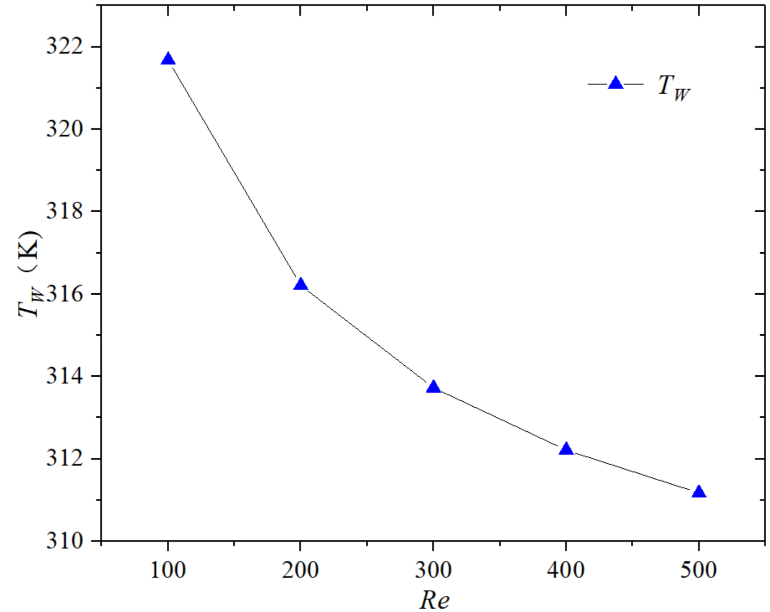
$$q_w A_s = hA_{con}\Delta T_m = C_p M(T''-T')$$

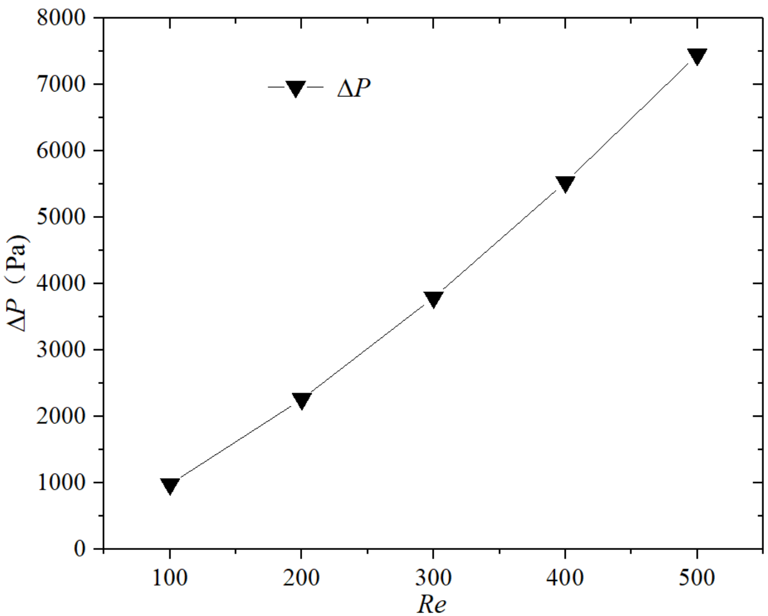
Average Nusselt number

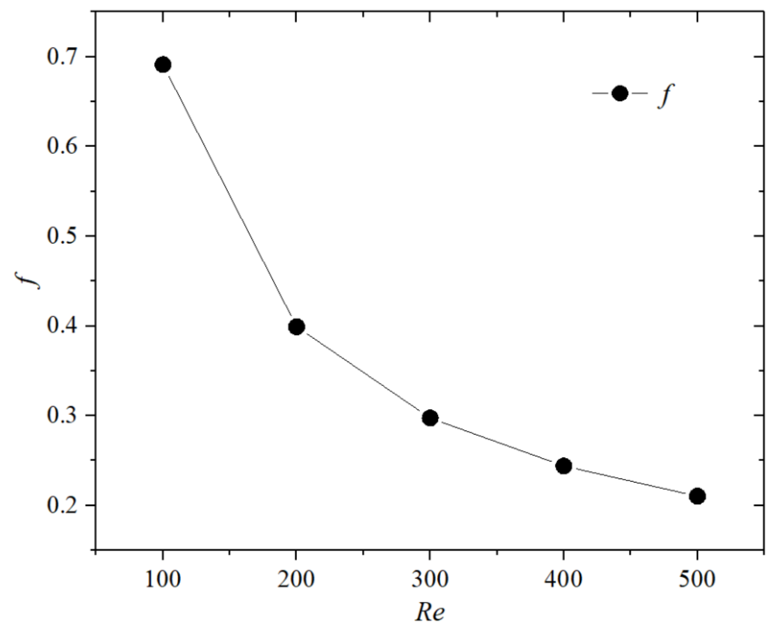
$$Nu_{ave}=\frac{h_{ave}D_h}{\lambda_f}$$

Re	100	200	300	400	500
Nu	6.21	7.89	9.158	10.2	11.12
ΔP(Pa)	979.63	2262.75	3793.16	5532.08	7451.92
f	0.691	0.399	0.297	0.244	0.21
T_W(K)	321.68	316.21	313.73	312.22	311.17


h(W•m^{-2}•K^{-1})
Nu
34000
32000
30000
28000
26000
24000
22000
20000
18000
12
11
10
9
8
7
6
100
200
300
400
500
Re


T_W
T_W（K）
322
320
318
316
314
312
310
100
200
300
400
500
Re


ΔP
ΔP（Pa）
8000
7000
6000
5000
4000
3000
2000
1000
0
100
200
300
400
500
Re


f
f
0.7
0.6
0.5
0.4
0.3
0.2
100
200
300
400
500
Re