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.-16

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16日，西安

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

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

导热问题

13.3 顶盖驱动流动换热问题

13.4 微通道内流动换热问题

混合对流问题

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

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

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

13.1 Heat transfer with source term

Known： Steady heat conduction occurs in a three-layer solid region with source term in the middle layer. The outer two layers are Aluminum(铝), while the middle layer is nuclear fuel zone. The domain and size is shown in Fig. 1. The boundary conditions are as follows:

- Left and right boundary---adiabatic
- Top and bottom boundary-- convective heat transfer

  Heat transfer coefficient: h=3500 W/(m^2 •K);
  Fluid temperature: T_f=150^{o}C.

- In the middle layer--- a constant source term

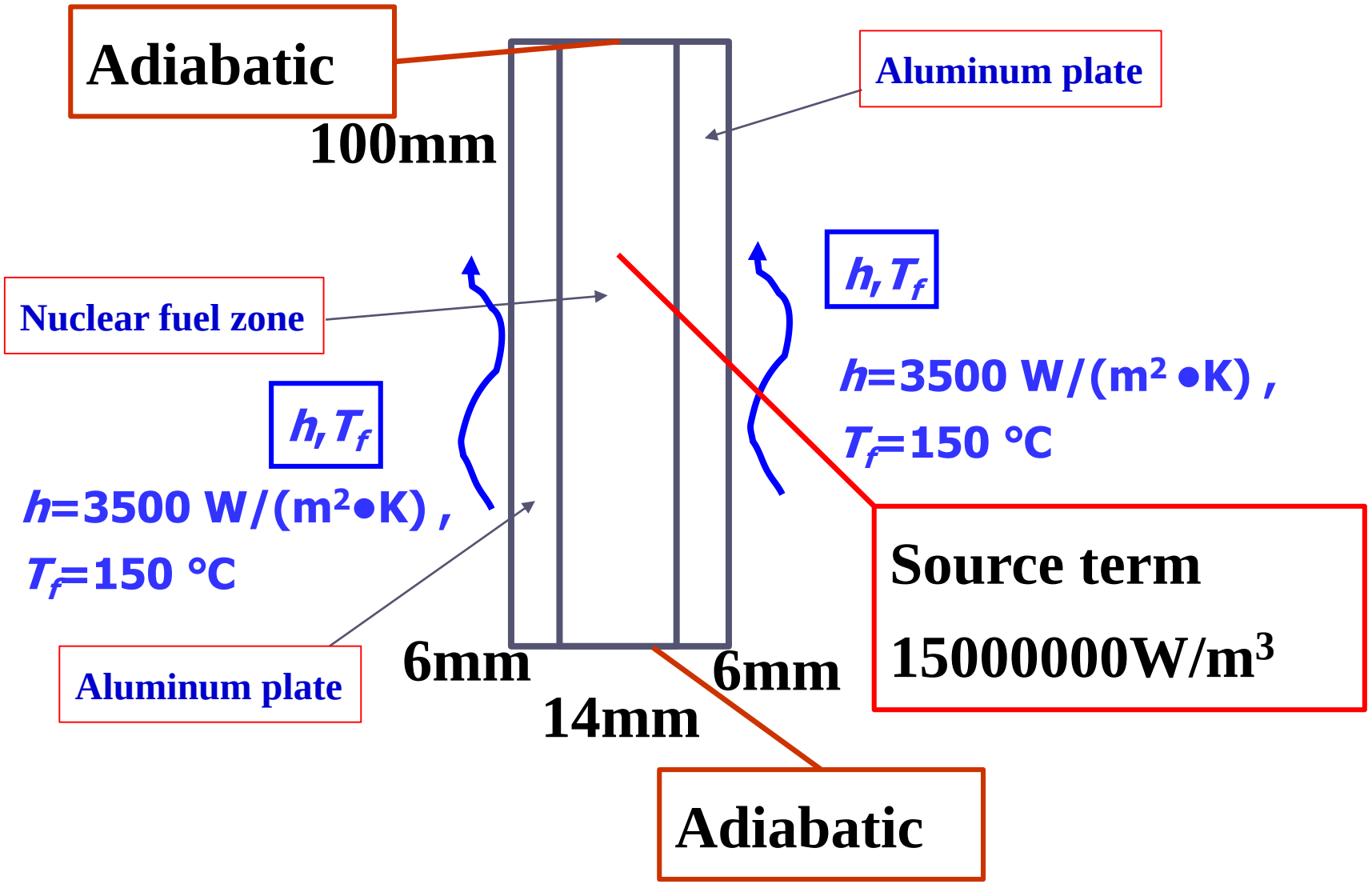


Fig.1 Computational domain

Find: **Temperature distribution in the domain.**

Solution:

$$div(\Gamma_\phi grad\phi) + S_\phi^* = 0$$

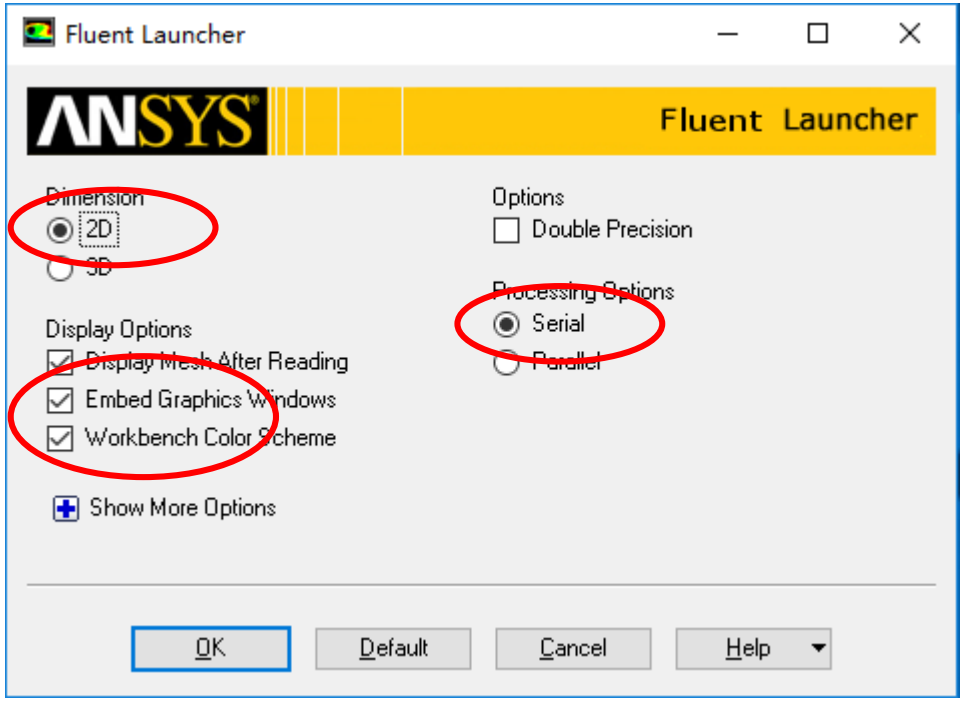
It is a heat conduction problem with given GAMA and source term. GAMA and source term are different for different layers.

Remark: Constructing the reasonable physical model and writing down the corresponding 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 to use the tool better.

Start the Fluent software



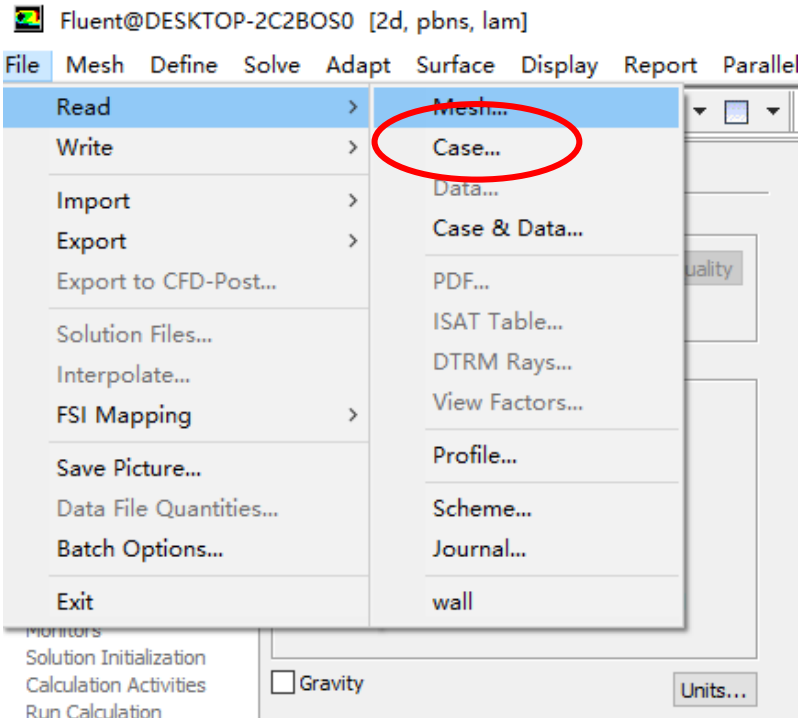
1. Choose 2-Dimension
2. Choose display options
3. Choose Serial processing option

Note: Double precision or Single precision

For most cases the single precision version of Fluent is sufficient. However, 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.

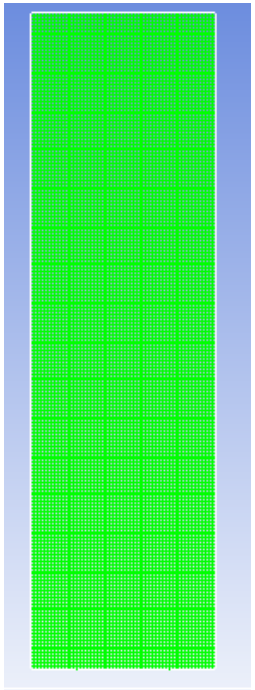


Mesh→Read

```
> Reading "F:\Fluent\case1\wall.msh"...
Done.
   11457 nodes.
     200 mixed wall faces, zone  5.
     200 mixed wall faces, zone  6.
      56 mixed wall faces, zone  7.
      56 mixed wall faces, zone  8.
   22144 mixed interior faces, zone 10.
    5600 quadrilateral cells, zone  2.
    2800 quadrilateral cells, zone  3.
    2800 quadrilateral cells, zone  4.
```

Step 1: Read and check the mesh

Mesh→Check



- Check the quality and topological information of the mesh

```
Mesh Check
      default-interior -> default-interior (10) and default-interior:017 (17)
 Domain Extents:
   x-coordinate: min (m) = -1.400000e-02, max (m) = 1.400000e-02
   y-coordinate: min (m) = -5.000000e-02, max (m) = 5.000000e-02
 Volume statistics:
   minimum volume (m3): 2.499982e-07
   maximum volume (m3): 2.500008e-07
     total volume (m3): 2.800000e-03
 Face area statistics:
   minimum face area (m2): 4.999973e-04
   maximum face area (m2): 5.000010e-04
 Checking mesh.........................
Done.
```

- **Sometimes the check will be failed if the quality is not good or there is a problem with the mesh.**

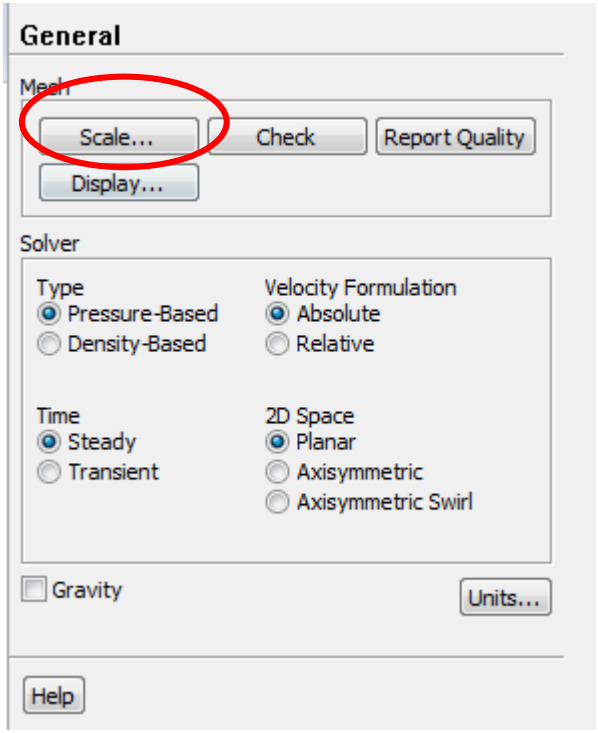
```
Face area statistics:
WARNING: invalid or face with too small area exists.
minimum face area (m2): 0.000000e+00
maximum face area (m2): 5.081937e-03
```

