Final Report: Simulation on Heat Exchanger

Zhijing Hu   Section 01   netID: zh168

- ## Abstract

During this simulation, incompressible water flow through heat exchangers is mainly studied in 3D. The steady state and transient heat transfer numerical simulation are two main focuses. Under the condition that the inlet velocities of cold water are 1m/s and 3m/s, in Fluent model, the steady-state temperature field and water flow field of the heat exchanger were calculated. The flow field and temperature field in the heat exchanger will change when the inlet velocity of cold water is increasing while keeping the inlet velocity of hot water remain unchanged. Additionally, when the inlet velocity of cold water increases, the effect of heat exchanging becomes more evident, and the temperature of water in the exchanger is relatively low. The temperature of the middle baffles is increasing much vastly during the exchanging process.

- ## Introduction

The convergence checking of calculation element sizing, time step of transient calculation and geometry are required, and reasonable element sizing and geometric model are selected based on the convergence results for the following heat exchanging simulations. The heat transfer situations inside the heat exchanger were studied by computational fluid dynamics, which uses finite volume method for discretization.

The original model structure of the heat exchanger is relatively complex, the shell of the heat exchanger and the hot water pipes are all having certain thickness, and there are four internal middle baffles inside the heat exchanger, the specific structures shown in the figures (Figure 1 & 2) below.
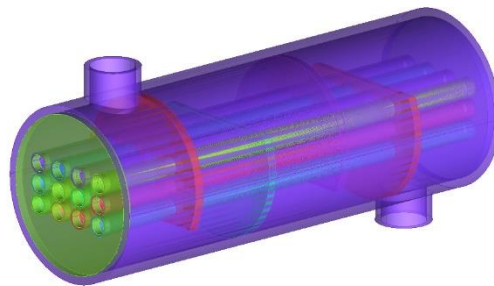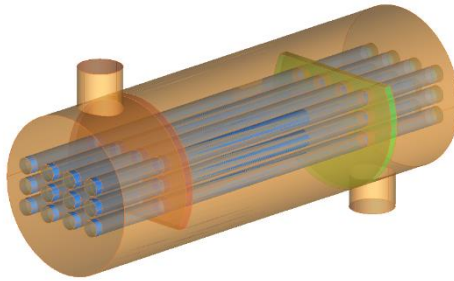

Figure 1: The Original Model of the Heat Exchanger

Figure 2: The Simplified (Used) Model of the Heat Exchanger

If the simulation is run based on the original structure, the number of element sizing of both the solid part and the fluid part would be too large to be estimated or calculated. Due to the limited resources and simulating time, (also my hardware part), the heat exchanger model is simplified to leave only two middle baffles inside the exchanger and ignore the thicknesses of the outer shell and the pipes.

While working on my project, there has two main difficulties I encountered: How to simplify the model of the exchanger reasonably, at the meantime, to ensure the element sizing is generated feasibly, as well as complete the entire simulation as detailed as possible. The final time steps are 0.02s, 0.05s and 0.1s respectively using for convergence checking, since a more smaller time step requires more time in iterations for computer.

As for several specific part of the problems, four main problems and corresponding explanations or solutions are listed below:

