



## **Numerical Heat Transfer**

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



#### Instructor Wen-Quan Tao; Qinlong Ren; Li Chen

#### CFD-NHT-EHT Center Key Laboratory of Thermo-Fluid Science & Engineering Xi'an Jiaotong University Xi'an, 2021-Dec.-22





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

箱讲任条龙,除 黎
西安交通大学能源与动力工程学院
热流科学与工程教育部重点实验室
2021年12月22日,西安



### 第 13 章 求解流动换热问题的Fluent软件基础应用举例

**13.1 Conductive heat transfer in a heat sink** 

**13.2 Unsteady cooling process of a steel ball** 

**13.3 Flow and heat transfer in a micro-channel** 

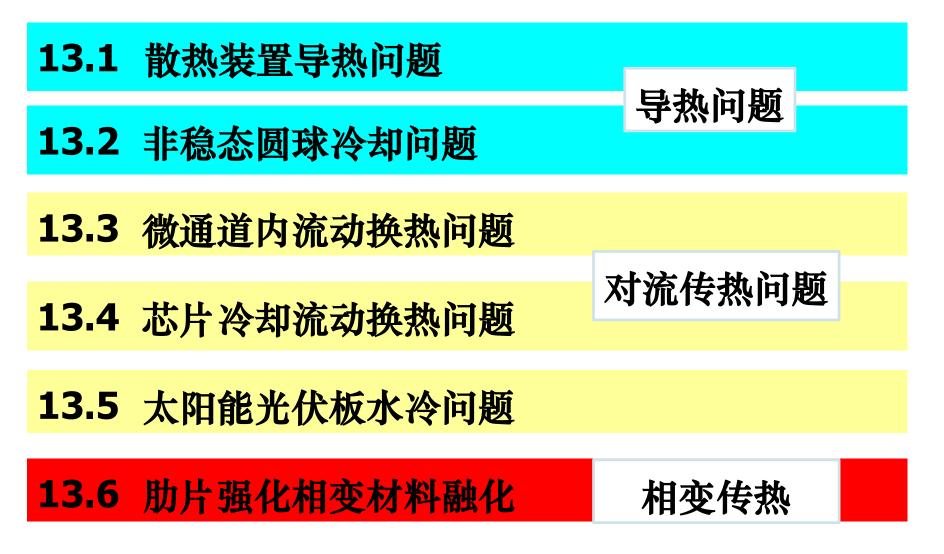
**13.4** Flow and heat transfer in chip cooling

**13.5 Liquid cooling of photovoltaic panel** 

**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





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
- 3. Choose model
- **5. Define zone condition**
- 7. Solution
- 9. Run the simulation.

- 2. Scale domain
- **4. Define material**
- 6. Define boundary condition
- 8. Initialization
- **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) 5/58





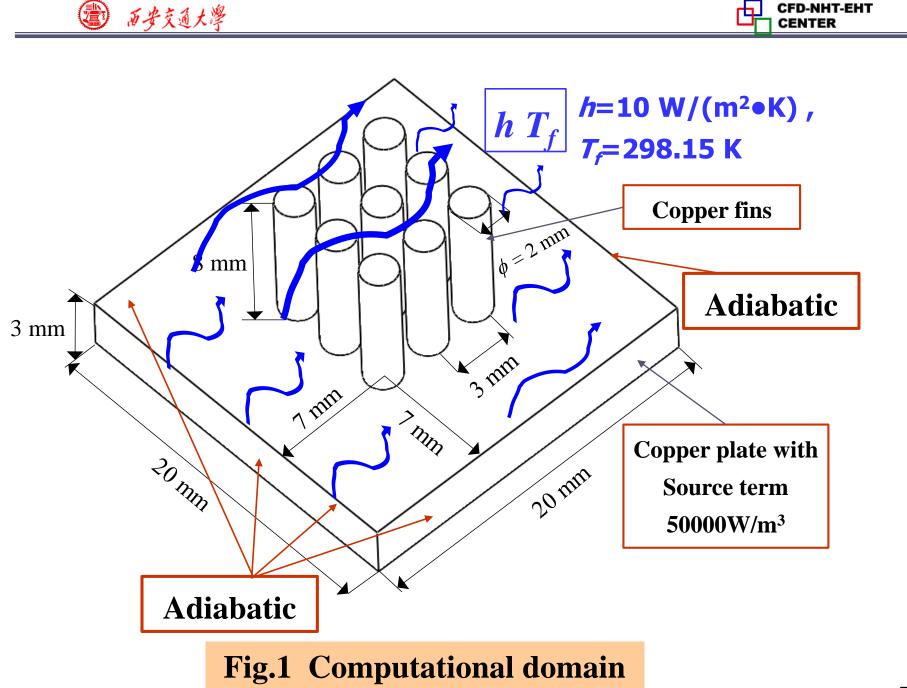
#### **13.1 Heat transfer with source term**

**Known:** Steady heat conduction occurs in a plate with source term using pin fins. The solid material is copper. The domain and size is shown in Fig. 1. The boundary conditions are as follows:

- Boundaries of plate except top surface---Adiabatic
- Plate top surface and fin surfaces---Convective heat transfer

Heat transfer coefficient: h=10 W/(m<sup>2</sup>K); Fluid temperature:  $T_f=298.15$  K

In the copper plate-- a constant source term





Solution:



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

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

It is a heat conduction problem with given GAMA and source term.

