



### **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.-28





## 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 2021年12月28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软件基础应用举例

13.1	散热装置导热问题	自我的原
13.2	非稳态圆球冷却问题	于然问题
13.3	微通道内流动换热问题	
13.4	芯片冷却流动换热问题	对流传热问题
13.5	太阳能光伏板水冷问题	
13.6	肋片强化相变材料融化	相变传热





#### **Example 4: Fluid-solid interface**



This wall type has fluid zone and solid zone on each side. This wall is called a "two-sided-wall".

When such kind wall is read into Fluent, a "shadow" (影子) zone is automatically created.





There are three options for the temperature boundary conditions of such "two-sided-wall".







If you choose "Coupled", no additional information is required. The solver will calculate heat transfer directly from the solution of adjacent cells. Such wall is not a boundary. 西安交通大學





Its shadow created by Fluent



#### **Pressure outlet boundary condition**

西安交通大學

Pressure Outlet	X
Zone Name	
out	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (pascal) 0 constant	<b>-</b>
Padefeur Direction Specification Method	
Normal to Boundary	-
Average Pressure Specification	
Target Mass Flow Rate	
OK Cancel Help	

#### Gauge Pressure (表压)



西安交通大學



For pressure outlet boundary condition, Fluent asks you to input a Backflow (回流) Total Temperature. However, it will play a role only if there is backflow. <u>There is no information provided by Fluent Help File</u> about what is the actual boundary condition for heat <u>transfer</u>.

Pressure Outlet
Zone Name
wall
Momentum Thermal Radiation Species DPM Multiphase UDS
Backflow Total Temperature (k) 300 constant
<b>Backflow Total Temperature</b>
OK Cancel Help



## The problem has been asked by many users.

Someone indicate online that the actual value of temperature is calculated using the value of last time step, or by interpolating methods from values of neighboring nodes.







**Pressure in Fluent** 

Atmospheric pressure (大气压)

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

Absolute pressure (真实压力): the true pressure

= Atmospheric pressure + Gauge pressure
Operating pressure (操作压力) : the reference
pressure (参考压力)
In our teaching code, a reference pressure point is
defined.





#### **Pressure in Fluent**

Absolute pressure (真实压力): the true pressure

= Reference Pressure + Relative Pressure

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

The same as relative pressure.

Dynamic pressure (动压): calculated by 0.5pU<sup>2</sup>

Is related to the velocity.

Total pressure (总压):

= Static pressure + dynamic pressure



#### 13.5 Flow and heat transfer in chip cooling

西安交通大學

#### 芯片冷却流动换热问题

Focus: compared with previous examples, this example is a relatively realistic problem. The domain of this Example contains fluid, board (电路 板) and chip (芯片).





#### 13.5 Flow and heat transfer in chip cooling

**Known**: Steady laminar flow and convective heat transfer around a board on top of which is a chip with source term. The domain and size is shown in Fig. 1. The boundary conditions are as follows:

■ Inlet: *u*---0.5m/s (constant)

Т---298К

- Pressure outlet: Gauge pressure (表压):0 Pa.
- Top and bottom boundary:  $3^{rd}$  boundary condition Heat transfer coefficient: h=1.5 W/(m<sup>2</sup>K); Free stream temperature:  $T_f=298$ K.





- Chip-- a constant source term, 904055 W/m<sup>3</sup>
- Front surface and back surface---symmetry



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

西安交通大學

Solution:  

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho_f} \frac{\partial p}{\partial x} + \frac{\mu_f}{\rho_f} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho_f} \frac{\partial p}{\partial y} + \frac{\mu_f}{\rho_f} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right)$$

$$u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho_f} \frac{\partial p}{\partial z} + \frac{\mu_f}{\rho_f} \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)$$

$$\frac{\partial (\rho_f C_{pf} u_f T_f)}{\partial x} + \frac{\partial (\rho_f C_{pf} v_f T_f)}{\partial y} + \frac{\partial (\rho_f C_{pf} w_f T_f)}{\partial z} = \lambda_f \left( \frac{\partial^2 T_f}{\partial x^2} + \frac{\partial^2 T_f}{\partial y^2} + \frac{\partial^2 T_f}{\partial z^2} \right)$$

$$0 = \lambda_c \left( \frac{\partial^2 T_c}{\partial x^2} + \frac{\partial^2 T_c}{\partial y^2} + \frac{\partial^2 T_c}{\partial z^2} \right) + s$$

$$0 = \lambda_b \left( \frac{\partial^2 T_b}{\partial x^2} + \frac{\partial^2 T_b}{\partial y^2} + \frac{\partial^2 T_b}{\partial z^2} \right)$$





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

Fluent Launcher	- 🗆 ×
<b>ANSYS</b>	Fluent Launcher
Dimension 2Di 3D Display options Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme Show More Options	Options Double Precision Meshing Mode Processing Options Serial Parallel
<u>D</u> K <u>D</u> efault	<u>C</u> ancel <u>H</u> elp ▼

1. Choose 3-Dimension

2. Choose display

options

**3. Choose Serial** 

processing option





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

The mesh is generated by pre-processing software such as ICEM, GAMBIT and MESHING. The document is with suffix (后缀名) "xx.msh"







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

#### Mesh→Check

Check the quality and topological information of the mesh

```
Mesh Check
Domain Extents:
    x-coordinate: min (m) = 0.000000e+00, max (m) = 1.651000e-01
    y-coordinate: min (m) = 0.000000e+00, max (m) = 2.794000e-02
    z-coordinate: min (m) = -2.540000e-07, max (m) = 1.270000e-02
Volume statistics:
    minimum volume (m3): 1.119834e-09
    maximum volume (m3): 7.845747e-09
    total volume (m3): 5.858386e-05
Face area statistics:
    minimum face area (m2): 8.370037e-07
    maximum face area (m2): 4.194085e-06
Checking mesh......
Done.
```





**2st step: Scale the domain size** 

General→Scale

**3st step: Choose the physicochemical model** 

*Re* number is calculated to determine the fluid state (laminar or turbulent)

$$\operatorname{Re}=\frac{\rho u l}{\mu}$$

The density of air is 1.29 kg/m<sup>3</sup>, the inlet velocity is 0.5 m/s, characteristic length is about 2 cm, and kinetic viscosity of air is 1.7894E-05. *Re* is 720 and thus flow is laminar.