1. In order to check the convergence of element sizing, at least four sizing values of heat exchanger should be tested. However, with the increasing of sizing numbers, the calculation time also increases continuously. Therefore, due to the limitations, only three element sizing are selected and tested.
2. Similarly, with the element sizing selections, the value of time steps is finally decided as 0.001s, 0.005s and 0.01s using for the convergence checkpoint. Although the time steps should be as small as possible, due to the limitations, the final time step selections may relatively large compare to the ideal value, which may lead to some uncertainty in the following calculations and results.
3. Different simplified models are supposed to be selected and compared in the convergence checking of the geometry and all the calculation results should be compared, then the most reasonable geometric model should be selected according to the results. Due to the complexity of comparisons and simulations, only one simplified model plan carried for simulations and is used in this paper.
4. In steady-state computation, the boundary condition on pressure is used to calculate the outlet of cold water and hot water. Since the distance between outflow and the inflow is considerably short (Figure 3), which is about 1.08m, the corresponding boundary conditions may cause some influences on the calculation results.
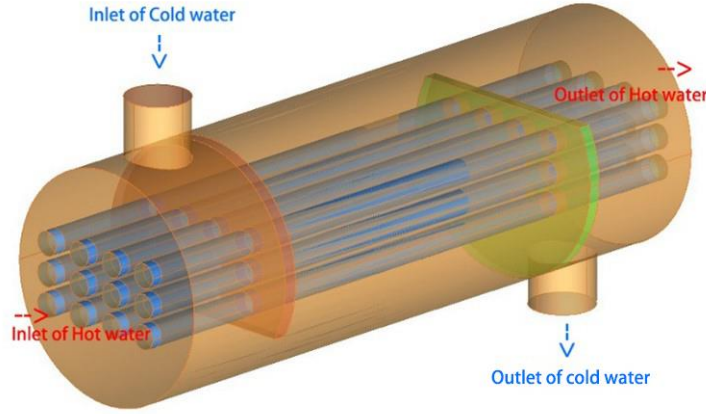
Figure 3: Inlet and outlet of hot/cold water

From my personal aspects (I later saw the revision of the outline of final paper and the aspects part is deleted, but I still write something here), and here are a few reflections: due to the limited understanding of heat transfer problem and the lack of familiarity with model simplification, it takes me a long time to finish the whole simulation project. Also, the theoretical knowledge of time step of transient heat transfer calculation is relatively insufficient, the final decision on time step is obtained by referring to relevant literature materials, which is listed in the references.

• **Methods**

During the simulations, RANS (Reynolds oil-Navier-Stokes) method is used for calculation, and are solved based on Fluent model, followed by Tecplot. The fluid equations include continuity calculations, momentum calculations and energy calculations.

Continuity

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \qquad \text{[1]}$$

Where ρ is the density of air, t is the flowing time of the fluid, and u, v, w are the velocities along 3D axes.

Momentum

$$\frac{\partial(\rho u)}{\partial t} + \text{div}(\rho u \boldsymbol{u}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + F_x \qquad \text{[2]}$$

$$\frac{\partial(\rho v)}{\partial t} + \text{div}(\rho v \boldsymbol{u}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + F_y \qquad \text{[3]}$$

$$\frac{\partial(\rho v)}{\partial t} + \mathrm{div}(\rho v\boldsymbol{u}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + F_y \qquad \textbf{[4]}$$

Where ρ is the density of air, t is the flowing time of the fluid, **u** is the velocity vector, p is the pressure. τ_{xx}, τ_{xy} and τ_{xz} are the component of the viscous stress on the surface, F_x, F_y and F_z are the corresponding forces.

Energy

$$\frac{\partial(\rho T)}{\partial t} + \mathrm{div}(\rho \boldsymbol{u} T) = \mathrm{div}[\frac{k}{c_p}\,\mathrm{grad}\,T] + S_T \qquad \textbf{[5]}$$

Where, c_p is the specific heat capacity, T is the temperature, K is the heat transfer coefficient of the fluid, and S_T is the internal heat.

In the following calculations, the thicknesses of the heat exchanger shell and the inside pipes are ignored. At the same time, in order to reduce the total number of element sizing, the two middle baffles are removed. The heat exchanger shell is set as adiabatic, without considering the heat exchanging with the external environment temperature. The metal material of the heat exchanger is stainless steel.

Assume that the heat exchanger structure is completely sealed, the water inside is set as an incompressible fluid with constant density in the following calculation process, and evaporation and condensation of water are ignored. Moreover, the heat radiation effect to the external is also ignored and only consider the internal convection and conduction of the heat.

Mesh Convergence

The steady state and the standard K-E turbulence model are set for the simulation. 1.13 million, 3.84 million and 5.55 million are set as the element sizing, respectively. Heat transfer calculation is carried out on these three numbers, getting the average temperatures of the hot water outlet as well as the cold one. The inlet boundary conditions are tested in velocities of both waters, as for outlet, are in pressure.

The inlet velocities of both waters are 1m/s, with the temperatures equal to 298K and 343K, respectively. The relative pressure at the outlet was zero.