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.



CFD-NHT-EHT

#### **Start the Fluent software**

Fluent Launcher

<b>ANSYS</b>						
Dimension 2D 3D Display Options Display Mesh Workbench Co Show Fewer Co				(	Options Double Precision Meshing Mode Processing Options Serial Parallel	
General Options	Parallel Settings	Scheduler	Environment			
Version						

<b>1.</b> C	hoose <b>3-Dimension</b>
<b>2.</b> C	hoose display options
<b>3.</b> C	hoose Serial processing
opti	on

#### **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 difference between various components is high, it is recommended to use Double Precision Version. 9/58

#### **Step 1: Read and check the mesh**

- The mesh is generated by pre-processing software such as ICEM, GAMBIT, or MESHING. The document is with suffix (后缀名) ".msh"
- This step is similar to the Grid subroutine (UGRID, Setup1) in our general teaching code.

	-			· ·
File Setting Up Domain	Setting Up Physics	User-Defined	Solving	Postproce
Read Write	Mesh Case	dra	Combine Separate	■ Delet Deactiv
Import Export	Data Case & Data	•	- Adjacency.	•
Export to CFD-Post	PDF ISAT Table			
Solution Files Interpolate FSI Mapping ♪	DTRM Rays View Factors	ck	Report Qu	ality
Save Picture Data File Quantities Batch Options	Profile Scheme Journal		ity Formuls Absolute	ntion
Exit Solution Methods Solution Controls Monitors Report Definitions	fluent-copper-con Geom-original-01 Geom-original-01 Geom-original-01	duc01 .cas02	Relative	В

💶 fluent-copper-conduc01 Fluent@DESKTOP-KOS20DI [3d, pbns, lam]

#### Mesh→Read

442026	nodes.
245440	hexahedral cells, zone 12.
169740	hexahedral cells, zone 13.
717460	quadrilateral interior faces, zone 14.
497700	quadrilateral interior faces, zone 15.
4140	quadrilateral wall faces, zone 16.
11200	quadrilateral wall faces, zone 17.
7040	quadrilateral wall faces, zone 18.
15340	quadrilateral wall faces, zone 19.
4140	quadrilateral interface faces, zone 20.
14760	quadrilateral wall faces, zone 21.

ilding...

mesh



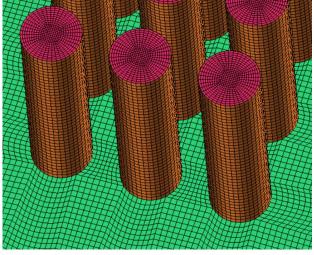
#### **Step 1: Read and check the mesh**