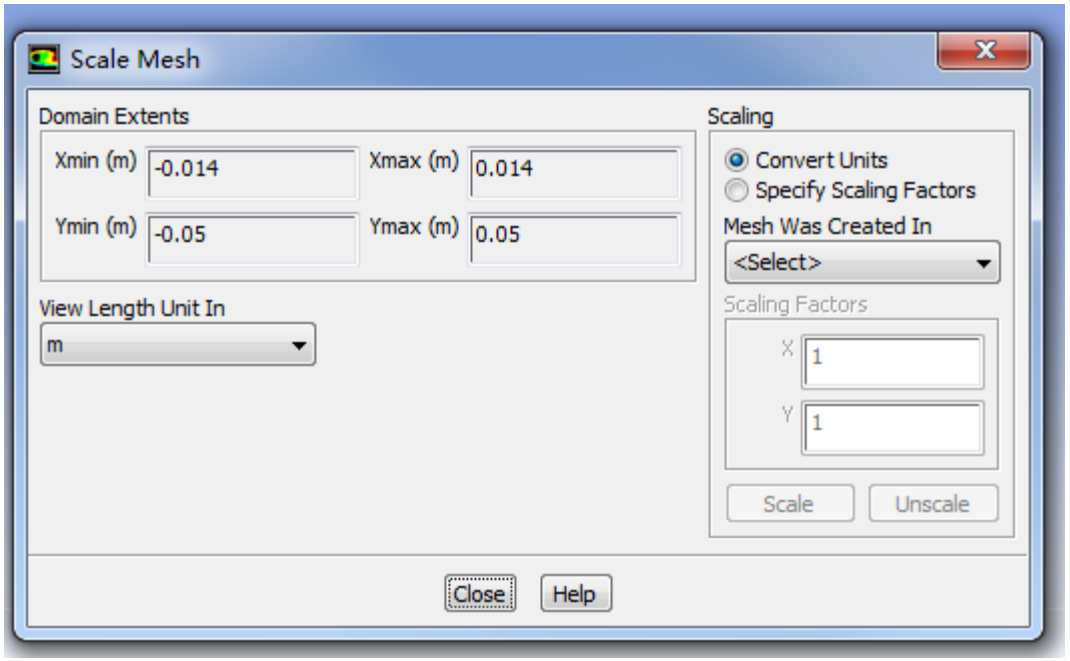
```
WARNING: Mesh check failed.
```

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

Step 2: Scale the domain size

General→Scale



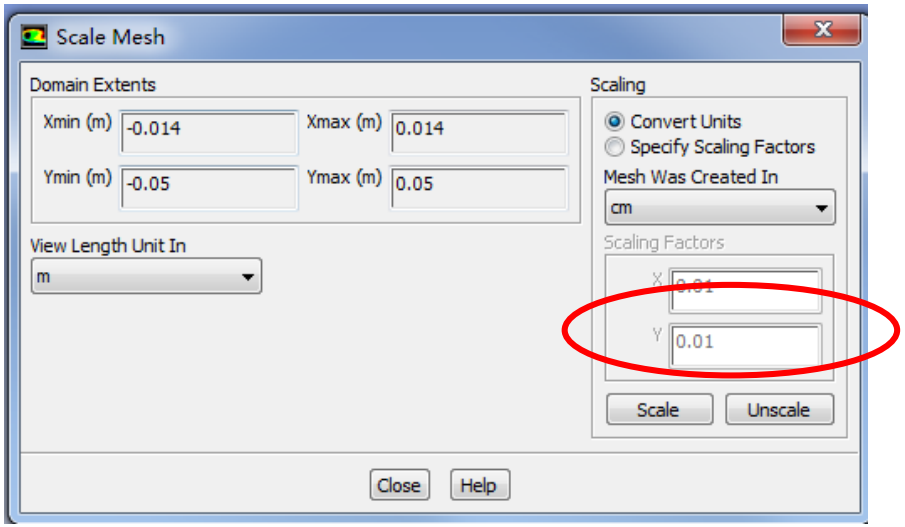


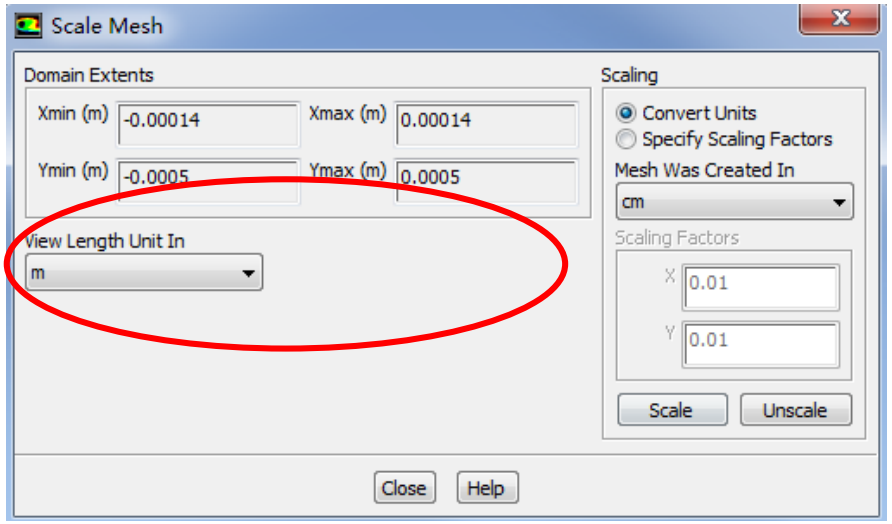
- **Fluent stores the mesh in units as "m", SI unit. You can show it in different units such as cm, mm, in, or ft.**

- **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.**

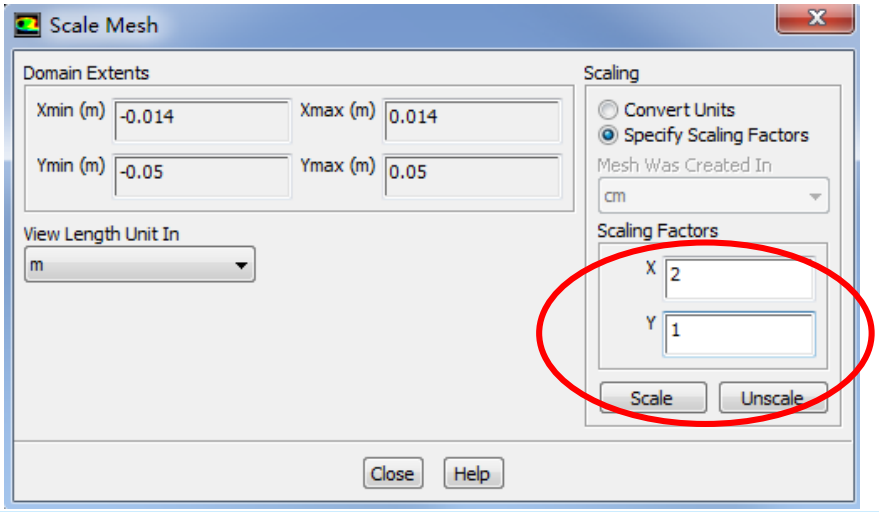
ICEM: 1 cm -> Fluent: 1m -> Scale: cm, factor: 0.01

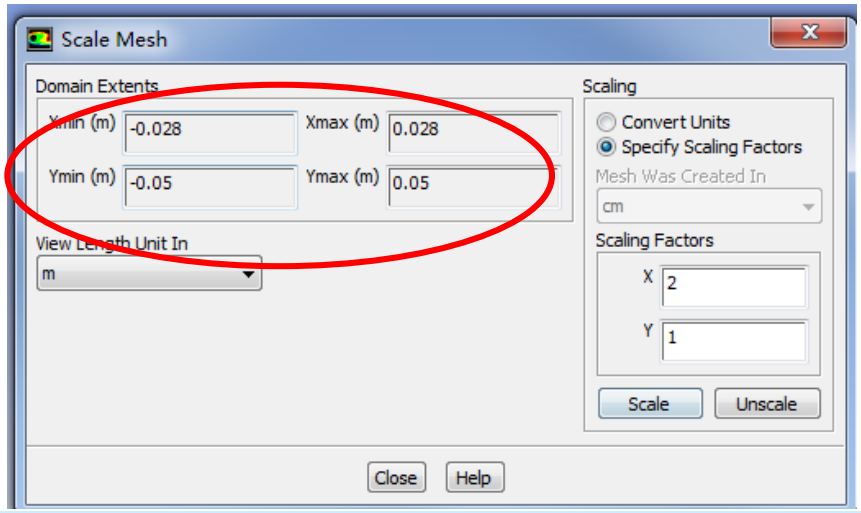




- **"Specify Scaling factors", by using this you can define your own scaling factor.**

Remark: In "Convert Unit", the scaling factor is fixed. You can also use "Specify Scaling Factors" to define your own scaling factor. For example, in the following, the size of *x* direction is doubled.





- **If you click the Scale button by accident or use the wrong Scaling factor, you can click the Unscale button. The old size will be recovered.**

Step 3: Choose the physicochemical model

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

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

$$div(\Gamma_\phi grad\phi) + S_\phi^* = 0$$

This is the equation we will solve, the energy equation for heat conduction in solid without radiation and convection.

Remark: **The governing equation of energy solved by Fluent is as follows:**

$$\frac{\partial}{\partial t}(\rho E)+\nabla\cdot(\vec{v}(\rho E+p))=\nabla\cdot\left(k_{eff}\nabla T-\sum_j h_j\vec{J}_j+(\bar{\bar{\tau}}_{eff}\cdot\vec{v})\right)+S_h$$

$$E=h-\frac{p}{\rho}+\frac{v^2}{2}$$ **Enthalpy**

8.1 Format Improvement of General Governing Equation

$$\frac{\partial(\rho c_p T)}{\partial t}+\frac{\partial(\rho c_p u T)}{\partial x}+\frac{\partial(\rho c_p v T)}{\partial y}=\frac{\partial}{\partial x}\left(\lambda\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\lambda\frac{\partial T}{\partial y}\right)+S_T.$$

$$\frac{\partial(\rho T)}{\partial t}+\frac{\partial(\rho u T)}{\partial x}+\frac{\partial(\rho v T)}{\partial y}=\frac{\partial}{\partial x}\left(\frac{\lambda}{c_p}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\lambda}{c_p}\frac{\partial T}{\partial y}\right)+\frac{S_T}{c_p}$$

$$-\frac{1}{c_p^2}\left[\rho c_p T\frac{\partial c_p}{\partial t}+\left(\rho c_p u T-\lambda\frac{\partial T}{\partial x}\right)\frac{\partial c_p}{\partial x}+\left(\rho c_p v T-\lambda\frac{\partial T}{\partial y}\right)\frac{\partial c_p}{\partial y}\right]$$

Improved format of the general G.E.

The frame work of the previous G.E. is retained (保留), but the diffusion coefficient is resumed to （恢复到） its original value by introducing a nominal density as follows:

$$\frac{\partial(\rho^{*}\phi)}{\partial t}+div(\rho^{*}\phi\vec{U})=div(\Gamma_{\phi}grad\phi)+S_{\phi}^{*}$$

The new form of G.E. are:

Equation	ρ^*	ϕ	Γ_ϕ	$S_\phi{}^*$
Continuity equation	ρ	1	0	0
Momentum eqn. (x direction)	ρ	u	μ	$\rho f_x-\frac{\partial p}{\partial x}$
Momentum eqn. (y direction)	ρ	v	μ	$\rho f_y-\frac{\partial p}{\partial y}$
Energy equation	ρc_p	T	λ	S_T

Fluent for solid phase:

V is rotational or translational motion of solids

h: enthalpy

$$\frac{\partial}{\partial t}(\rho h)+\nabla\cdot(\vec{v}\rho h)=\nabla\cdot(k\nabla T)+S_h$$

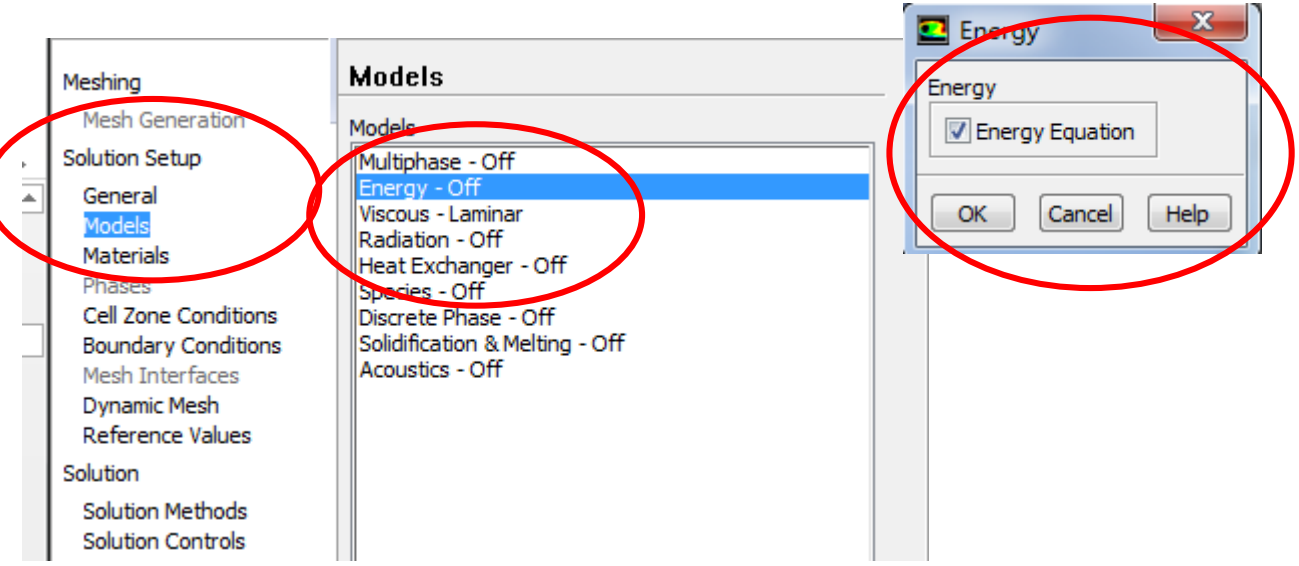
Our general Code:

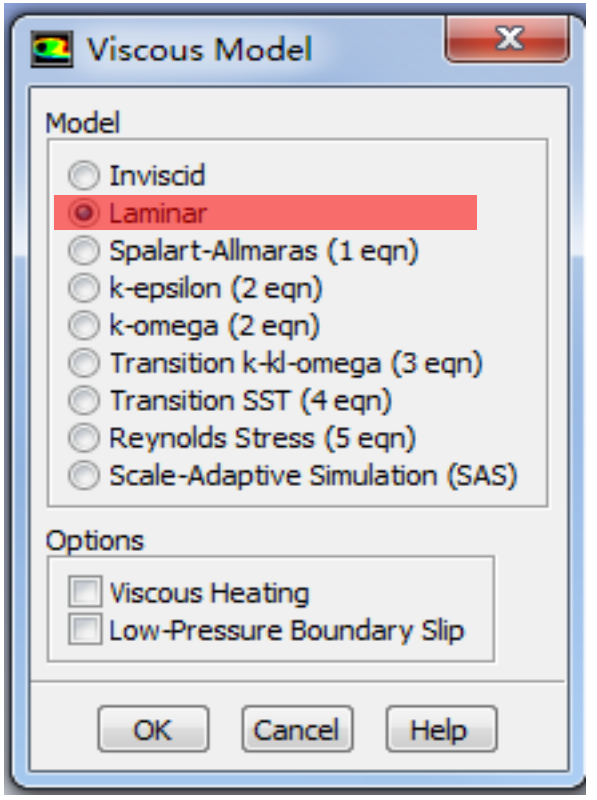
$$\frac{\partial(\rho c_p T)}{\partial t}+\frac{\partial(\rho c_p u T)}{\partial x}+\frac{\partial(\rho c_p v T)}{\partial y}=\frac{\partial}{\partial x}\left(\lambda\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(\lambda\frac{\partial T}{\partial y}\right)+S_T.$$

Such a treatment is much better than taking Γ / c_p as a nominal diffusion coefficient and S_T / c_p as a nominal source term.