Element Sizing	Sizing Numbers (Million)	The average outlet temperatures of cold water （K）	The average outlet temperatures of hot water （K）
1	1.13	299.81	341.83
2	3.84	299.87	341.76
3	5.55	299.90	341.72

Table 1: Testing on mesh sizing

Through the calculation, with the increase of sizing numbers, the average outlet temperature of cold water outlet is increasing, absorbing the heat increases; While the average outlet temperature of hot water is decreasing, causing by the heat transferring to the cold water. When the mesh sizing is 3.84 million, continuous increasing of the sizing number has less effect on the average temperatures, which meet the requirements of convergence. (Figure 4, 5)
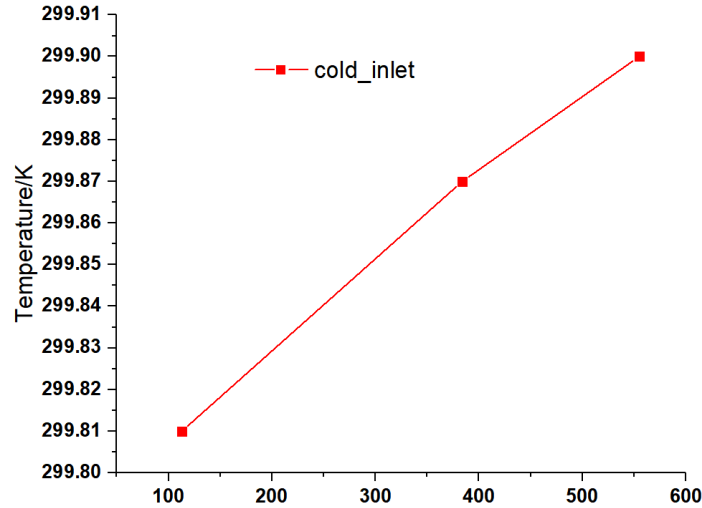
Figure 4: Average temperature of cold water vs Sizing
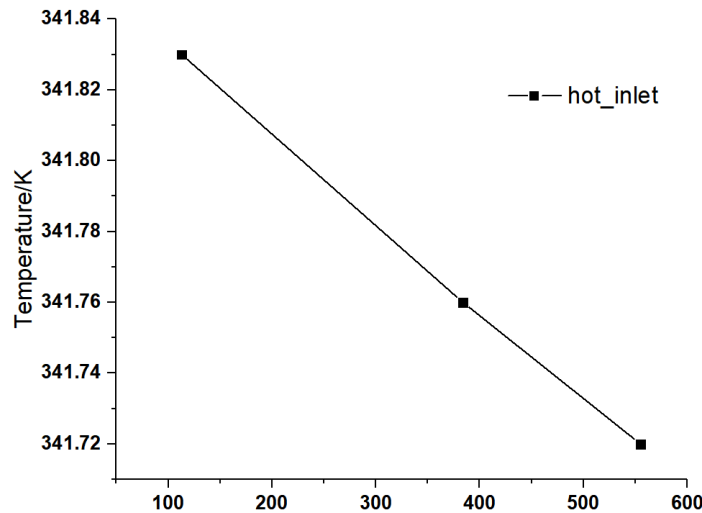
Figure 5: Average temperature of hot water vs Sizing