Mesh→Check

#### Check the quality and topological information of the

#### mesh

```
Domain Extents:
    x-coordinate: min (m) = -9.999999e-02, max (m) = 9.999999e-02
    y-coordinate: min (m) = 0.000000e+00, max (m) = 1.100000e-01
    z-coordinate: min (m) = -9.999999e-02, max (m) = 9.999999e-02
Volume statistics:
    minimum volume (m3): 6.009586e-10
    maximum volume (m3): 7.264403e-09
    total volume (m3): 1.425178e-03
Face area statistics:
    minimum face area (m2): 3.079453e-07
    maximum face area (m2): 3.930792e-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.



#### **Step 2: Scale the domain size**

#### **General**→**Scale**

General Neck	Scale Mesh	×
Scale Check Report Quality Display Solver Type Velocity Formulation Pressure-Based Absolute Density-Based Relative Time Steady Transient	Domain Extents         Xmin (m)       -0.09999999         Ymin (m)       0         Ymax (m)       0.11         Zmin (m)       -0.09999999         Zmax (m)       0.09999999         View Length Unit In	Scaling Convert Units Specify Scaling Factors Mesh Was Created In mm Scaling Factors X 0.001 Y 0.001 Z 0.001 Scale Unscale
Gravity Units	Close Help	

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.

**<u>Remark:</u>** 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

Scale Mesh	×	Scale Mesh		×
Domain Extents	Scaling	Domain Extents		Scaling
Xmin (m)         -0.014         Xmax (m)         0.014           Ymin (m)         -0.05         Ymax (m)         0.05	<ul> <li>Convert Units</li> <li>Specify Scaling Factors</li> <li>Mesh Was Created In</li> </ul>	Xmin (m) -0.00014 Ymin (m) -0.0005	Xmax (m)         0.00014           Ymax (m)         0.0005	<ul> <li>Convert Units</li> <li>Specify Scaling Factors</li> <li>Mesh Was Created In</li> <li>cm</li> </ul>
View Length Unit In	cm Scaling Factors X 0.01 Y 0.01 Scale Unscale	view Length Unit In m  ▼		Scaling Factors          X       0.01         Y       0.01         Scale       Unscale
Close Help			Close Help	



CFD-NHT-EHT

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

**<u>Remark:</u>** 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.

Scale Mesh	×	Scale Mesh	X
Domain Extents         Xmin (m)       -0.014         Ymin (m)       -0.05         Ymax (m)       0.014         Ymax (m)       0.05         View Length Unit In       •	Scaling Convert Units Specify Scaling Factors Mesh Was Created In Cm Scaling Factors X 2 Y 1 Scale Unscale	Domain Extents Xmln (m) -0.028 Ymin (m) -0.05 Ymax (m) View Length Unit In m	<ul> <li>Specify Scaling Factors</li> </ul>
Close Help		٩	Close Help

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.

**<u>Remark:</u>** 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. 15/58



CFD-NHT-EHT

## **<u>Remark:</u>** The governing equation of energy solved by Fluent is as follows:

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot (\vec{v}(\rho E + p)) = \nabla \cdot \left(k_{eff}\nabla T - \sum_{j}h_{j}\vec{J}_{j} + (\vec{\tau}_{eff}\cdot\vec{v})\right) + S_{h}$$
$$E = h - \frac{p}{\rho} + \frac{v^{2}}{2}$$
Enthalpy