To select the model, the command is as follows:

Solution Setup→Model

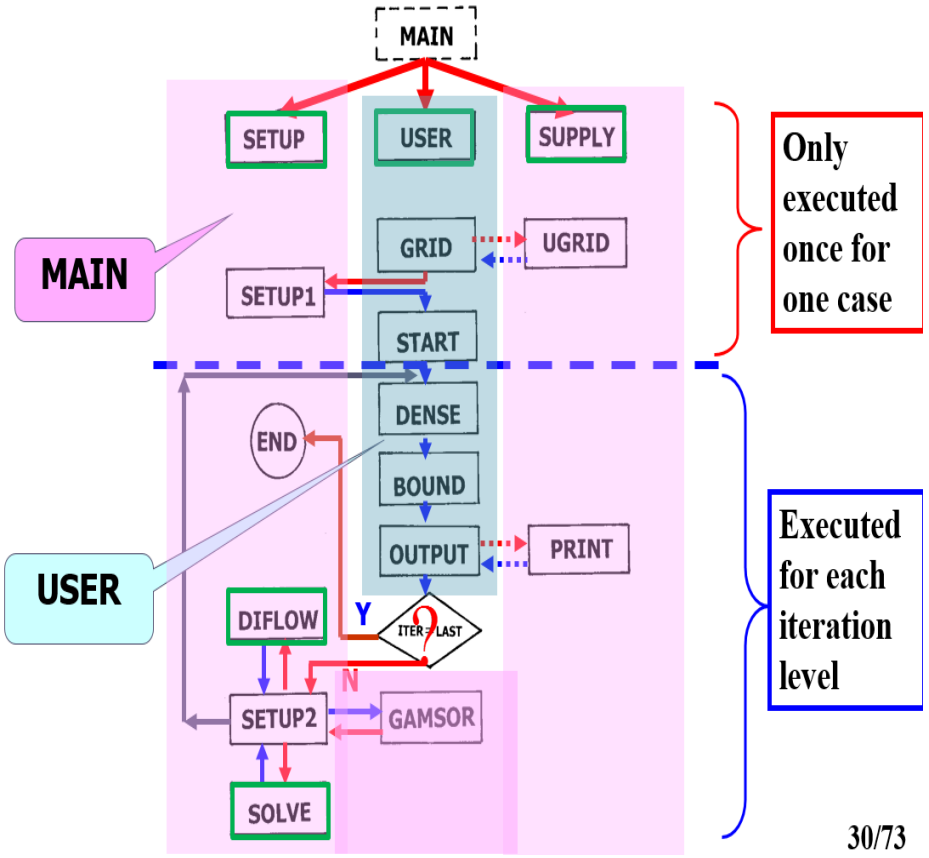




Remark: **It is interesting to note that no matter what kind of problems you will solve, the model "Viscous-Laminar" is always activated. We can cancel it later.**

Remark: **In our general code,**

In SETUP2, Visit NF from 1 to NFMAX in order; **When some value of NF is visited and LSOLVE(NF)=.T. , then 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.**



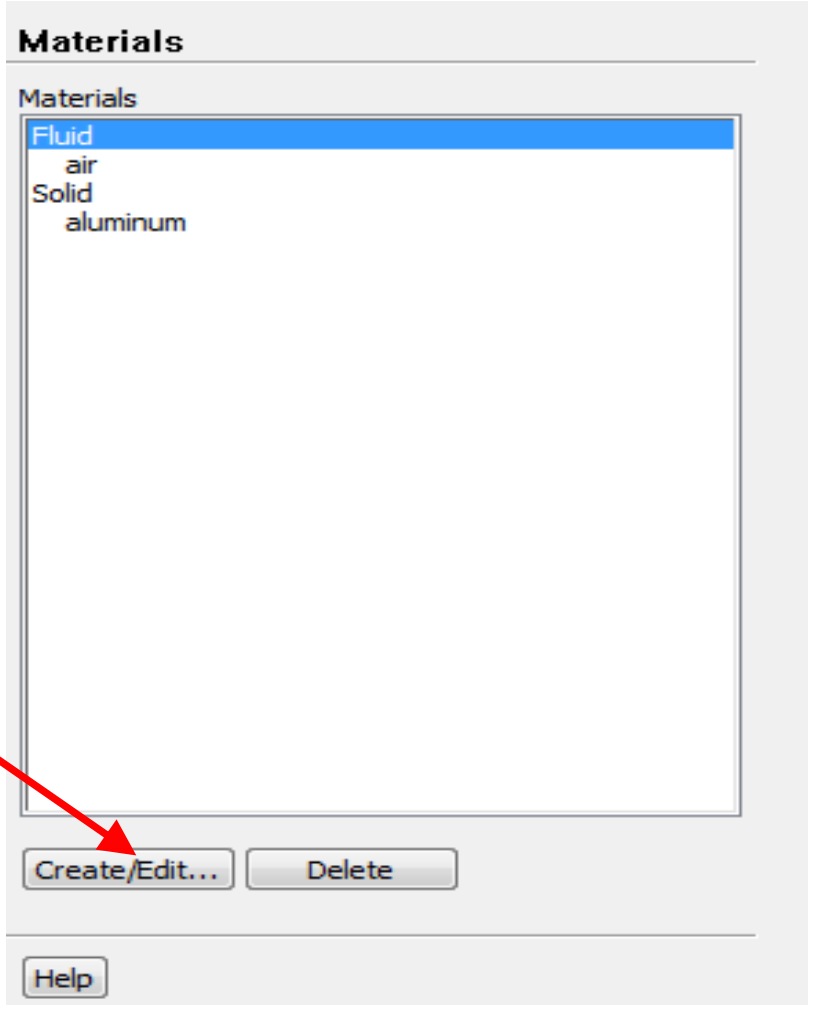
Step 4: Define the material properties

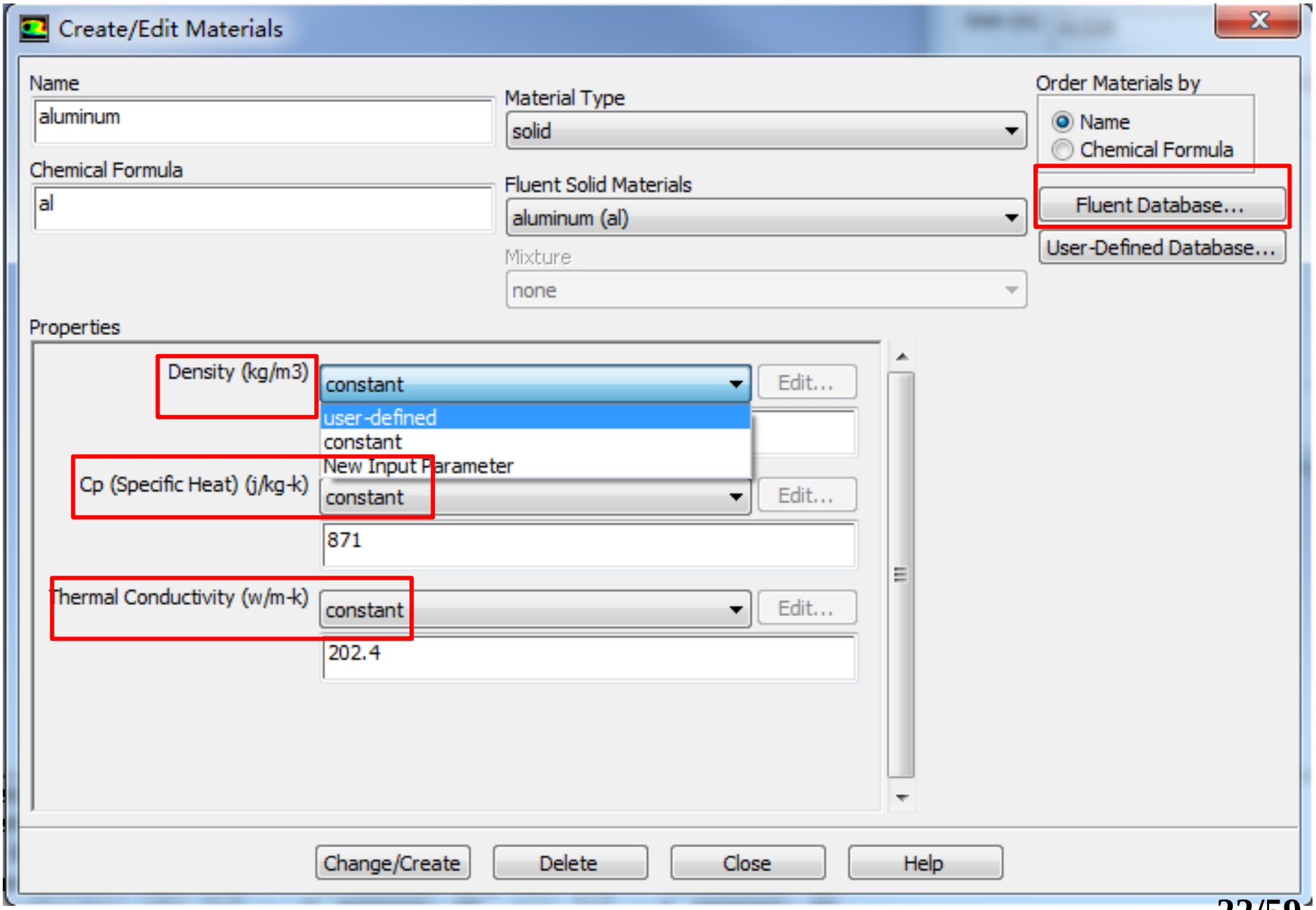
Define the properties required for modeling! For pure heat conduction problem studied here, ρ, Cp and λ should be defined.

Solution Setup→Materials

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

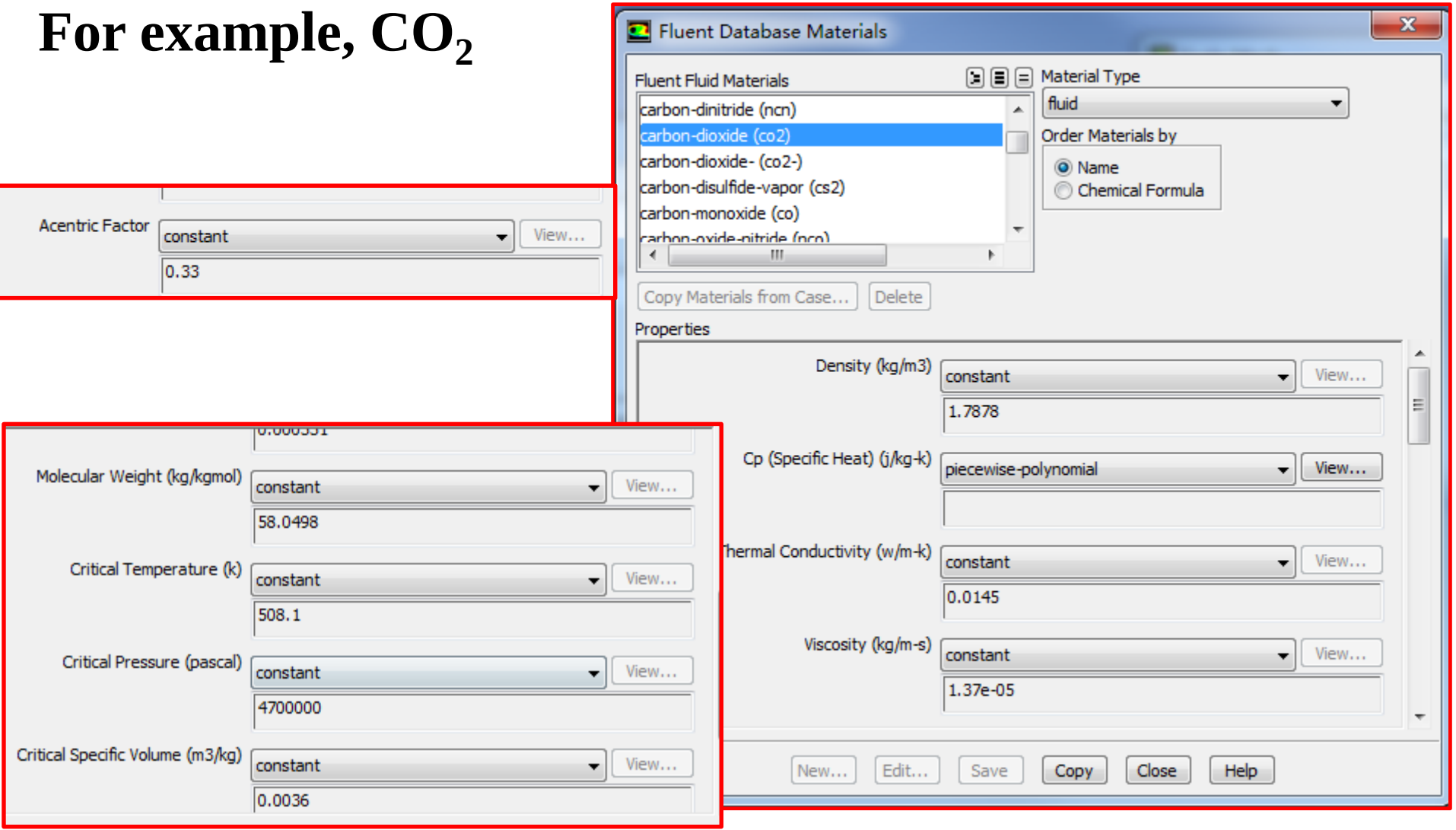
Click the Create/Edit button to find or define new material!



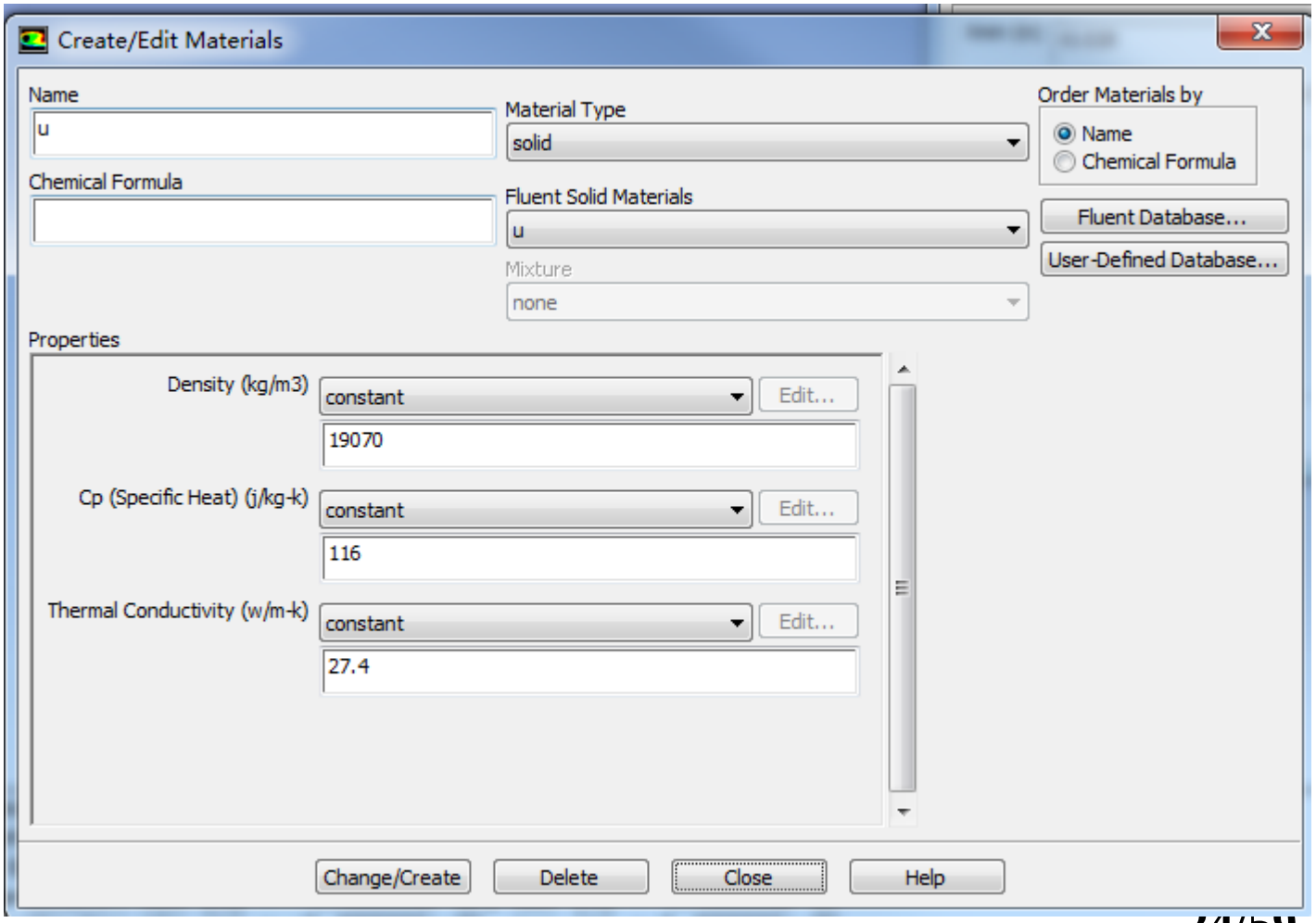

Create/Edit Materials
Name
aluminum
Material Type
solid
Order Materials by
Name
Chemical Formula
Chemical Formula
al
Fluent Solid Materials
aluminum (al)
Fluent Database...
User-Defined Database...
Mixture
none
Properties
Density (kg/m3)
constant
Edit...
user-defined
constant
New Input Parameter
Cp (Specific Heat) (j/kg-k)
constant
Edit...
871
Thermal Conductivity (w/m-k)
constant
Edit...
202.4
Change/Create
Delete
Close
Help

Fluent provide a lot of materials in its database. Usually, You can find the material you need in the database.

