



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, 2020-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 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 2020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 Heat transfer with source term

13.2 Unsteady cooling process of a steel ball

13.3 Lid-driven flow and heat transfer

13.4 Flow and heat transfer in a micro-channel

13.5 Flow and heat transfer in chip cooling

13.6 Phase change material melting with fins



第 13 章 求解流动换热问题的Fluent软件基础应用举例

13.1 有内热源的导热问题 导热问题 13.2 非稳态圆球冷却问题 13.3 顶盖驱动流动换热问题 混合对流问题 13.4 微通道内流动换热问题 13.5 芯片冷却流动换热问题 微通道问题 13.6 肋片强化相变材料融化 相变传热



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**





13.4 Flow and heat transfer in a micro-channel

微通道内流动换热问题

Focus: compared with previous examples, the focus of this example is about **pressure-out boundary** condition and 'two-side-wall" boundary condition.





13.1 Single-phase fluid flow and heat transfer in micro channel with rectangular ribs (MC-RR)

Known: Cold water at $T_f=20^{\circ}$ C flows into the inlet of a MC-RR with velocity u=0.1m/s. The side wall of MC-RR is heated with a uniform heat flux q = 30W/cm².

Assumption: (1) steady state, (2) laminar flow, (3) incompressible fluid, (4) constant fluid properties, (5) negligible radioactive and natural convective heat transfer from the micro channel heat sink.





Fig.1 Computational domain

Table .1 Geometrical parameters of MC-RR

Geometrical Parameters	W	L _{RR}	$W_{\mathbf{R}}$	L_{R}	L_{t}
Value/mm	0.5	0.7	0.1	0.3	3

1 西安交通大學

CFD-NHT-EHT CENTER

Find: Temperature distribution and pressure distribution in the domain.

Governing equations:

Continuity equation:

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

Momentum equations:

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

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



CFD-NHT-EHT

Energy equation:

$$\frac{\partial(\rho_f c_{pf} u_f T_f)}{\partial x} + \frac{\partial(\rho_f c_{pf} v_f T_f)}{\partial y} = k_f \left(\frac{\partial^2 T_f}{\partial x^2} + \frac{\partial^2 T_f}{\partial y^2}\right)$$

where T_f is the coolant's temperature, c_{Pf} is fluid specific heat and k_f is fluid thermal conductivity. Energy equation for the solid region:

$$0 = k_s \left(\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} \right)$$

where T_s is solid temperature and k_s is solid thermal conductivity



Boundary condition:

1.	channel inlet $x=0$
	$u = u_f$
	For fluid $T_f = T_{in} = 293.15 \text{K}$
2.	Channel outlet $x = 3$ mm
	$P_f = P_{out} = 1$ atm
	For fluid $-k_f(\frac{\partial T_f}{\partial x}) = 0$
3.	fluid/solid surface
	u = v = 0
	$-k_s(\frac{\partial T_s}{\partial n}) = -k_f(\frac{\partial T_f}{\partial n})$
	Where (n) is the coordinate
	normal to the wall
4.	At side wall
	$-k_s(\frac{\partial T_s}{\partial y}) = q = 30 \text{W/cm}^2$

11/36

Start the Fluent software

Fluent Launcher	-		×	
ANSYS	Fluent	Launc	her	
Dimension	Options □ Double Precision			1. Choose 2-Dimension
Display Options	Processing Sptions			2. Choose display options
Display Mesh After Reading Embed Graphics Windows Workbench Color Scheme	- Parallet			3. Choose Serial processing
Show More Options				option
<u>D</u> efault	<u>C</u> ancel <u>H</u> elp	·		

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.



CFD-NHT-EHT

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) "xx.msh"

			:=======:::::::::::::::::::::::::::::::	,		
·	1		 		··	

> Reading "C:\Users\lichennht\Desktop\ansys\4fluent_ Done.

14340 quadrilateral cells, zone 15, binary. 2124 quadrilateral cells, zone 16, binary. 28270 2D interior faces, zone 17, binary. 3987 2D interior faces, zone 18, binary. 44 2D velocity-inlet faces, zone 19, binary. 44 2D pressure-outlet faces, zone 20, binary. 177 2D wall faces, zone 21, binary. 321 2D wall faces, zone 22, binary. 411 2D wall faces, zone 23, binary. 24 2D symmetry faces, zone 24, binary. 321 shadow face pairs, binary. 17129 nodes, binary. 17129 node flags, binary.