#### **8.1** Format Improvement of General Governing Equation

$$\begin{split} \frac{\partial(\rho c_p T)}{\partial t} + \frac{\partial(\rho c_p u T)}{\partial x} + \frac{\partial(\rho c_p v T)}{\partial y} &= \frac{\partial}{\partial x} \left( \lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda \frac{\partial T}{\partial y} \right) + S_T. \\ \frac{\partial(\rho T)}{\partial t} + \frac{\partial(\rho u T)}{\partial x} + \frac{\partial(\rho v T)}{\partial y} &= \frac{\partial}{\partial x} \left( \frac{\lambda}{c_p} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{\lambda}{c_p} \frac{\partial T}{\partial y} \right) + \frac{S_T}{c_p} \\ - \frac{1}{c_p^2} \left[ \rho c_p T \frac{\partial c_p}{\partial t} + \left( \rho c_p u T - \lambda \frac{\partial T}{\partial x} \right) \frac{\partial c_p}{\partial x} + \left( \rho c_p v T - \lambda \frac{\partial T}{\partial y} \right) \frac{\partial c_p}{\partial y} \right] \end{split}$$
16/58

#### **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:

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

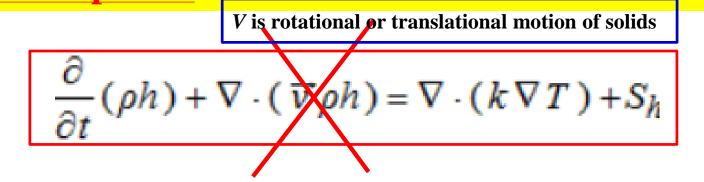
## The new form of G.E. are:

Equation	$ ho^*$	φ	$\Gamma_{\phi}$	$S_{\phi}^{*}$
Continuity equation	ρ	1	0	0
Momentum eqn. ( <i>x</i> direction)	ρ	U	μ	$\rho f_x - \frac{\partial p}{\partial x}$
Momentum eqn. (y direction)	ρ	υ	μ	$\rho f_y - \frac{\partial p}{\partial y}$
Energy equation	$ ho c_p$	Т	λ	$S_T$



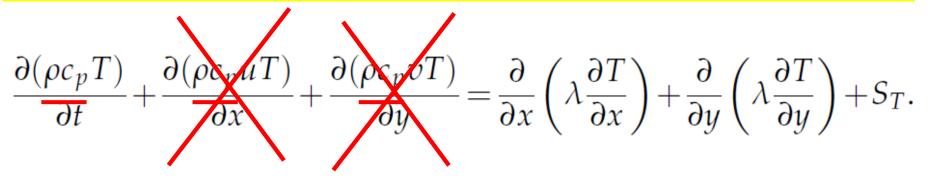


#### **Fluent for solid phase:**



h: enthalpy

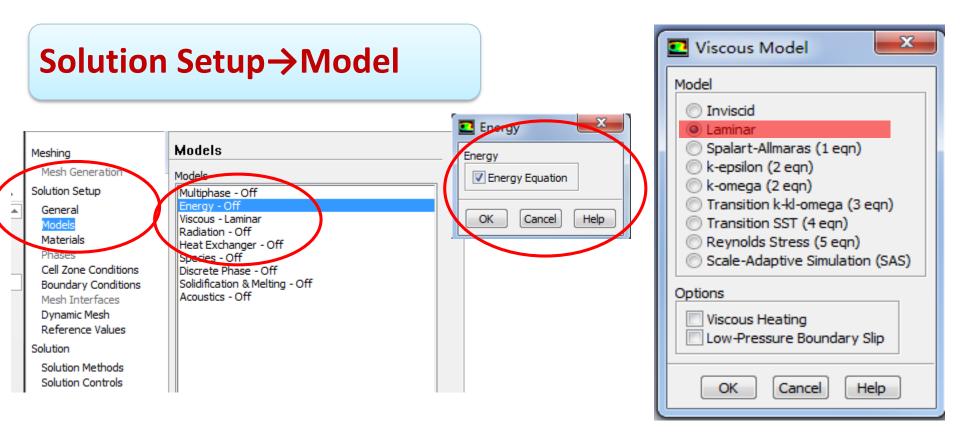
#### **Our general Code:**



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



#### To select the model, the command is as follows:



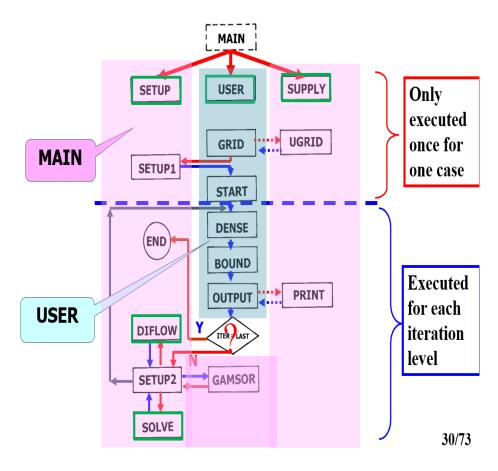
**<u>Remark:</u>** It is interesting to note that no matter what kind of problems you will solve, the model "Viscous-Laminar" is always activated. We can cancel it later. 19/58



#### **<u>Remark:</u>** In our general code,

西安交通大學

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.



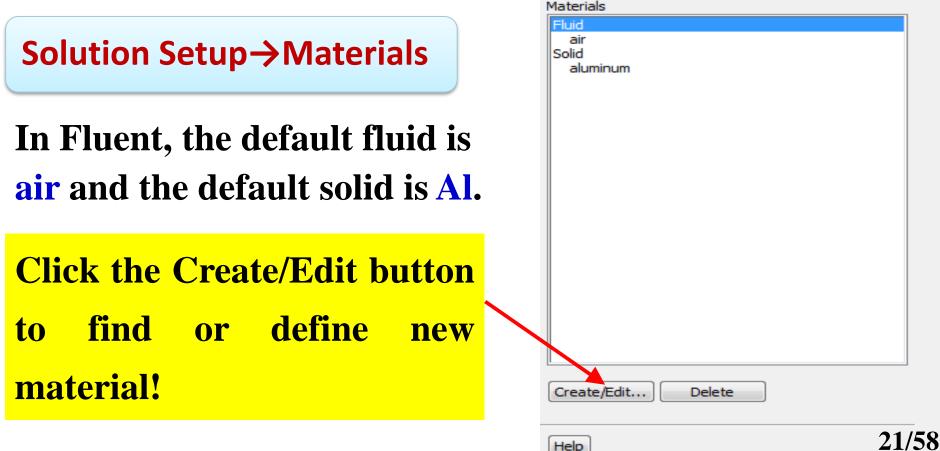
20/58





#### **Step 4: Define the material properties**

Define the properties required for modeling! For pure heat conduction problem studied here,  $\rho$ , Cp and  $\lambda$  should be defined.





ле			
opper	Fluent Solid Materials [1/13]		
emical Formula	calcium-carbonate (caco3)	<u></u>	
u	calcium-oxide (cao)		
	calcium-sulfate (caso4)		
	copper (cu)		
operties	dolomite (cao_mgo_2co2)		
Density (kg/m3) constant	Copy Materials from Case Delete		
8978	Properties		
Cp (Specific Heat) (j/kg-k) constant	Density (kg/m3) constant 💌	View	
Second Se	8978		
301	Cn (Snecific Heat) (i/ke-k) constant	View	
Thermal Conductivity (w/m-k) constant		V1ew	
387.6	381	1	
	Thermal Conductivity (w/m-k) constant	View	
	387.6		
	Electrical Conductivity (siemens/m) constant	View	
	5.8e+07		



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

	Create/Edit Materials		×
Materials Materials Fluid air Solid U aluminum	Name u Chemical Formula	Material Type solid Fluent Solid Materials u	<ul> <li>Order Materials by</li> <li>Name</li> <li>Chemical Formula</li> <li>✓ Fluent Database</li> <li>User-Defined Database</li> </ul>
	Properties	Mixture none	· · · · · · · · · · · · · · · · · · ·
		enstant   Edit Edit	
	Cp (Specific Heat) (j/kg-k)		
		Edit	
Create/Edit Delete			
Help	Chi	nange/Create Delete Close He	lp

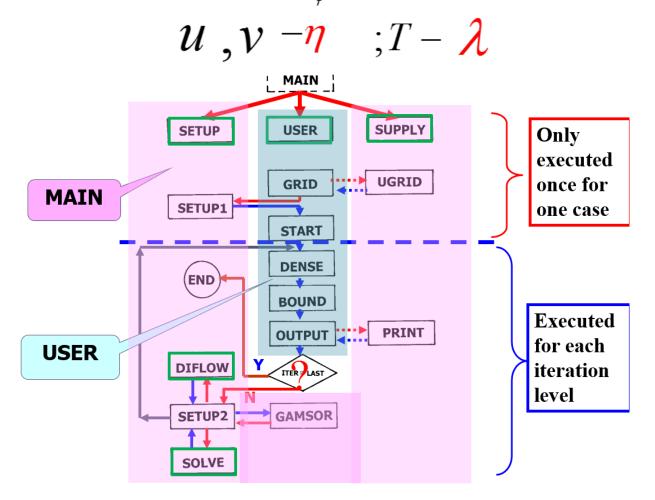




#### **Our general Code:**

## 12. GAMSOR

(1) Determine  $\Gamma_{\phi}$  for different variables:





CFD-NHT-EHT CENTER