For example, CO_2



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

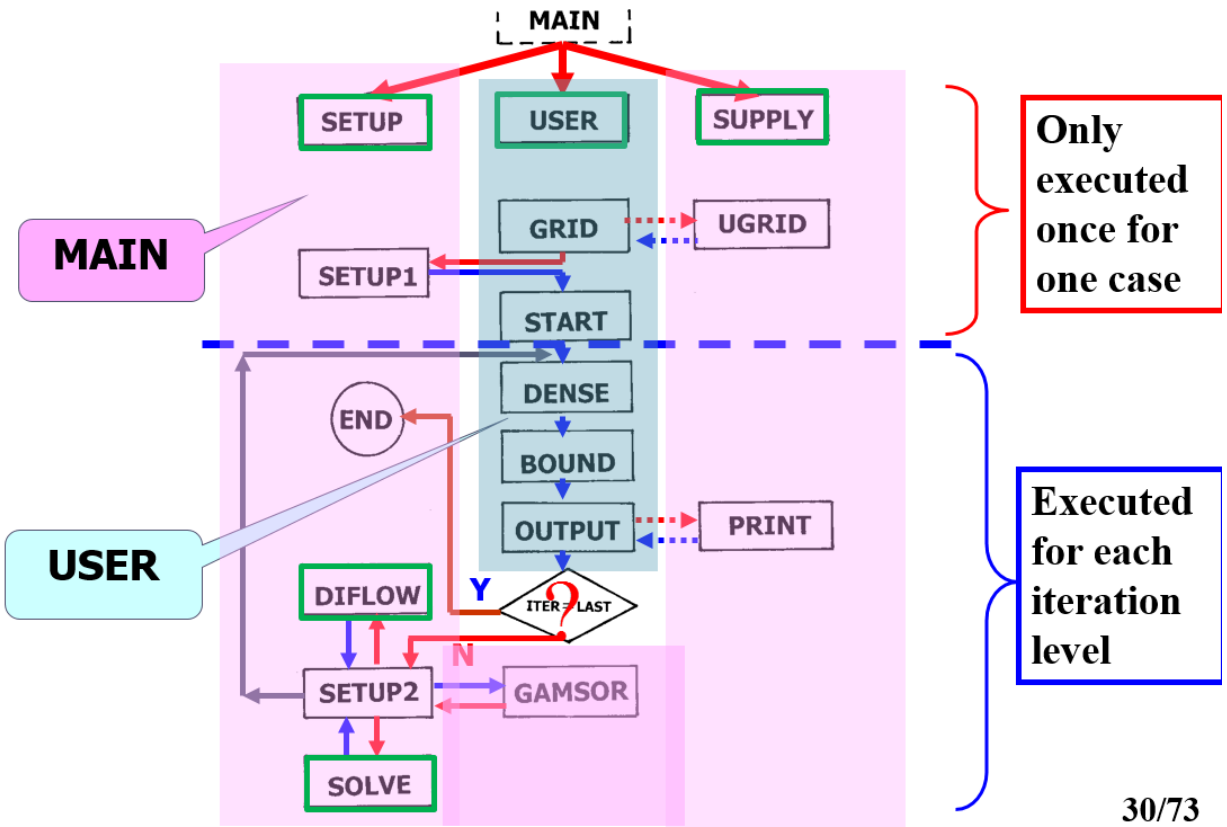


Our general Code:

12. GAMSOR

(1) Determine Γ_ϕ for different variables:

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



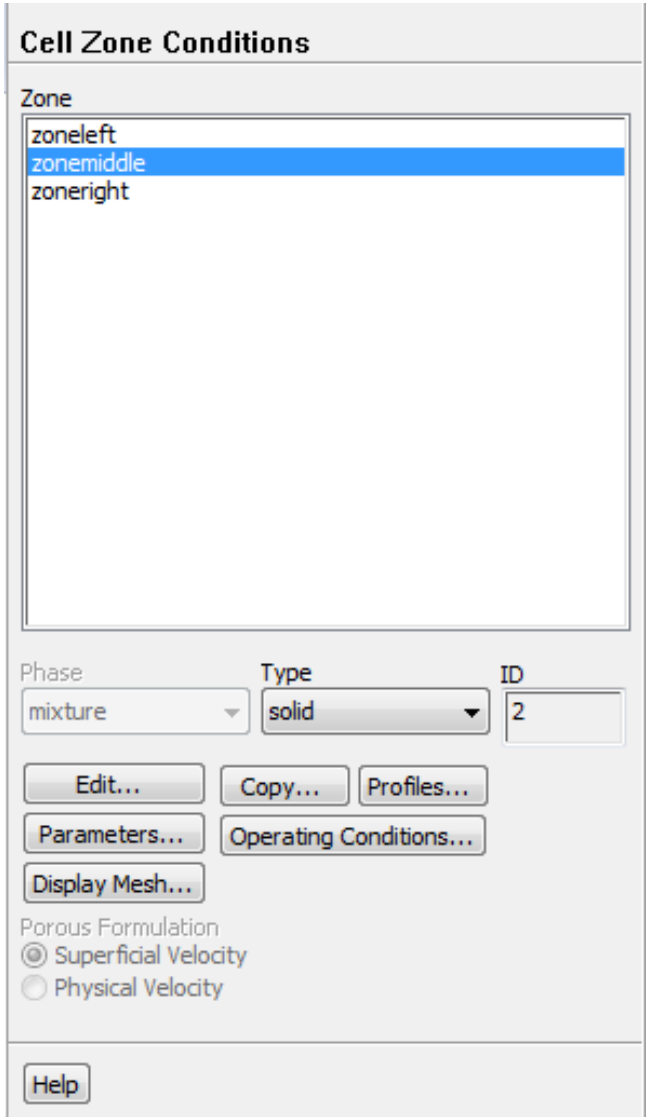
Step 5: Define zone condition

Solution Setup→Cell Zone Condition

Zone type:

- **Fluid**
- **Solid**

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.

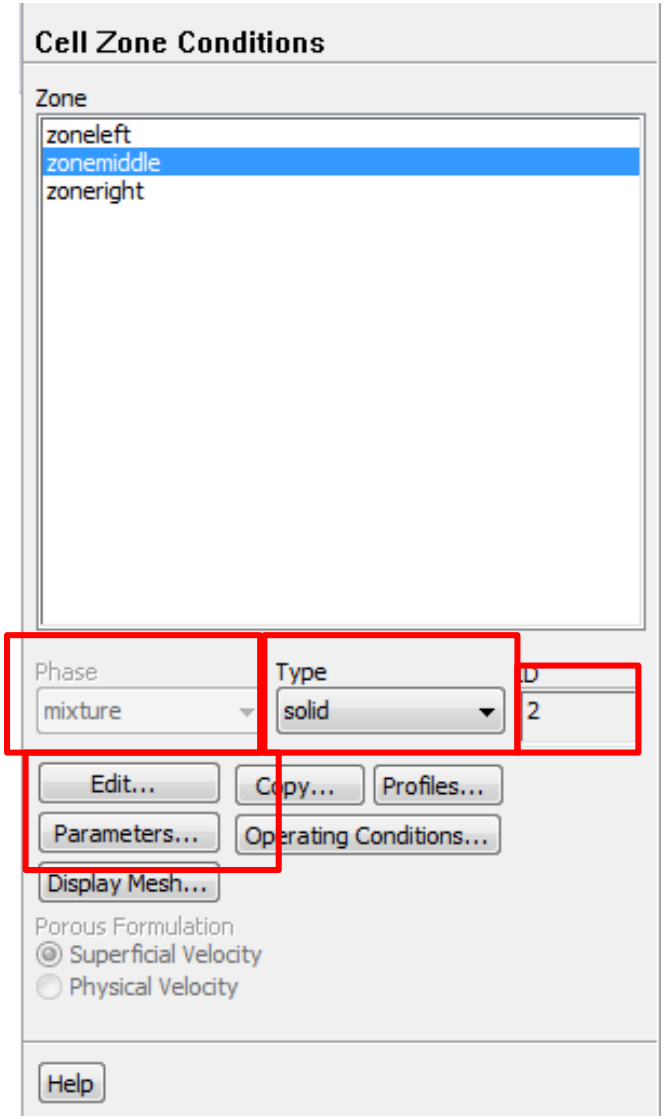


Each zone has its ID.

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

Phase is not activated here. It can be edited under other cases such as when multiphase（多相流） flow model is activated.

Click Edit to define the zone condition of each zone.

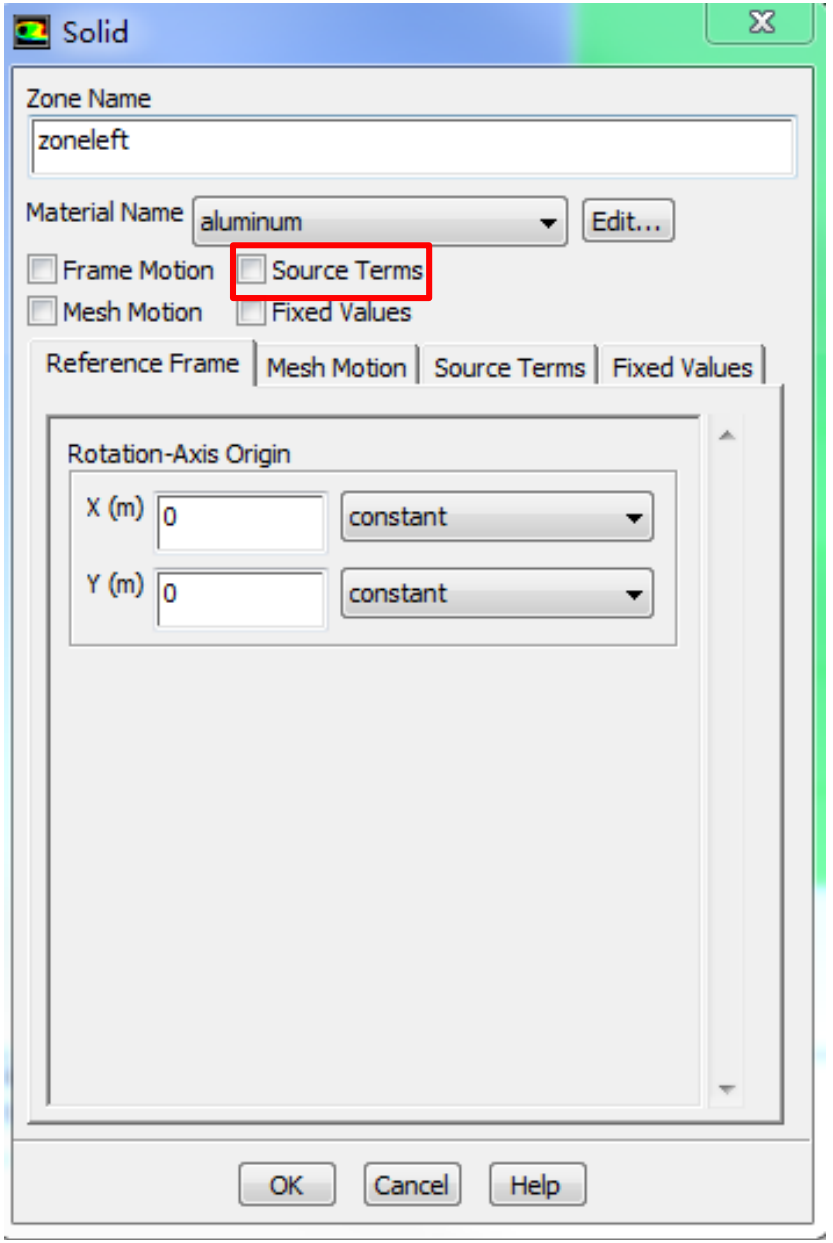


A kind of material should be given to the zone, such as Al here for the two out layers.