Step 1: Read and check the mesh

Mesh→Check

Check the quality and topological information of the mesh

Mesh Check

```
Domain Extents:
    x-coordinate: min (m) = 2.500000e-04, max (m) = 3.250000e-03
    y-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-04
Volume statistics:
    minimum volume (m3): 9.997533e-13
    maximum volume (m3): 5.455531e-10
        total volume (m3): 1.500000e-06
Face area statistics:
    minimum face area (m2): 9.997748e-07
    maximum face area (m2): 2.495997e-05
Checking mesh......
Done.
```





Step 2: Scale the domain size

General→**Scale**

General		💶 Scale I	Mesh			X
Scale Check	Report Quality	Domain Ex Xmin (m)	tents 0.00025	Xmax (m)	0.00325	Scaling Convert Units Specify Scaling Factors Mech Was Created In
Solver Type Velocity Form Image: Pressure-Based Image: Absolute Density-Based Relative Time 2D Space Steady Planar Transient Axisymm Gravity Gravity	nulation etric etric Swirl Units	View Lengt	th Unit In		0.0005	Scaling Factors X 0.001 Y 0.001 Scale Unscale
Help				C	Close Help	

The mesh is generated in Fluent using unit of mm. Fluent import it as unit of m. Thus, "Convert units" is used.





Step 3: Choose the physicochemical model

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



Example 4 is about single-phase laminar flow and heat transfer, so Energy model should be activated.





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 \rightarrow Materials In Fluent, the default fluid is air and the default solid is Al. **Click the Create/Edit button** to add Water and Copper.





Step 5: Define zone condition

Solution Setup→Cell Zone Condition

Choose water for Fluid zone Choose copper for Solid zone

₽ Fluid	× 🖬 solid X
Zone Name	Zone Name solid Material Name copper Edit Frame Motion Source Terms Mesh Motion Fixed Values Reference Frame Mesh Motion Source Terms Fixed Values
Zone Name fluid Material Name water-liquid	Constant ~ Constant ~ Constant ~





Step 6: Define the boundary condition

Inlet

Velocity Inlet	
n	Velocity
Momentum Thermal Radiation Species DPM	Multiphase UDS
Velocity Specification Method Magnitude	e and Direction
Reference Frame Absolute	•
Velocity Magnitude (m/s)	constant
Supersonic/Initial Gauge Pressure (pascal)	constant -
X-Component of Flow Direction 1	constant -
Y-Component of Flow Direction 0	constant
OK	icel Help
ne Name	
1	Temperatu
Momentum Thermal Radiation Species	DPM Multiphase UDS
Cemperature (k) 203 15	constant -





Outlet: pressure outlet

Pressure Outlet	x
Zone Name	
out	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (pascal) 0 constant	-
Backflow Direction Specification Method Normal to Boundary	•
Average Pressure Specification	
Target Mass Flow Rate	
OK Cancel Help	

Gauge Pressure (表压)





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 $0.5\rho U^2$

Is related to the velocity.

Total pressure (动压):

= Static pressure + dynamic pressure

⑦ 西安交通大學





💶 Wall		X
Zone Name		~
fluid_wall		
Adjacent Cell Zone		
fluid		
Momentum Thermal Rad	iation Species DPM Multiphase	UDS Wall Film
Thermal Conditions		
Heat Flux	Heat Flux (w/m2	2) 300000 constant 👻
 Temperature Convection Radiation 		Wall Thickness (m)
Mixed via System Coupling	Heat Generation Rate (w/m3	3) 0 constant 🔻
Material Name copper	▼ Edit	

⑦ 西安交通大學





Zees Marrie	
Zone Name	
fluid_wall	
Adjacent Cell Zone	
fluid	
Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film	_
Thermal Conditions	
Heat Flux (w/m2) 300000 Constant	
Convection Wall Thickness (m) 0	
Mixed Heat Generation Rate (w/m3) via System Coupling	
Copper Edit	





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".

Thermal Conditions	
Heat Flux	
Temperature	
Coupled	

- Heat flux
- Temperature
- Coupled

Material Name

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.





You also can uncouple the sides of the wall and give different boundary condition for different sides of the wall.

🖸 Wall		
Zone Name fluid_solid_inter Adiacent Cell Zone		Adjacent Cell Zone
fluid Shadow Face Zone fluid_solid_inter-shadow		fluid
Momentum Therma	al Radiation Species DPM Multiphase UDS Wa	all Film
Thermal Conditions Heat Flux Temperature Coupled 	Heat Flux (w/m2)	Constant ▼ Wall Thickness (m) 0

The adjacent cell zone of this wall is fluid!





💽 Wall	
Zone Name fluid_solid_inter-shadow Adjacent Cell Zone solid Shadow Face Zone fluid_solid_inter	Adjacent Cell Zone solid
Momentum Thermal Radiation Species DPM Multiphase U This page is not applicable under current settings.	IDS Wall Film

The adjacent cell zone of this shadow wall is solid!

You can find the wall and its shadow created automatically by Fluent are adjacent to fluid and solid, respectively. So you can specify different BC for different walls.



西安交通大學

Its shadow created by Fluent 28/36





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

ressure-Velocity Coupling	
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





Residuals







Contours of static pressure (Pa)







Velocity magnitude







Temperature (K)

Temperature (K) of velocity as 0.01

(1) あ安交通大学

西安交通大學

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