Timestep Convergence

The time steps used in the calculation are 0.02s, 0.05s and 0.1s, respectively, and the total calculation time is 3s. At the measuring point (-0.15,0.07 and 0.56), the velocities are taken for comparison after each iteration, and the influence of time steps on the convergence could be determined. The figures (Figure 6, 7, 8) below are showing the velocity changes at the specific measuring point in three different time steps.
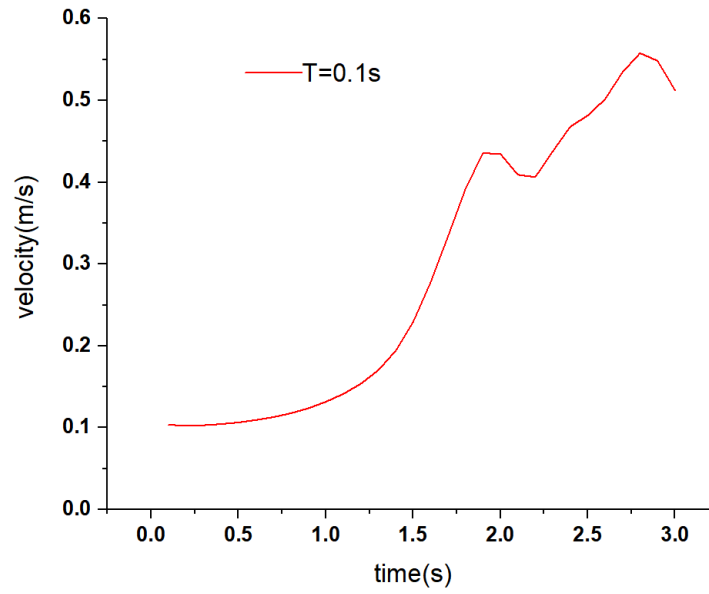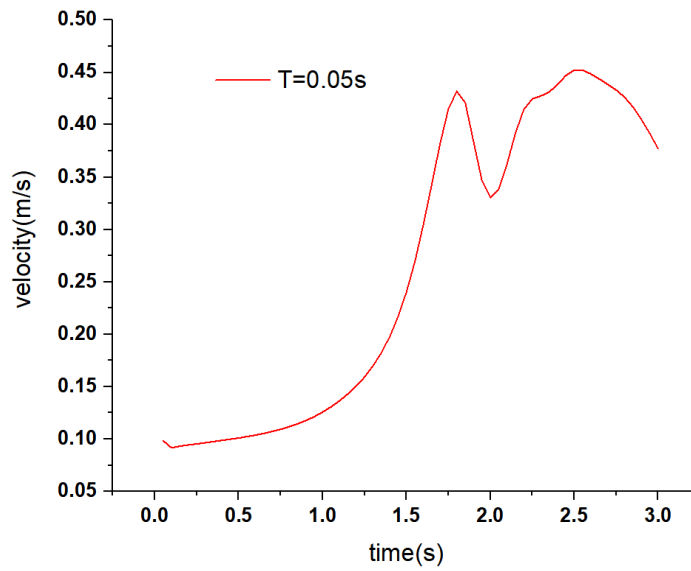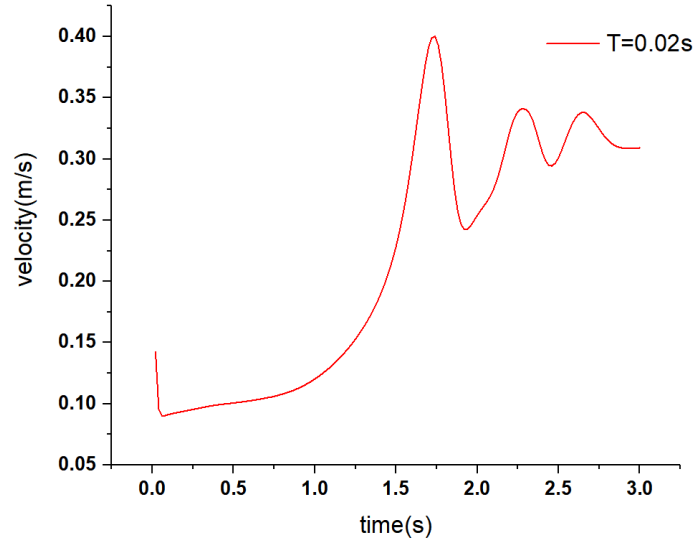
**Figure 6: Timestep = 0.1s**

Figure 7: Timestep = 0.05s

Figure 8: Timestep = 0.02s

It can be seen that the velocities at the specific measuring point are different under different time steps, which indicates that the size of time step has a significant influence on the calculation results. In order to obtain more accurate and reliable results, the time step should be as small as possible, however, considering the factors such as computing resources and computing efficiency, the time step should not be too small. Compared with the three different time steps, the curve under the time step of t = 0.02s is smoother at the end of the simulation, especially the fluctuations of velocities after 2s. Therefore, t = 0.02s could be selected for further simulation as the timestep is convergent.