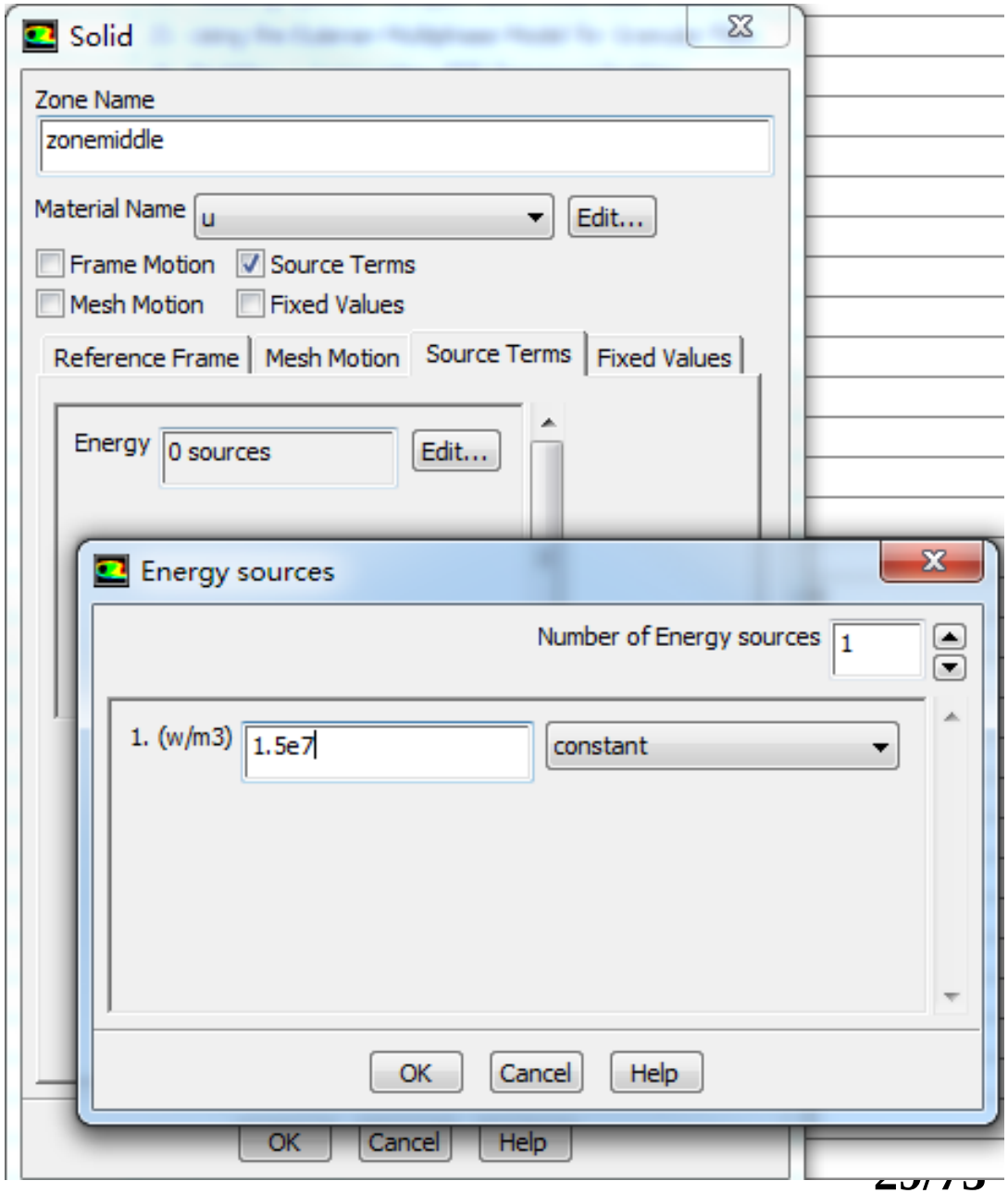
Frame motion and Mesh motion is used if the solid or the frame is moving.

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

Source term should be selected if there is a source term in the zone.



For the middle zone, source term is activated, and the value of the source term is input in the **Energy Sources** interface.



Remark: Source term is given in GAMSOR, which is linearized.

$$S = S_C + S_P \phi_P \,,\; S_P \le 0$$

Remark: In Fluent, if the source term is not a constant and is a function of the variable solved, local linearization of source term is also adopted.

$$S = S_C + S_P \phi_P \,,\; S_P \le 0$$

Specifying a value for S_p can enhance the stability of the solution and help convergence rates due to the increase in diagonal terms on the solution matrix.

$$S_c = S^* - (\partial S/\partial \emptyset)^* \emptyset^*, S_p = (\partial S/\partial \emptyset)^*$$

For general source term that is not a constant, user defined function (UDF) is required in Fluent. Define_Source is adopted to specify custom source term for different transport equations.

Step 6: Define the boundary condition

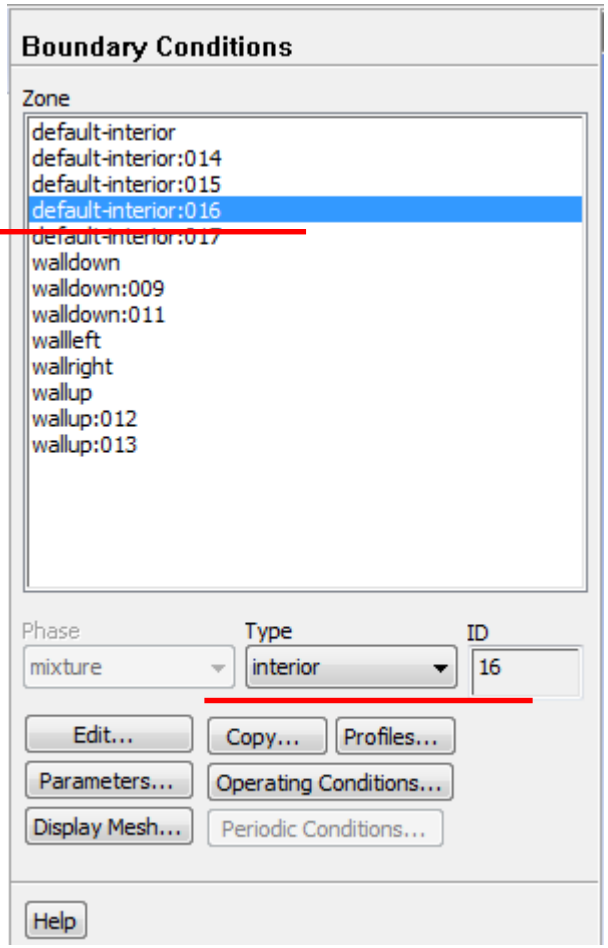
Boundary condition definition is one of the most important and difficult step during Fluent simulation. General boundary conditions in Fluent can be divided into two kinds:

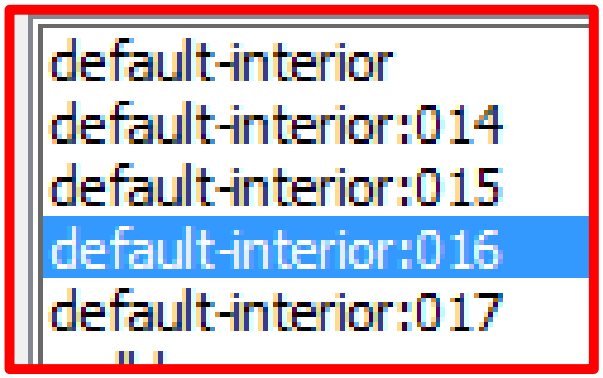
1. BC at inlet and outlet: pressure, velocity, mass flow rate, outflow…

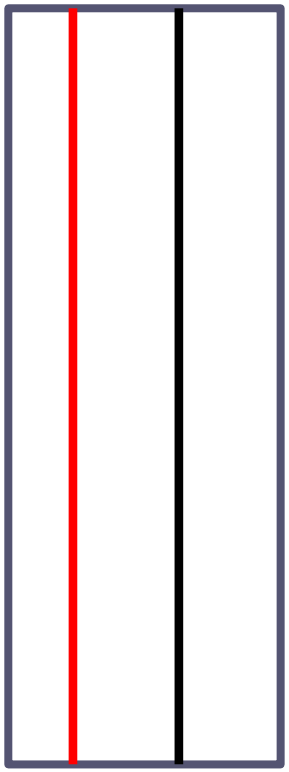
2. BC at wall: wall, periodic, symmetric…

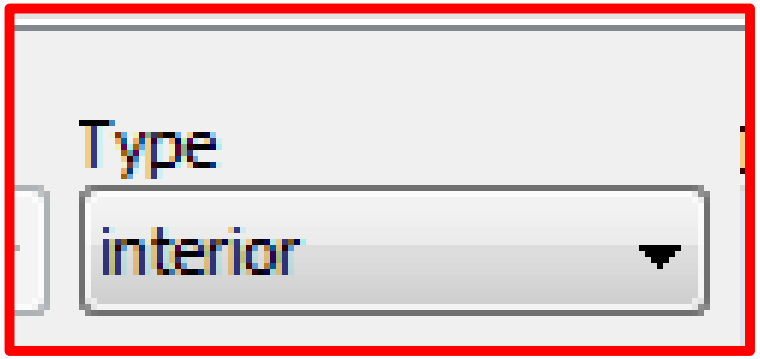
Remark: Interior cell zone and interior interface will also shown in the BC Window.

For example, Default-interior:016 is also listed here. However, it is not a Boundary of the domain. Its type is Interior here.



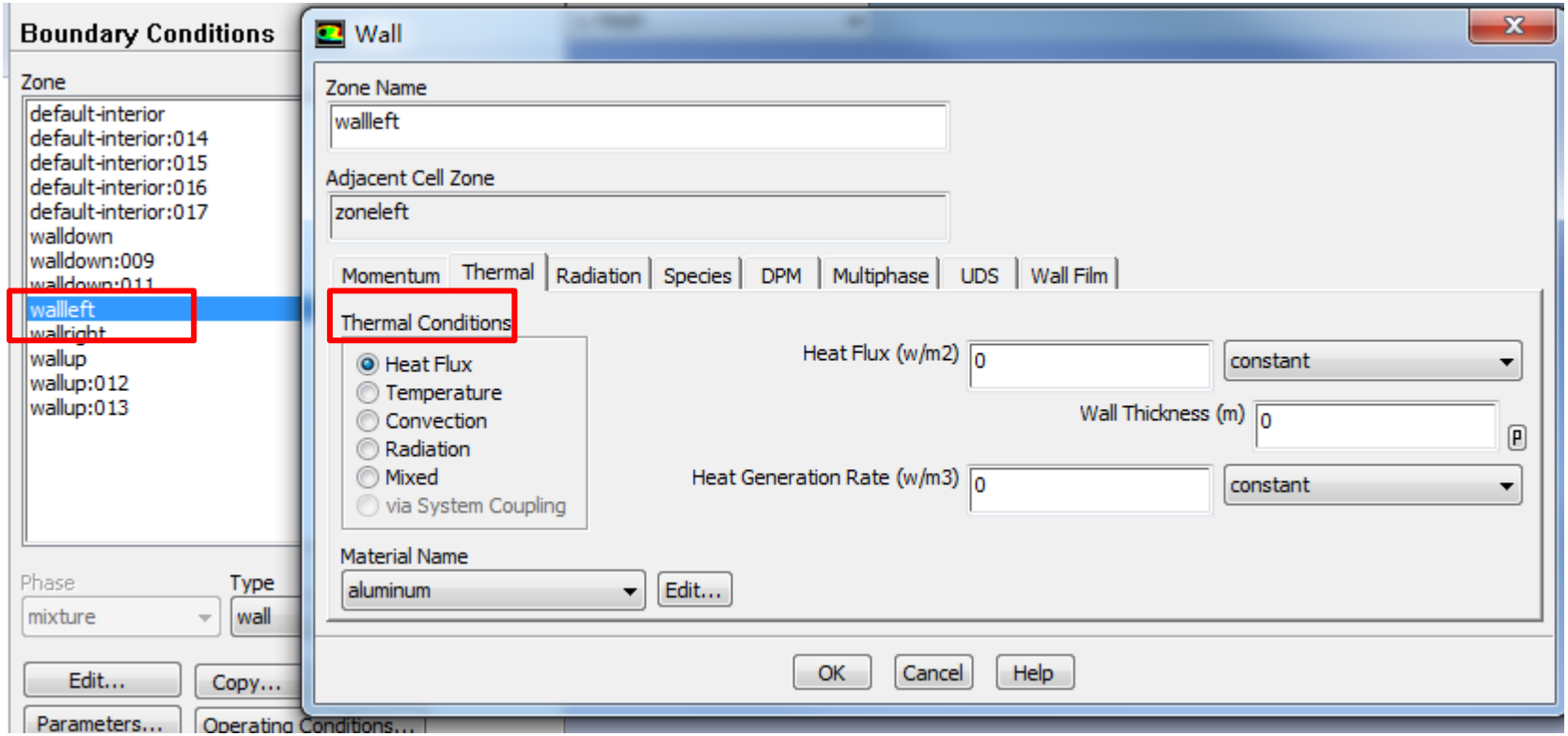






Here, only the BCs related to the heat conduction problem studied here are introduced. Other types of BCs will be introduced in other examples.

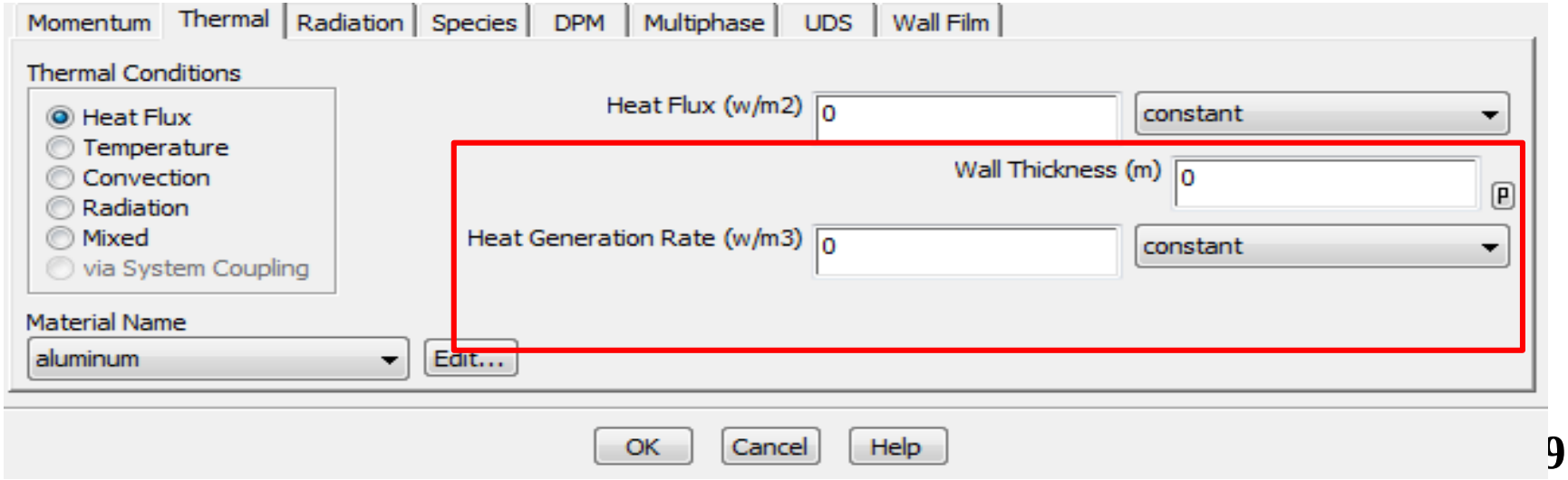
Solution Setup→Boundary conditions



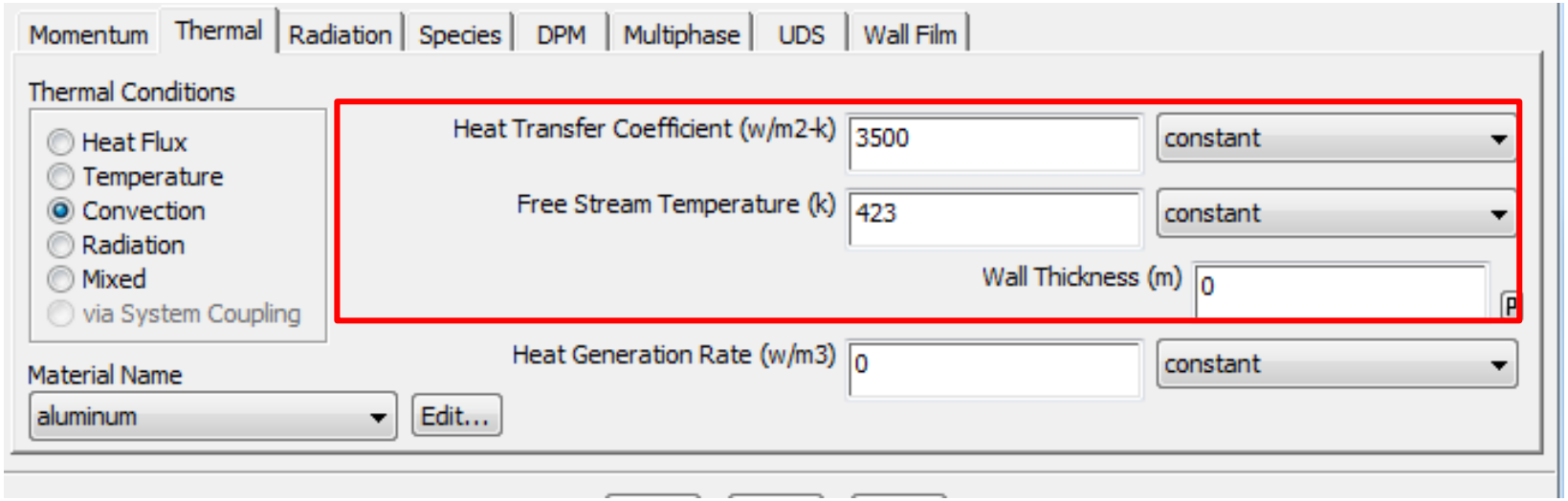
1. **Heat Flux (2nd BC):**
2. **Temperature (1st BC)**
3. **Convection (3rd BC)**