#### **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.

fin-zone	
source-zone	
hase Type	ID
hase Type	ID -1
7	-1
hase Type Edit Parameters	



#### 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.

Zone			•	
fin-zone			Đ,	
source-zone				
💶 Solid				×
Mesh Motion	rce Terms să values să Notion Source Term		lues	
-Rotation-Axis Origin		Rotation-Axi	s Direction	
X (m) 0	constant 🔻	χ 0	constant	•
¥ (m) 0	constant 🔻	Ϋ́O	constant	•
Z (m) 0	constant 🔻	Z 1	constant	•

OK Cancel Help

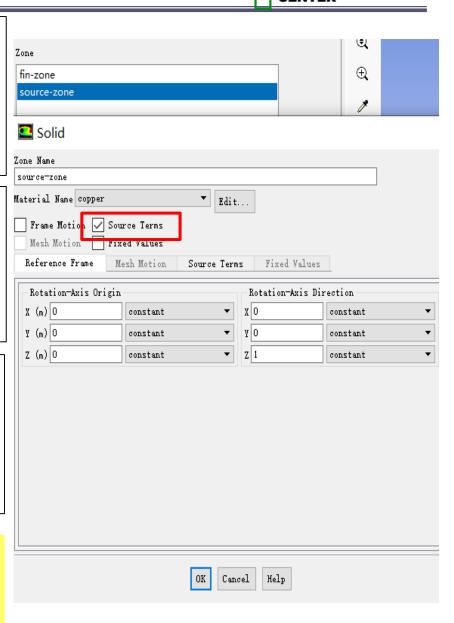


A kind of material should be given to the zone, such as **Cu** here for the two zones.

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.



CFD-NHT-EHT





For the plate 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.

 $S = S_C + S_P \phi_P, S_P \leq 0$ 

Solid	×
Zone Name source-zone	
Material Name copper 👻 Edit	
Frame Motion Source Terms Mesh Motion Fixed Values	
Reference Frame Mesh Motion Source Terms Fixed Values	
Energy 1 source Edit	
Energy sources	×
Number of Energy sources 1	•
1. (w/m3) 50000 constant	•
OK Cancel Help	
OK Cancel Help	





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.

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

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. 29/58

#### **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...

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



# For example, interface is also listed here. However, it is not a Boundary of the domain. Its type is **Interior** here.

Zone	C	1
fin-top fin-wall int_fin-zone int_source-zone interface	fin-top fin-wall int_fin-zone int_source-zone interface	
slab-bot slab-top slab-wall	slab-bot slab-top slab-wall	
Phase Type ID mixture Tinterior TO Edit Copy Profiles Parameters Display Mesh Highlight Zone	Type interior -	





## 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

Zone		Zone	~
fin-top	⊕ <b>,</b>	fin-top	€
fin-wall		fin-wall	
int_fin-zone	1	int_fin-zone	1
int_source-zone		int_source-zone interface	
interface		interface-shadow	•
		slab-bot	•
🞴 Wall	×	slab-top	
Zone Name		slab-wall	Ϋ́t
fin-top		💶 Wall	×
Adjacent Cell Zone		Zone Name	
fin-zone		finwall	
Momentum Thermal Radiation Species DPM Multiphase	VDS Wall Film Potential	Adjacent Cell Zone fin-zone	
			Multiphere INS Wall File Patential
Thermal Conditions	N 10	Momentum Internal Radi tion Species DPM	Multiphare
Heat Flux Heat Transfer Coefficient (w/m2 Temperature		Thermal Conditions Heat Flux Heat Transfer Coe	201 L ( ( 0 1) 10
Convection Free Stream Temperature	() 298.15 constant •	Transmitters	fficient (1/m2-k) 10 constant -
🔘 Radiation	Wall Thickness (m) 0	Convection	m Temperat re (k) 298.15 oonstant V
Mixed Via System Coupling Heat Generation Rate (w/mi	3) 0 constant 👻	Mixed	
via Mapped Interface	Shell Conduction 1 Layer Edit	via System Coupling	ation Rate (w/m3) 0 constant
	Edit	via Mapped Interface	Shell Conduction 1 Layer Edit
Material Name Edit		Material Name	
copper 👻 Edit		copper 👻 Edit	
OK Cancel	Help		OK Cancel Help





- **1.** Heat Flux  $(2^{nd} BC)$ :
- 2. Temperature (1<sup>st</sup> BC)
- **3.** Convection (3<sup>rd</sup> 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.

				3.036102	
fin-top		Ð,		3.69e+02	
fin-wall				3.69e+02	
int_fin-zone		1			
int_source-zone				3.69e+02	
interface		۹		3.69e+02	
slab-bot				3.69e+02	