Geometry Convergence

Under the settings as previously mentioned, the following figures are showing the calculated geometry convergence. The average temperature and average flow velocity of cold water at the outlet can be observed to reach a convergence state after 2000 iterations. (Figure 9, 10, 11)
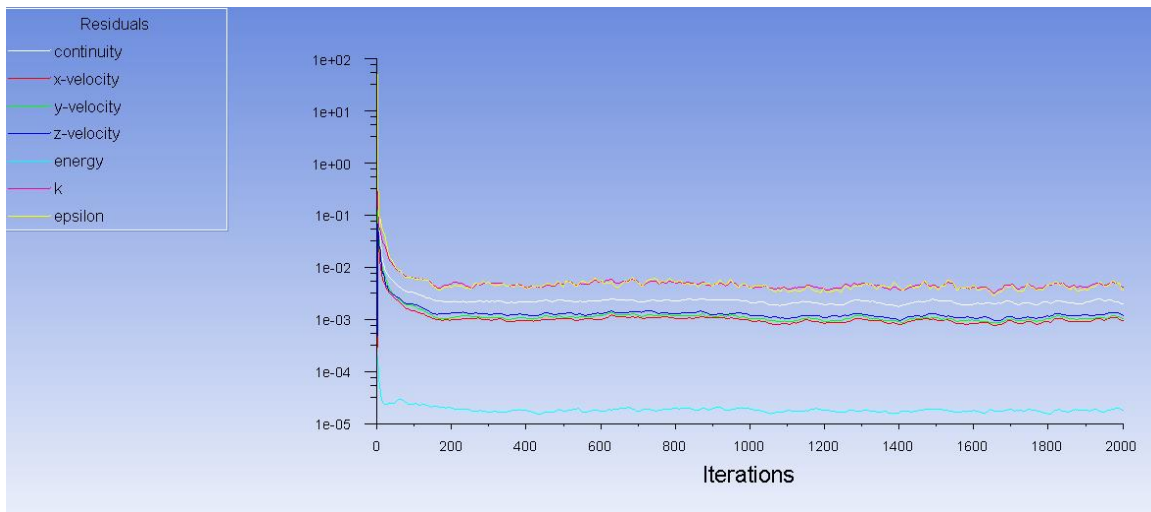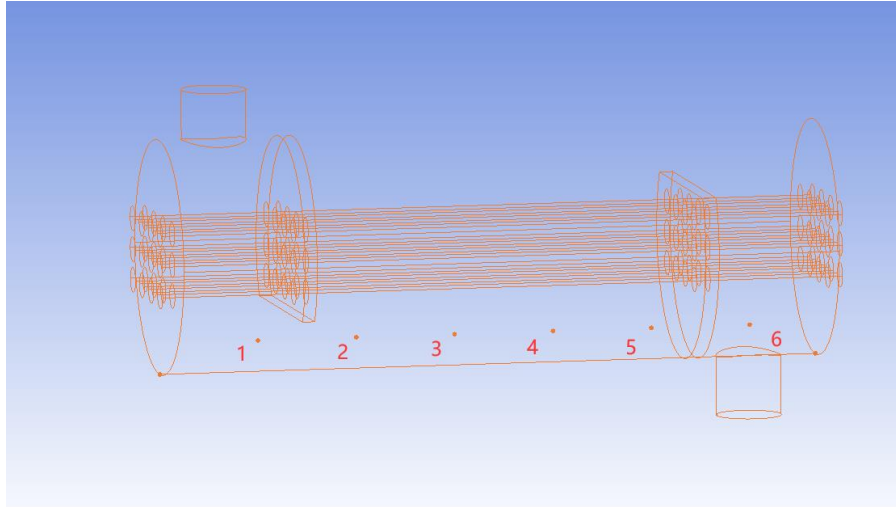
Figure 9: Total iterations
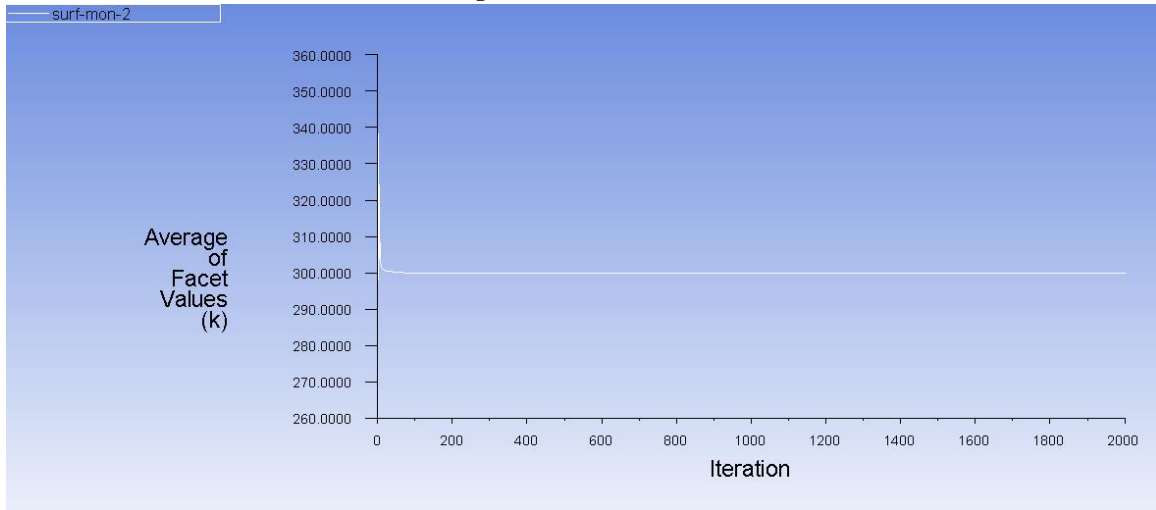
Figure 10: Iterations on Average temperature of Cold Water
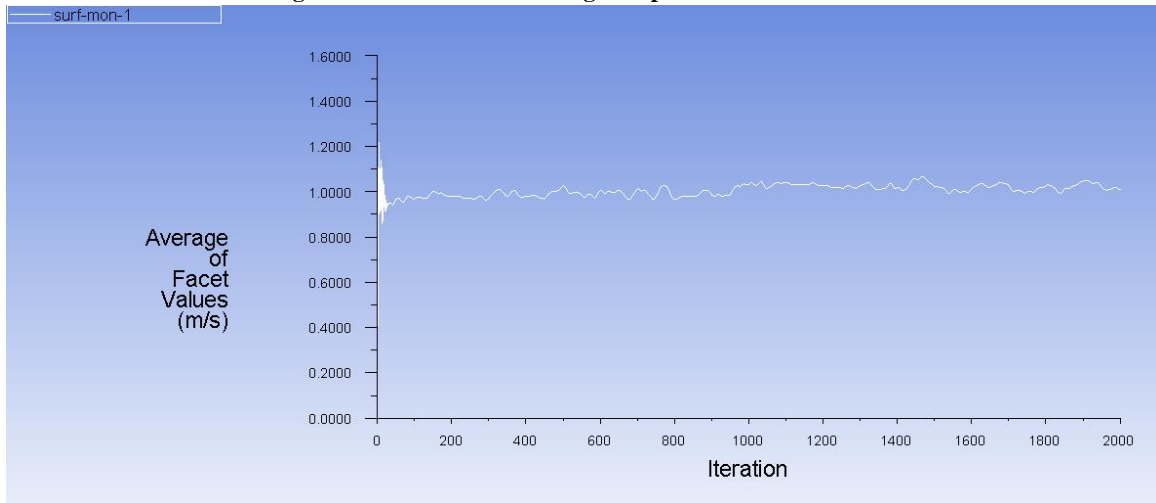
**Figure 11: Iterations of continuity**