Input the related values

Here, if the wall thickness is greater than zero, and you describe it as a line in your model, a heat generation rate within the wall should also be given. You should also assign the wall a kind of material.



h, Tf *h*=3500, *Tf*=150

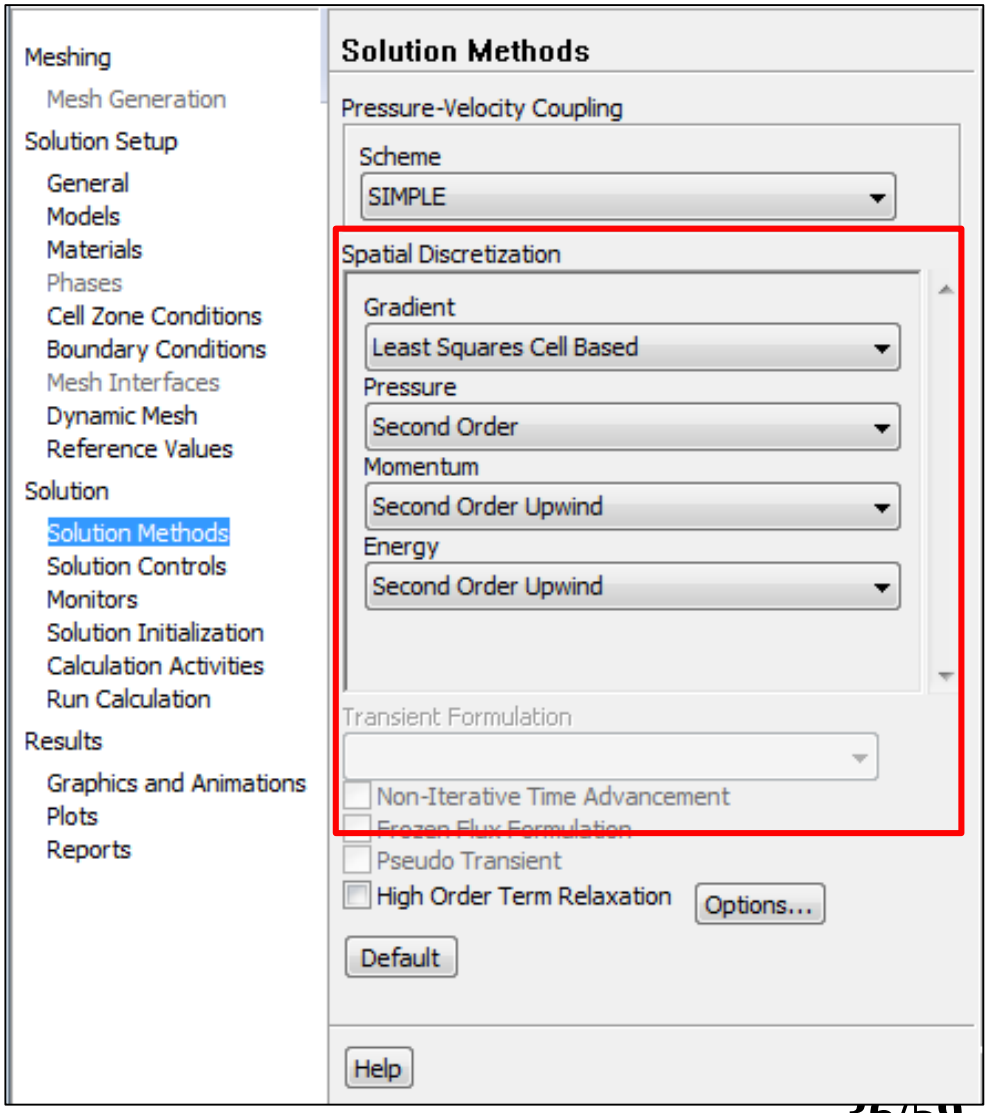


Remark: **Other types of thermal BC in Fluent include radiation BC, mixed BC(combined radiation and convection BC), thermal conditions for two-sides walls, shell conduction…. For more details, you can refer to the Help File of Fluent.**

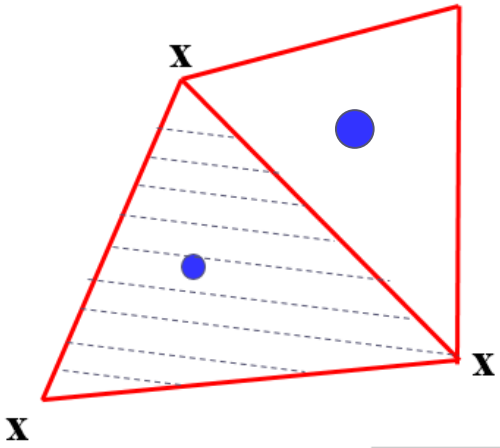
Step 7: Solution setup: algorithm and scheme

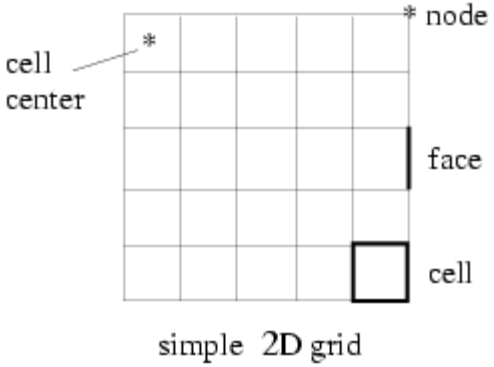
Remark: In Fluent, for the SIMPLE series algorithms, only SIMPLE and SIMPLEC are included.

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



Remark: **Difference between the terminology in our NHT and Fluent software about the mesh information.**



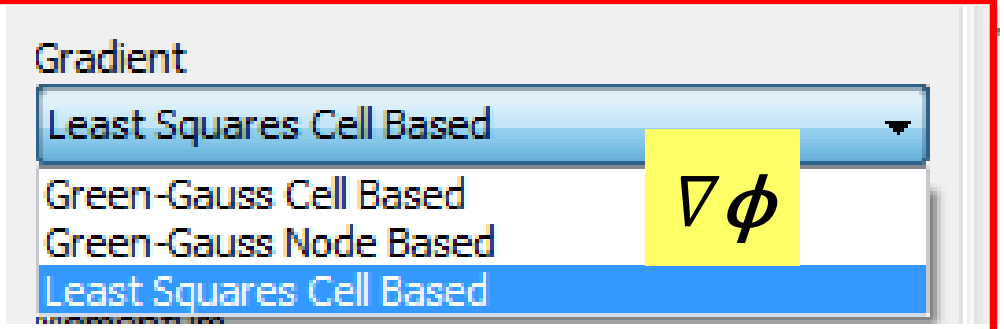


Our NHT	Fluent
Node/cell center	Cell center
x	Node
Interface	Interior face

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

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:

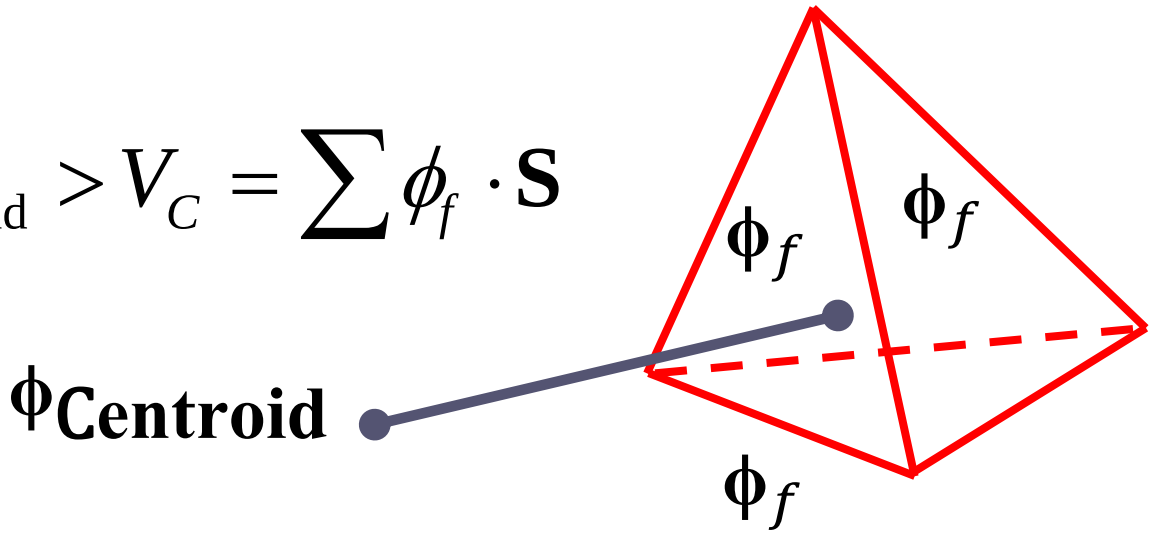
$$<\nabla\phi>=\frac{1}{V_C}\int_{V_C}\nabla\phi dV$$

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:

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



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

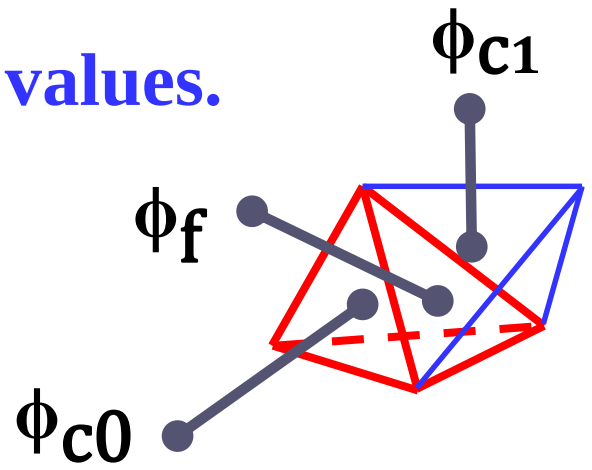
The problem of calculating gradient is transferred into the following equation:

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.

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



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

Calculate ϕ_f by the average of the node values. (代数平均值)

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

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

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 nods C_i,

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

True value — Calculated value

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

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

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

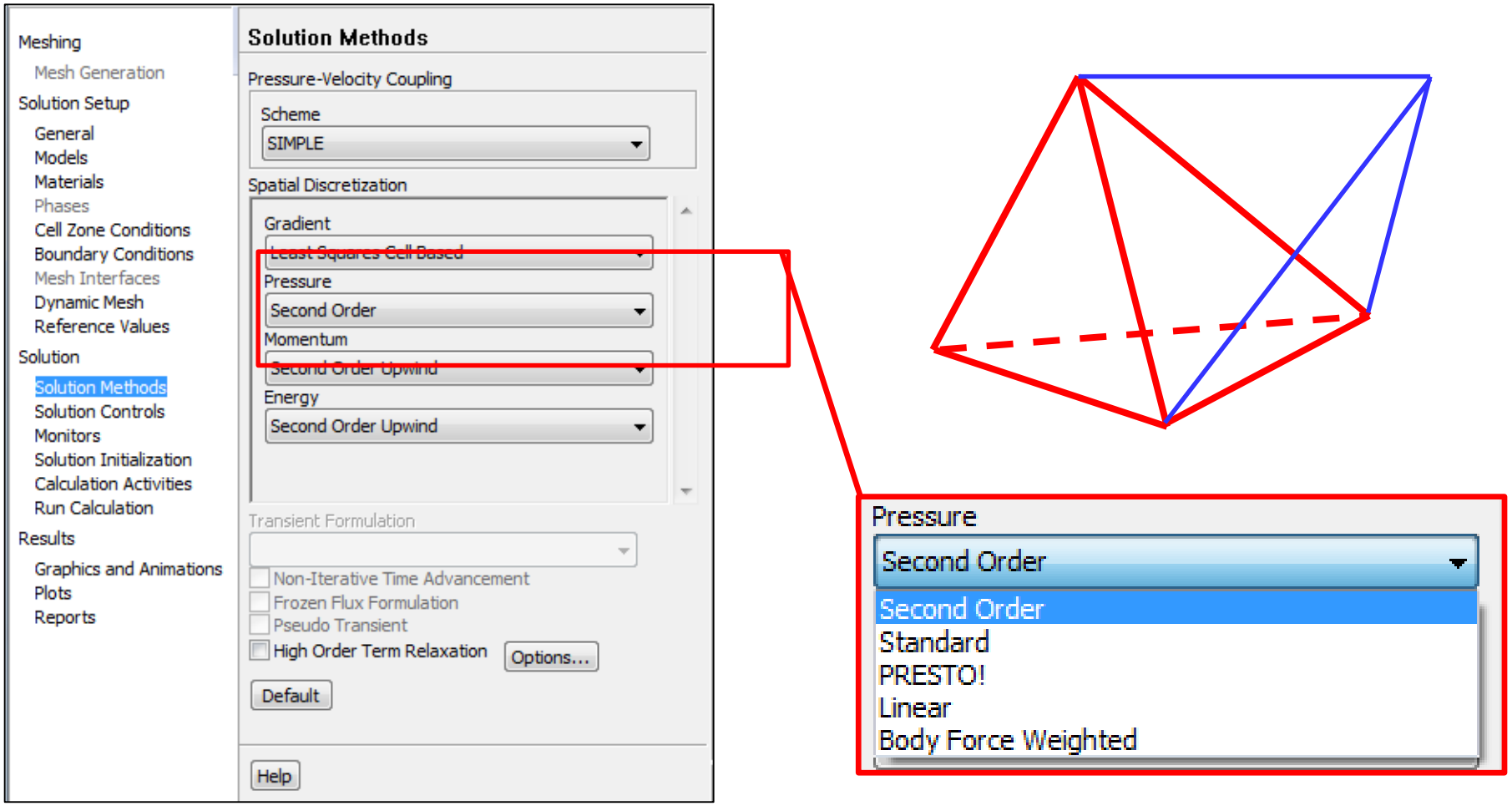
Therefore, to calculate the gradient $\nabla\phi$ is to find the one leading to the minimum ξ!