slab-top		୍			
slab-wall		7+		3.69e+02	
💶 Wall					×
Zone Name					
slab-bot					
Adjacent Cell Zone					
source-zone					
	Radiation Species DFM Ma	ltiphase VDS	Wall Film	Potential	
	Radiation Species DPM Mo	ltiphaze UDS	Wall Film	Potential	
Momentum Thermal			Wall Film	Potential	•
Momentum Thermal Thermal Conditions Meat Flux Temperature	Radiation Speciez DFM Ma	(w/m2) 0	Wall Film	constant	
Momentum Thermal Thermal Conditions Meat Flux Temperature Convection Radiation		(w/m2) 0 Wall :		constant	
Momentum Thermal Thermal Conditions Meat Flux Temperature Convection Redistion Mixed vis System Coupling vis Mapped Interface	Heat Flux	(w/m2) 0 Wall 1 (w/m3) 0		constant constant	P
Momentum Thermal Thermal Conditions Meat Flux Temperature Convection Radiation Mixed via System Coupling	Heat Flux	(w/m2) 0 Wall 1 (w/m3) 0	Thickness (m)	constant constant	P V

Radiation Mixed		Shall Candu	ction 1 Layer	Edit.
	Heat Generation Rate (w/m3) 0	_	constant	
Convection		Wall Thickr	ess (m) 0	
Heat Flux     Temperature	Heat Flux (w/m2) 0		constant	
source-zone Momentum Thermal Radiat Thermal Conditions	ion Species DPM Multiphase	UDS Wall	Film Potential	
Adjacent Cell Zone				
slab-wall				
💽 Wall				
slab-wall		Y†	3.69e+02	
slab-top		Q	3.69e+02	
slab-bot		۹		
int_source-zone interface			3.69e+02	
int_fin-zone		× _	3.69e+02	
fin-wall			3.69e+02	
		÷.	3.69e+02	

@ 西步交通大学				
Zone fin-top fin-wall int_fin-zone int_source-zone interface slab-bot slab-top slab-top slab-wall Wall Zone Name slab-top Adjacent Cell Zone source-zone meetical Thread neticion		3.69e+02 3.69e+02 3.69e+02 3.69e+02 3.69e+02 3.69e+02 3.69e+02	×	h, Tf
hermal Conditions Heat Flux Temperature Convection Radiation Mixed via System Coupling via Mapped Interface Material Name copper	 298.15 Wall Thie	constant constant kness (m) 0 constant duction 1 Layer	P Edit	<i>h</i> =10, <i>Tf</i> =298.15 K

**<u>Remark:</u>** 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**

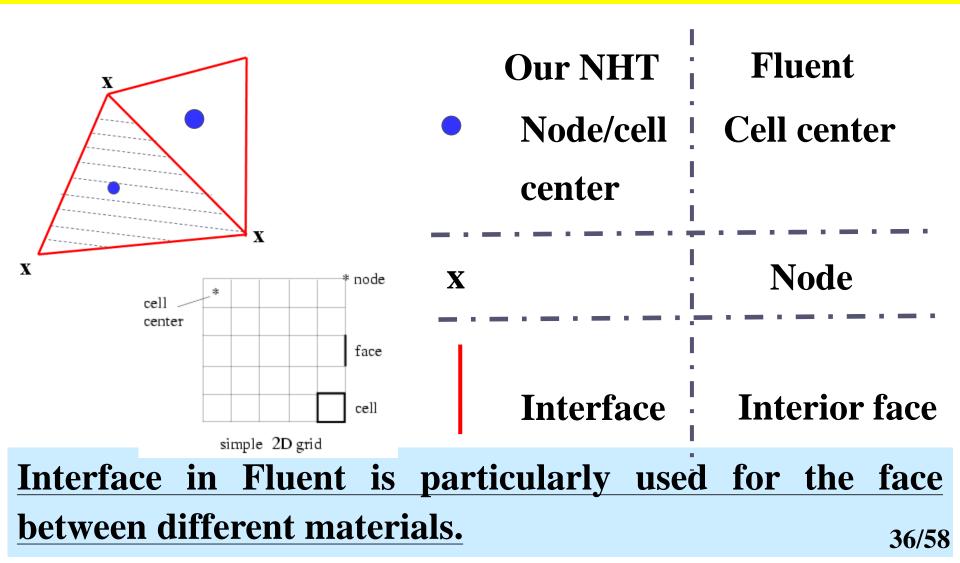
Remark:In Fluent, fortheSIMPLEseriesalgorithms, onlySIMPLEandSIMPLECareincluded.

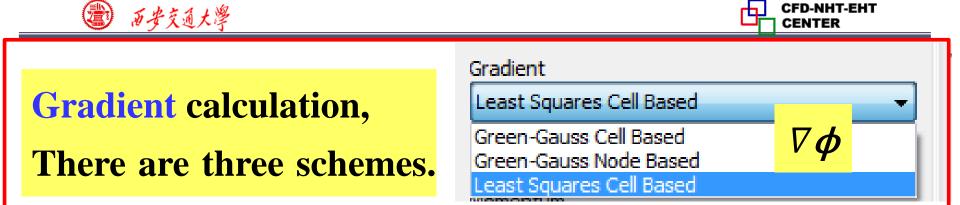
Review:What is thedifferencebetweenSIMPLE, SIMPLEC andSIMPLER?

Meshing	Solution Methods
Mesh Generation	Pressure-Velocity Coupling
Solution Setup General Models	Scheme SIMPLE
Materials Phases	Spatial Discretization
Cell Zone Conditions Boundary Conditions	Gradient Least Squares Cell Based  ▼
Mesh Interfaces Dynamic Mesh Reference Values	Pressure Second Order
Solution Solution Methods	Momentum Second Order Upwind
Solution Controls Monitors Solution Initialization Calculation Activities	Energy Second Order Upwind
Run Calculation Results	Transient Formulation
Graphics and Animations Plots Reports	Non-Iterative Time Advancement
	Pseudo Transient High Order Term Relaxation Options
	Default
	Help



## **<u>Remark:</u>** Difference between the terminology in our NHT and Fluent software about the mesh information.





- 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:

$$<\nabla\phi>=rac{1}{V_C}\int\limits_{V_C}\nabla\phi dV$$



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

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

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

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





**<b>0C1** 

39/58