- **Results**

Take six measuring points on the heat exchanger, (0,0.14,0.182), (0,0.14,0.344), (0,0.14,0.506), (0,0.14,0.668), (0,0.14,0.830), (0,0.14,0.992), showing in the figure. (Figure 12)

Figure 12: 6 Measuring Points

After the total simulations, the velocities of cold water at those specific measuring points could be plotted, as well as the corresponding temperature. (Figure 12, 13)
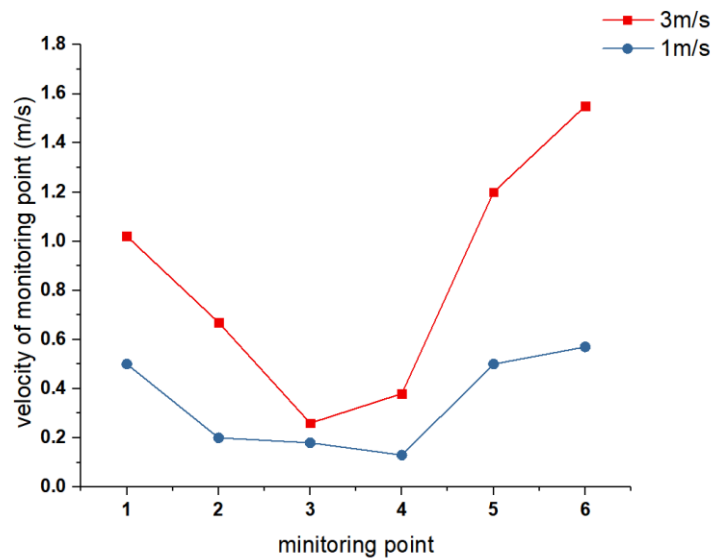
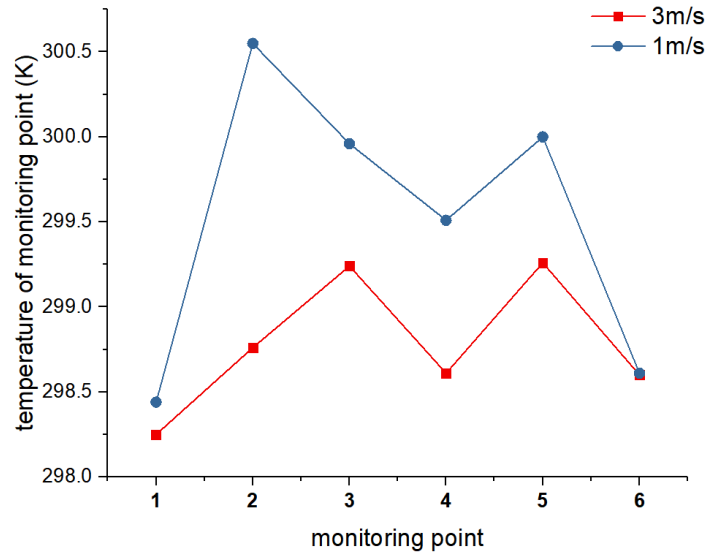**Figure 12: Velocities of the cold water at 6 different points**

Figure 13: Temperatures at 6 different points

It can be seen from the figure that the flow velocities at both ends of the heat exchanger are obviously greater than the one at the middle position, in which the flow velocity at the outlet side of cold water is relatively larger. The temperature at both ends of the heat exchanger was significantly lower than the water flow temperature at the middle position, which was mainly due to the large flow velocity at both ends of the heat exchanger and the large amount of heat taken away in unit time, resulting in the insignificant temperature rise of water flow.

With a higher the cold water inlet velocity, higher the cold water flow rate inside the heat exchanger, the more obvious the heat transfer effect would be, and eventually would lead to a lower temperature inside the heat exchanger.