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

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 less expensive to compute the least-squares gradient than 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.

$$P_f = \frac{\frac{P_{c_0}}{a_{p,c_0}} + \frac{P_{c_1}}{a_{p,c_1}}}{\frac{1}{a_{p,c_0}} + \frac{1}{a_{p,c_1}}}$$

3. Second Order

Calculate the pressure value using a central difference scheme

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

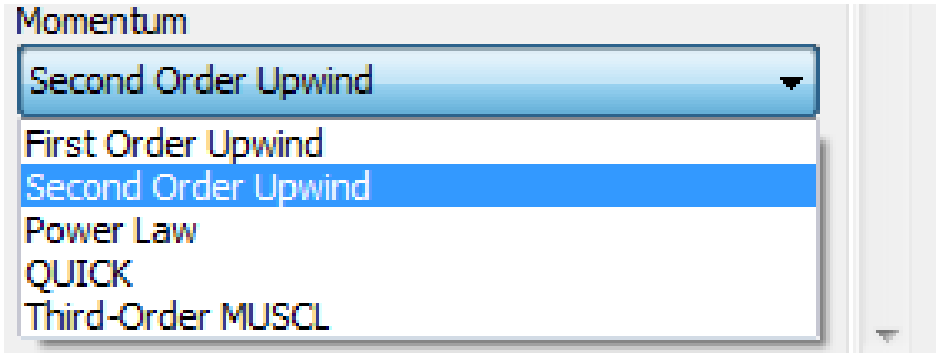
4. Body Force Weighted scheme

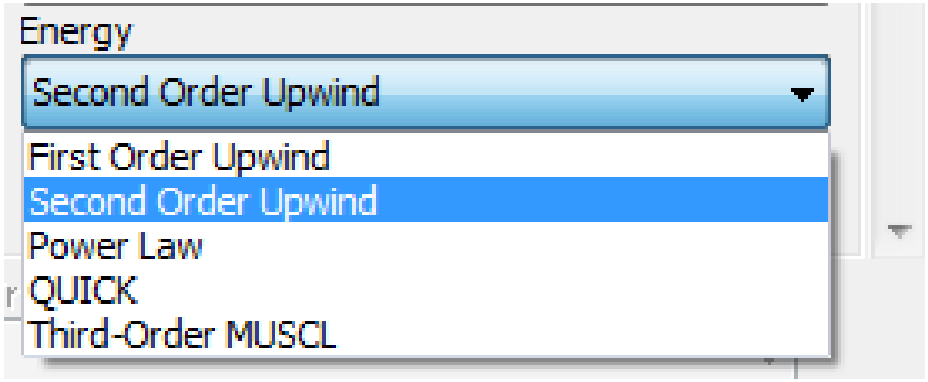
Calculate the pressure according to the body force. For multiphase flow such as VOF (Volume of Fluid，体积函数法） or LS (Level Set, 水平集），it is recommended. Not for porous media!

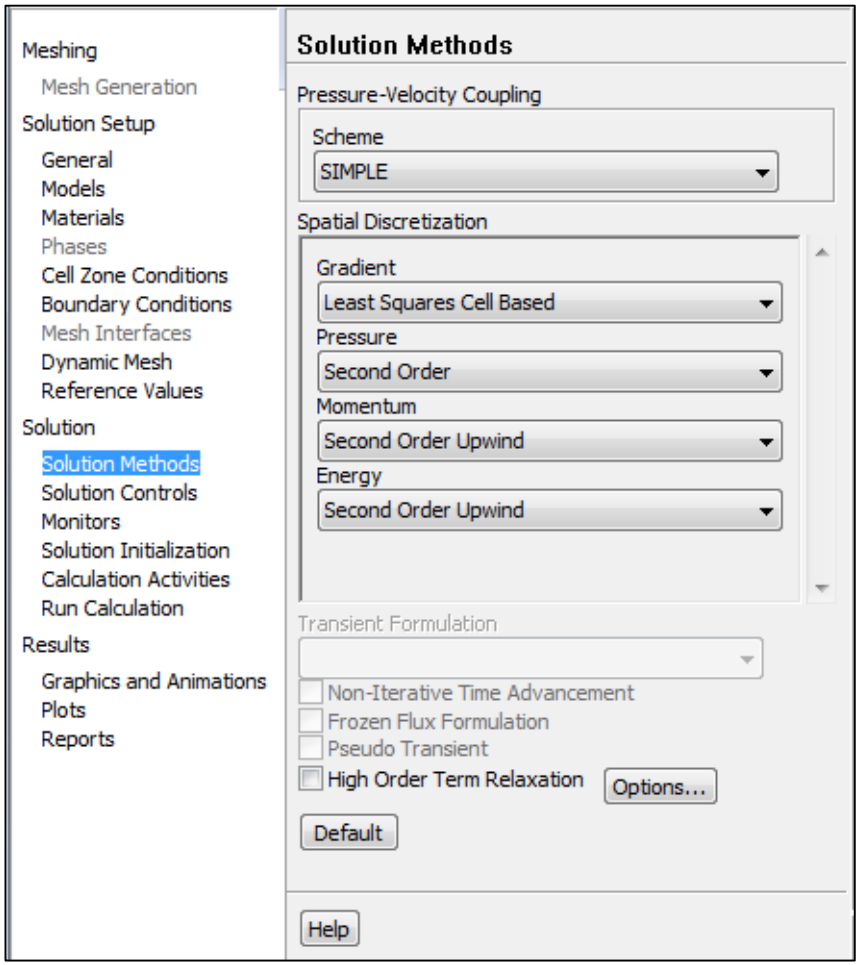
5. PRESTO! (Pressure Staggering Option) scheme

For problem with high pressure gradient.

For convective term scheme, we are very familiar!



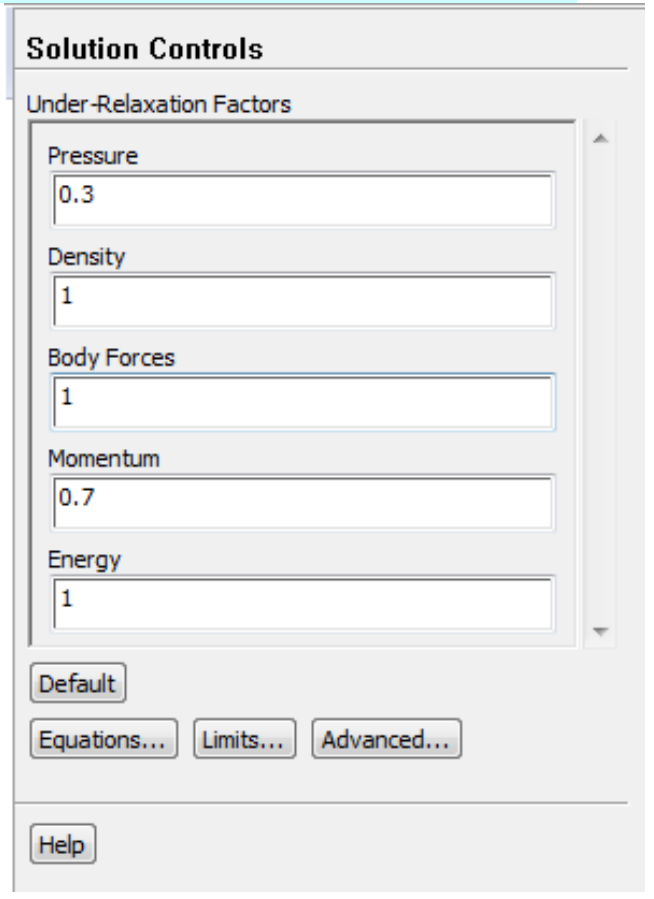




Step 7: Solution setup: relaxation

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!

Remark: In our teaching code, except the pressure correction equation, under-relaxation of other equations are implemented into the solution process.

$$\phi = \phi_P^0 + \alpha[\frac{\sum a_{nb}\phi_{nb} + b}{a_P} - \phi_P^0]$$

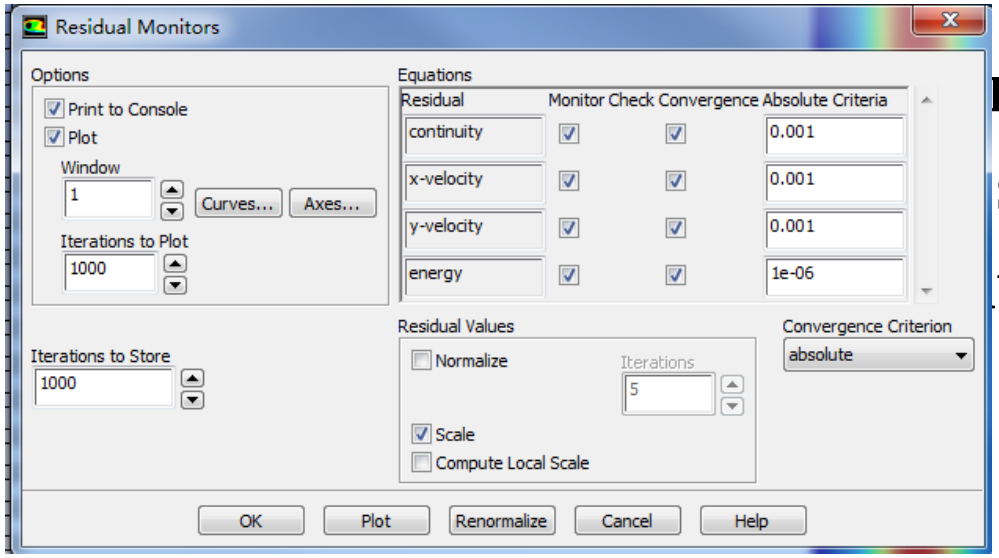
$$(\frac{a_P}{\alpha})\phi_P = \sum a_{nb}\phi_{nb} + b + (1-\alpha)\frac{a_P}{\alpha}\phi_P^0$$

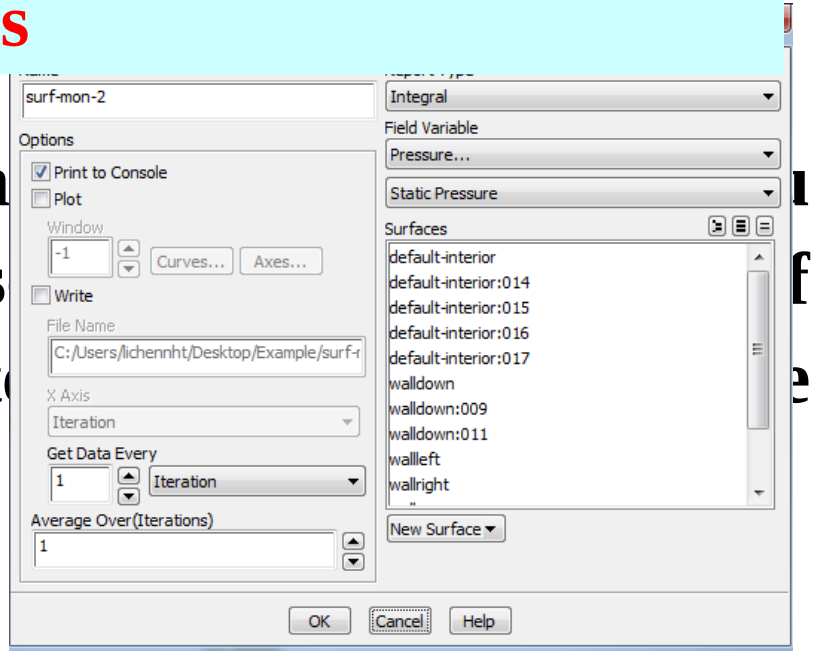
New a_P, $a_P^{'}$

New ***b***,

$$a_P'\phi_P = \sum a_{nb}\phi_{nb} + b'$$

Step 7: Solution setup: monitors

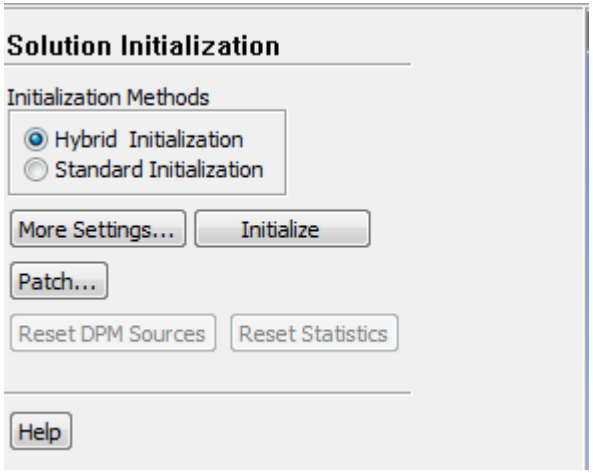


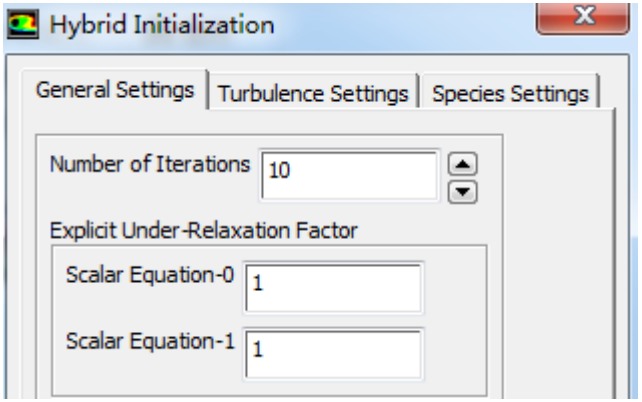


The Residuals are the most important values to be monitored. You can double click it to set the related values.

You can also set other variables to monitor, such as temperature at a certain point.