Models	🛃 Viscous Model 🛛 🗙 🗙
Models   Multiphase - Off   Energy - Off   Viscous - Laminar   Radiation - Off   Heat Exchanger - Off   Species - Off   Discrete Phase - Off   Solidification & Melting - Off   Acoustics - C   Eulerian Wall   Energy   Vertex   Energy   Energy   Energy En	Model         Inviscid         Laminar         Spalart-Allmaras (1 eqn)         k-epsilon (2 eqn)         k-omega (2 eqn)         Transition k-kl-omega (3 eqn)         Transition SST (4 eqn)         Reynolds Stress (7 eqn)         Scale-Adaptive Simulation (SAS)         Detached Eddy Simulation (DES)         Large Eddy Simulation (LES)

@ 历安交通大学





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

If you calculate the density using the ideal gas law, the solver will compute the density according to ideal gas state equation.

Density (kg/m3)	incompressible-ideal-gas 🔹	Edit

#### **Define a new material as Chip:**

density 1000 kg/m<sup>3</sup>, Cp 500 J/(kg K) and thermal conductivity 1 W/(mK)

**Define a new material as Board:** 

density 2000 kg/m<sup>3</sup>, Cp 600 J/(kg K) and thermal conductivity 0.1 W/(mK)





#### **Step 5: Define zone condition**

# Assign different regions with the corresponding materials.

For the chip, there is a source term with value of 904055 W/m<sup>3</sup>

Solid				
Zone Name cont-solid-chip				
	Energy sources			x
Material Name chip	source	e term	Number of Energy sources 1	
Mesh Motion Fixed Values Reference Frame Mesh Motion Source Terr	1. (w/m3) 904055		constant	^
Energy 1 source Edit				





#### **Step 6: Define the boundary condition**

#### Inlet: *u* and *T* are specified.

Velocity Inlet
Zone Name
Momentum   Thermal   Radiation   Species   DPM   Multiphase   UDS
Magnitude, Normal to Boundary       Magnitude, Normal to Boundary
Zone Name
Momentum Thermal Radiation Species DPM Multiphase UDS
Temperature (k) 298 T=298 constant





#### **Step 6: Define the boundary condition**

#### **Outlet: pressure outlet, Gauge pressure as 0.**

Pressure Outlet	×
Zone Name	
outlet	
Momentum Thermal Radiation Species DPM Multiphas	e UDS
Gauge Pressure (pascal)	constant 👻
Backflow Direction Specification Method Normal to Boundary	•
Radial Equilibrium Pressure Distribution	
Average Pressure Specification	
Target Mass Flow Rate	
OK Cancel Help	Þ





#### **Step 6: Define the boundary condition**

#### Top and bottom wall: convective boundary condition

🖸 Wall		×
Zone Name		
wall-board-bottom		
Adjacent Cell Zone		
cont-solid-board		
Momentum Thermal Rac	liation Species DPM Multiphase	UDS Wall Film
Thermal Conditions		
Heat Flux	Heat Transfer Coefficient (w/n	1.5 constant ▼
<ul> <li>Temperature</li> <li>Convection</li> </ul>	Free Stream Temperatur	ure (k) 298 constant
Radiation     Mixed		Wall Thickness (in)
via System Coupling		P
Material Name	Heat Generation Rate (w	(w/m3) 0 constant
aluminum	▼ Edit	Shell Conduction Define
	OK Can	ancel Help

1 历安交通大学



#### **Step 6: Define the boundary condition**

For the front and back boundaries, keep the default set up of Symmetry.

For all the other "two-side-

walls" boundaries in the domain, keep the default set up for thermal conditions, namely "Coupled". For details of "Coupled" and "uncoupled" conditions, refer to Example 3 in Chapter 13.

Momentum T	hermal Radiation Species
Thermal Condit	ions
<ul> <li>Heat Flux</li> <li>Temperatu</li> <li>Coupled</li> </ul>	ure Heat Ge
Material Name	
aluminum	▼ Edit





#### There are many two-sided-wall in this Example.







#### **7st step: Define the solution**

For algorithm and schemes, keep it as default. For more details of this step, one can refer to Example 1 of Chapter 13.

#### Solution Methods

Scheme SIMPLE	•
patial Discretization	
Gradient	
Least Squares Cell Based	•
Pressure	
Second Order	•
Momentum	
Second Order Upwind	-
Energy	
Second Order Upwind	•
ransient Formulation	
	-
Non-Iterative Time Advancement	
Frozen Flux Formulation	

#### **Algorithm:** simple

**Gradient:** Least Square Cell Based

**Pressure:** second order

**Momentum:** second order upwind

**Energy:** second order Upwind





**7st step: Define the solution** 

For under-relaxation factor, keep it default. For more details, refer to **Example 1**.

**8st step: Initialization** 

Use the standard initialization, for more details of Hybrid initialization, refer to Example 1.

**Step 9: Run the simulation** 

**Step 10: Post-processing results** 



#### Static Temperature(K) of back boundary

(a) 百步交通大學











People in the same boat help each other to cross to the other bank, where....