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

The problem of calculating gradient is transferred into the following equation: How to determine  $\phi_f$  at the face?

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

**Calculate**  $\phi_f$  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 \$\overline{f}\$ by the average of the node values. (代数 平均值)

Nf: number of nodes on the face,  $\Phi_n$ : node value.  $\Phi_n$ , is calculated by weighted average of the cell values surrounding the nodes  $\phi_{Ci}$ .

**<u>Review:</u>** 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<sub>0</sub> and C<sub>i</sub>, 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  $\phi$ !



#### 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  $\Phi_{Ci}$  with a weighting factor  $w_i$ 

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



CFD-NHT-EHT

## Therefore, to calculate the gradient $\nabla \phi$ it to find the one leading to the minimum $\xi$ !

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

#### 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.**

西安交通大學

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	Solution Methods  Pressure-Velocity Coupling  Scheme SIMPLE  Spatial Discretization  Gradient  Least Squares Cell Dased  Pressure Second Order  Momentum Second Order Upwind Energy Second Order Upwind	
Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Transient Formulation  Transient Formulation  Non-Iterative Time Advancement  Frozen Flux Formulation  Pseudo Transient  High Order Term Relaxation  Options  Default  Help	Pressure Second Order Second Order Standard PRESTO! Linear Body Force Weighted



#### 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.  $P_{co} = P_{co}$ 

$$P_{f} = \frac{\frac{P_{c_{0}}}{a_{p,c_{0}}} + \frac{P_{c_{1}}}{a_{p,c_{1}}}}{\frac{1}{a_{p,c_{0}}} + \frac{1}{a_{p,c_{1}}}}$$





46/58

#### **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!

Momentum Second Order Upwind First Order Upwind Second Order Upwind Power Law QUICK Third-Order MUSCL	Meshing Mesh Generation Solution Setup General Models Materials Phases Cell Zone Conditions Boundary Conditions Mesh Interfaces Dynamic Mesh Reference Values Solution Solution Solution Methods Solution Controls	Solution Methods Pressure-Velocity Coupling Scheme SIMPLE  Spatial Discretization Gradient Least Squares Cell Based Pressure Second Order Momentum Second Order Upwind Energy
Energy Second Order Upwind First Order Upwind Second Order Upwind Power Law QUICK Third-Order MUSCL	Monitors Solution Initialization Calculation Activities Run Calculation Results Graphics and Animations Plots Reports	Second Order Upwind  Transient Formulation  Non-Iterative Time Advancement  Frozen Flux Formulation  Pseudo Transient  High Order Term Relaxation Options  Default  Help



### **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.

Solution Controls	
Under-Relaxation Factors	
Pressure	*
0.3	
Density	
1	
Body Forces	
1	
Momentum	
0.7	
Energy	
1	
	Ŧ
Default	
Equations Limits Advanced	
Help	

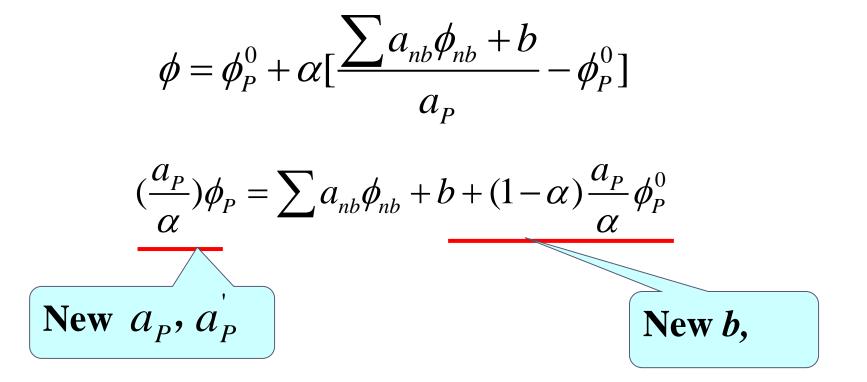
In some cases, if your simulation is not converged, and you are sure there is no problem with other setting, you can try to reduce  $\alpha$ !





CFD-NHT-EHT

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



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