Step 8: Initialization



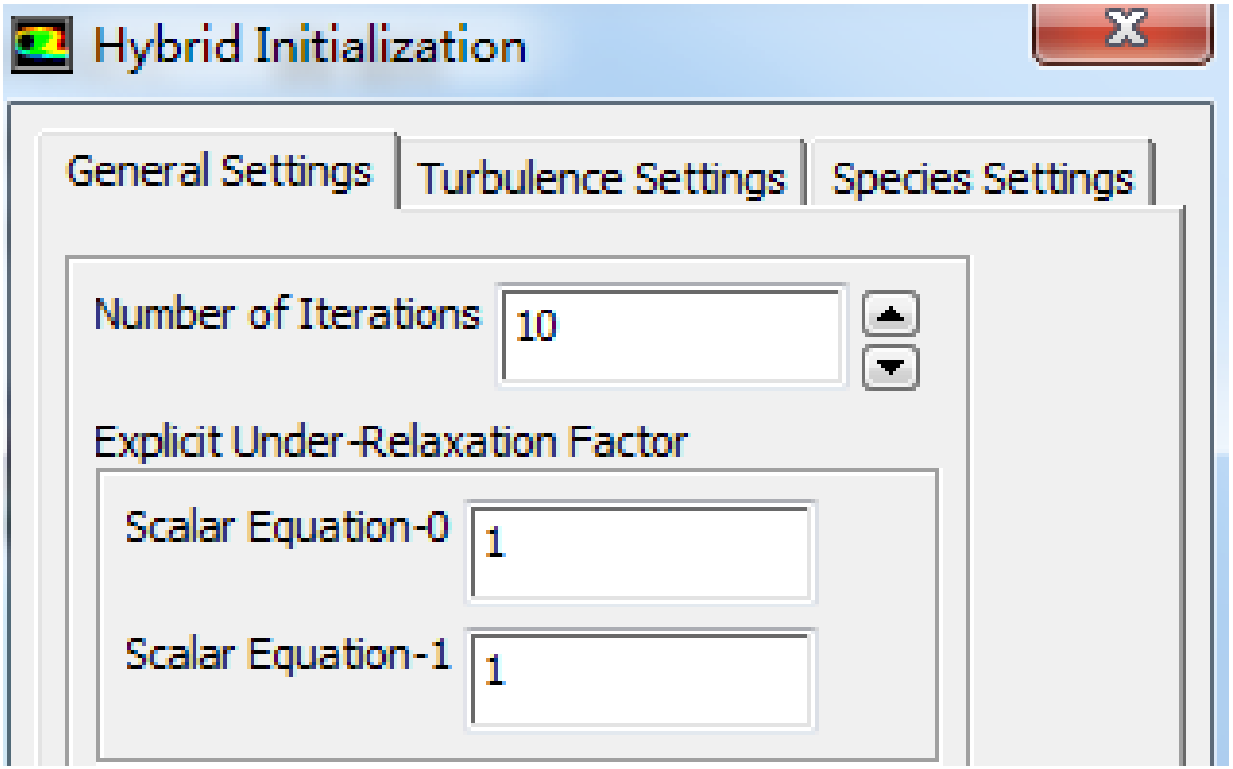


The default selection is Hybrid initialization (混合初始化).

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

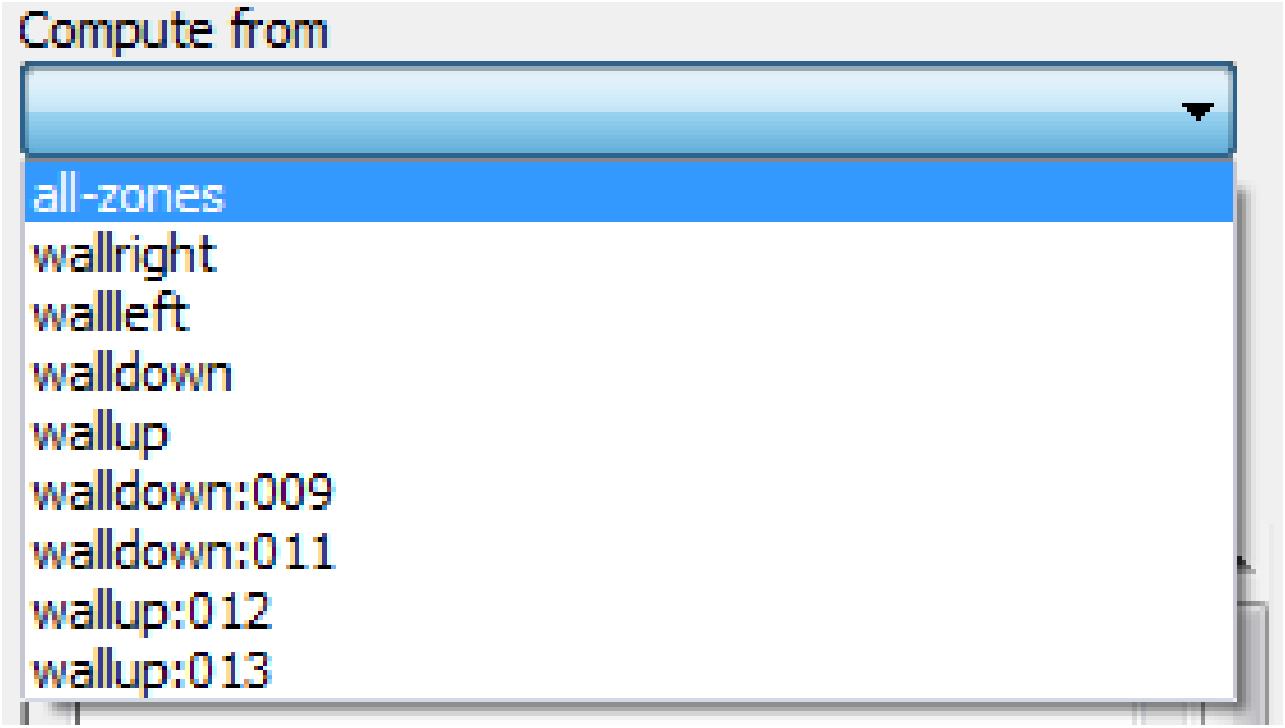
In SIMPLER algorithm, we solved an additional Poisson equation for pressure based on given velocity.

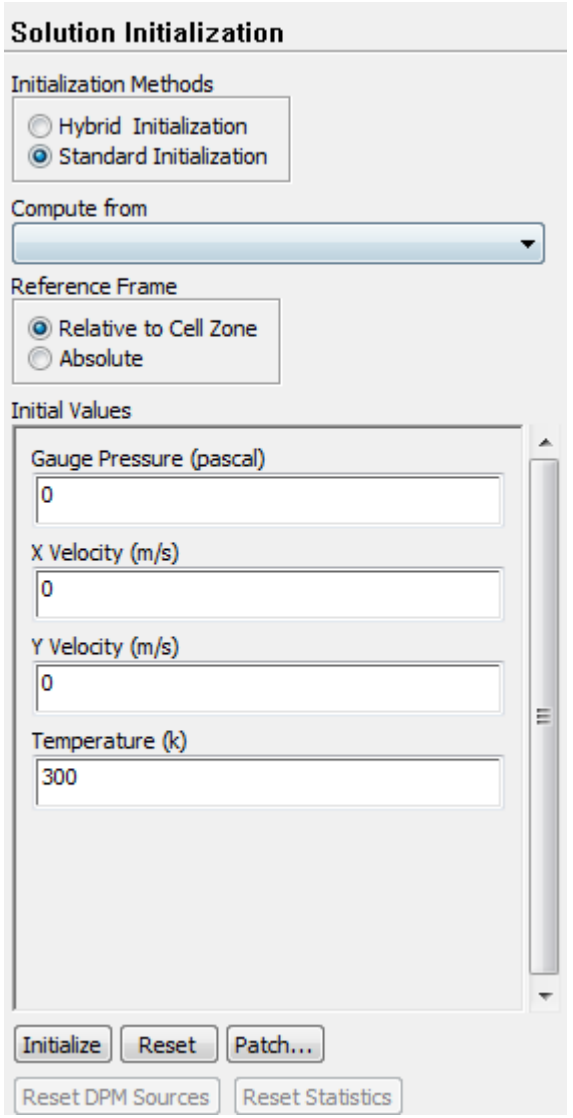
The Hybrid initialization method is similar that Poisson equation is solved to initialize the velocity and pressure equation. You can set the number of iterations to make sure the initial velocity and pressure are consistent.



Or you can simply chose Standard initialization method.

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



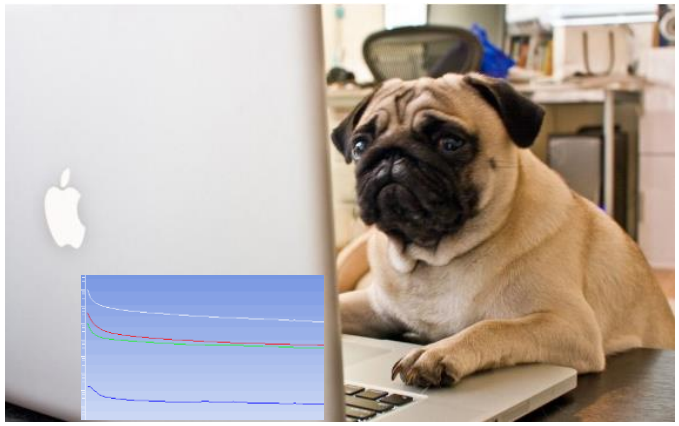


The eight steps for preparing a Fluent simulation have been completed!

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.

Step 10: Result post-processing

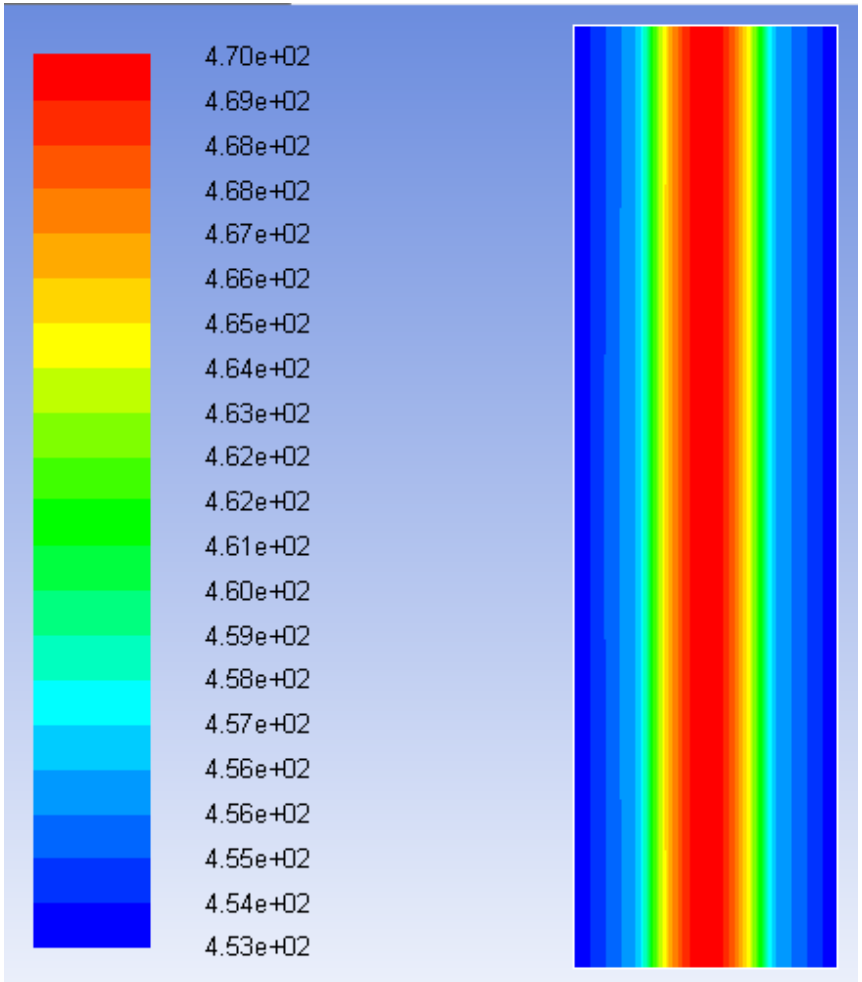


Fig.2 Temperature distribution

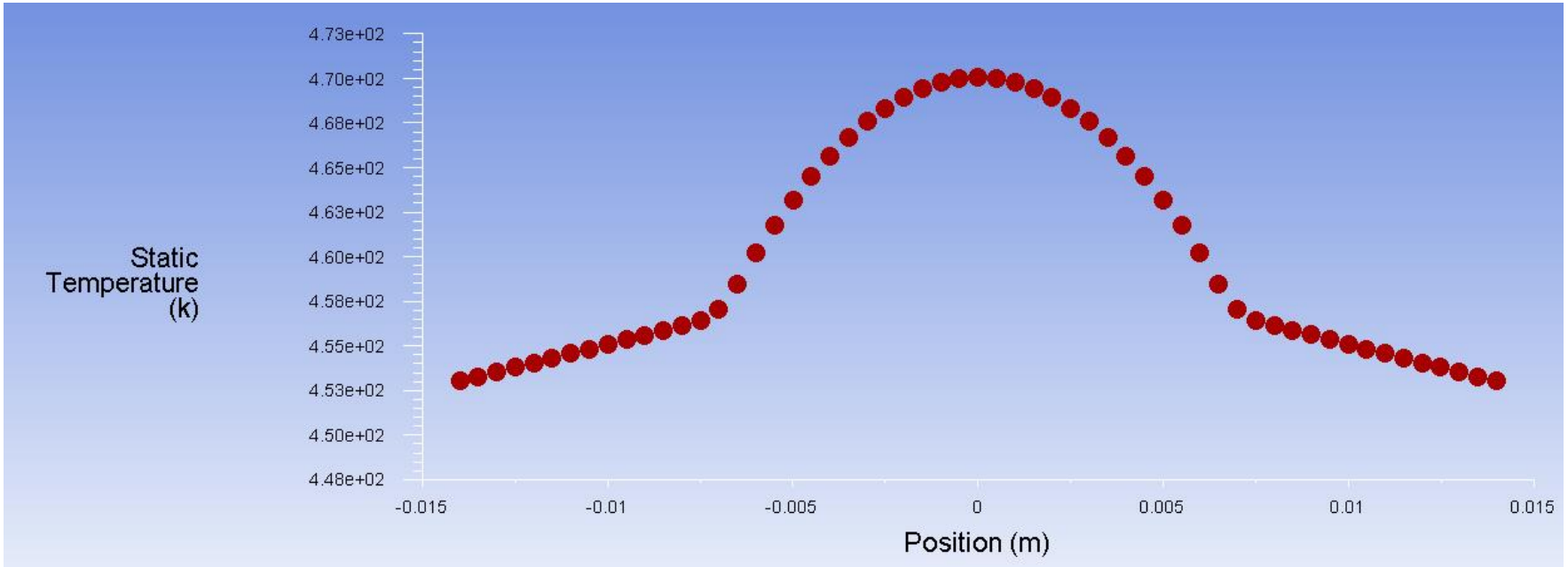


Fig.3 Temperature along *x* direction at half height

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.

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

Uranium: density: 19090 kg/m3; Cp: 116 J/(kg.K)
Thermal conductivity: 27.4 W/(m.K)