### **Step 7: Solution setup: monitors**

Residual Monitors		×	< _	Surface Monitor		×	_
Options Print to Console	Equations   continuity		^	Name surf-t Options	Report Type Area-Weighted Average Custom Vectors	•	L
Plot Window 1 Curves Axes	x-velocity // // // // // // // // // // // // //	0.001	0	✓ Print to Console ✓ Plot Window 2 ♀ Curves Axes	Vectors of Custom Vectors Field Variable	f	f
Iterations to Plot	energy V Residual Values Normalize Iterations	Convergence Criterion absolute 💌	<b>~</b>	Write           File Name           E:/REM-LAOSHIKEJIAN-CASE/01/surf-mon-1.out	Temperature Static Temperature	• E • E	<b>9</b>
Iterations to Store 1000 CK	5 Scale Compute Local Scale Plot Renormalize Cancel Help		_	X Axis Iteration  Get Data Every  I  Iteration  Average Over (Iterations)  I	int_fin-zone int_source-zone interface slab-bot slab-top slab-wall		

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. 50/58





### **Step 8: Initialization**

k Page	-	
plution Initialization Initialization Methods	Run Calculation	
Hybrid Initialization Standard Initialization	Check Case	Update Dynamic Mesh
e from ones 🔻		* *
rence Frame		
) Relative to Cell Zone ) Absolute	Number of Iterations	Reporting Interval
al Values	200	1
e Pressure (pascal)		
ty (m/s)	Profile Update Interval	1
city (m/s)	1	J
bolty (m/s)	Data File Quantities	Acoustic Signals
city (m/s)	bata file qualitities	Acoustic Dignais
rature (k)		Acoustic Sources FFT
8.15		noodstro bodroes III
	Calculate	

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. 51/58





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.

Hybrid Initialization
General Settings Turbulence Settings Species Settings
Number of Iterations 10
Explicit Under-Relaxation Factor
Scalar Equation-0
Scalar Equation-1



CFD-NHT-EHT

#### Or you can simply chose Standard initialization method.

## Click Compute from and the dropdown list will show and you can select an region.

Solution Initialization
Initialization Methods
O Hybrid Initialization
Standard Initialization
Compute from
<b>•</b>
all-zones
fin-top slab-top
slab-wall
slab-bot fin-wall
fin-wall
Gauge Pressure (pascal)
0
X Velocity (m/s)
0
¥ Velocity (m/s)
0
Z Velocity (m/s)
0
Temperature (k)
298.15
Initialize Reset Patch

Solution Initialization	
initialization Methods O Hybrid Initialization O Standard Initialization	
Compute from	
Reference Frame	
Relative to Cell Zone     Absolute	
initial Values	
Gauge Pressure (pascal)	
X Velocity (m/s)	
0 Y Velocity (m/s)	
0	
Temperature (k)	_
300	
Initialize Reset Patch	
Reset DPM Sources Reset Statistics	





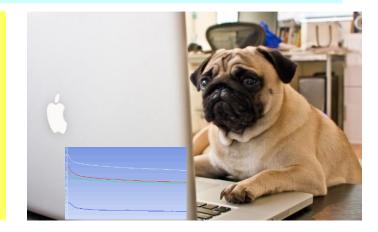
## The eight steps for preparing a Fluent simulation have been completed!

- 1. Read mesh
- 3. Choose model
- **5. define zone condition**
- 7. Solution step
- 9. Run the simulation.

- 2. scale domain
- **4.define material**
- 6. define boundary condition
- 8. Initialization
- **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.

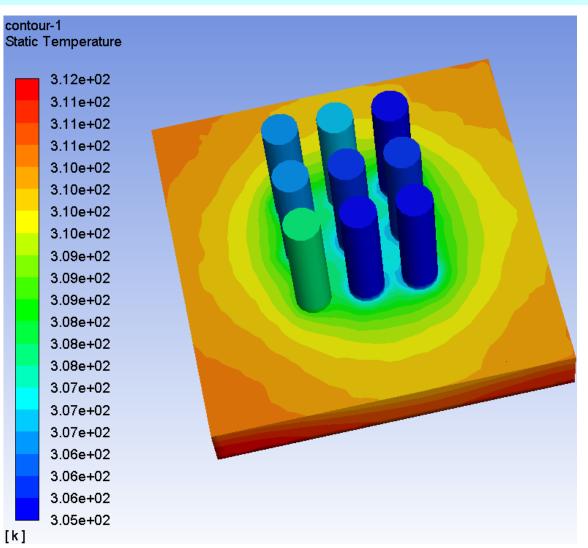


## **Diverged?** Go back to Steps 1 to 8.



#### **Step 10: Result post-processing**

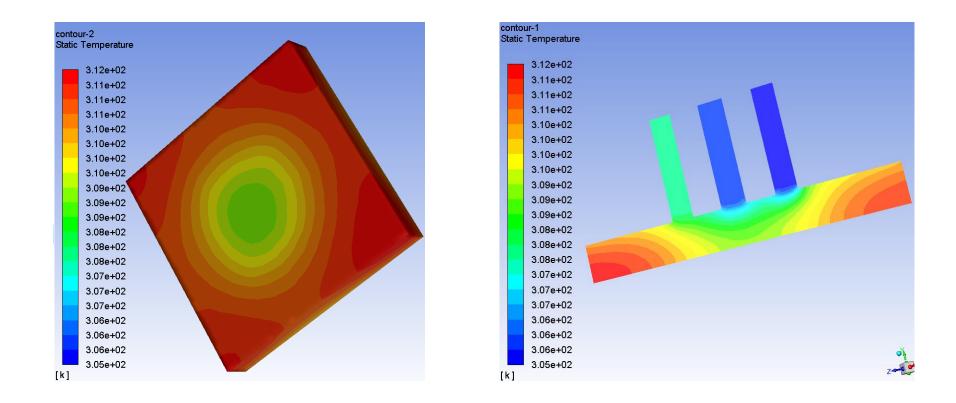
ு 雨安交通大學



#### **Fig.2 Temperature distribution**







#### **Fig.3 Temperature of bottom and fins**



#### **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)