The following figures are showing the velocities and heat situations of cold water under different inlet velocities, where the figures are looked from when X=0 and Y=0 (Figure 14, 15):
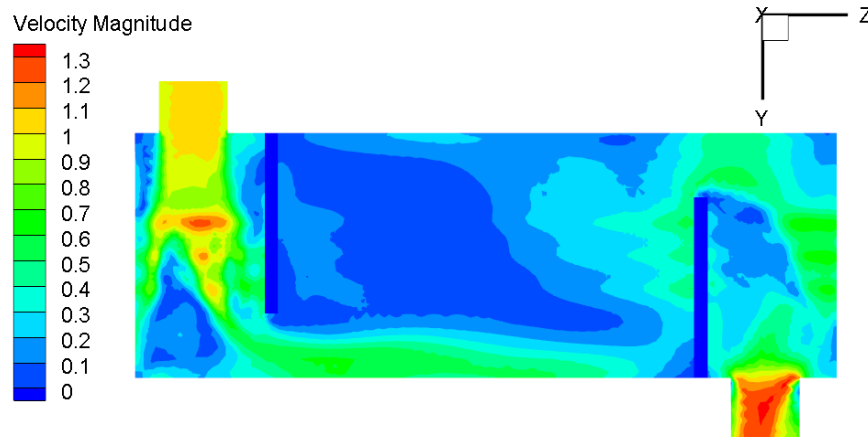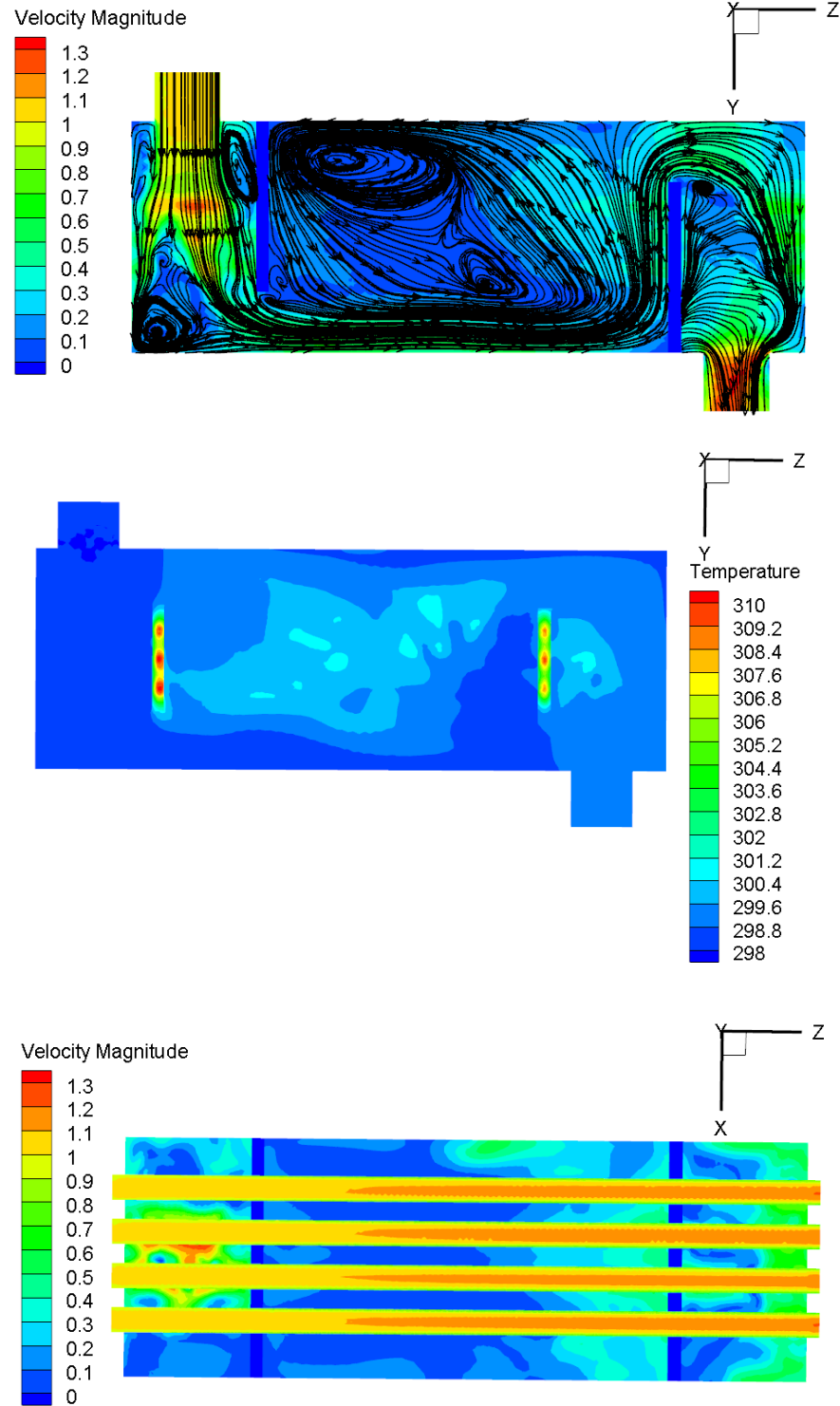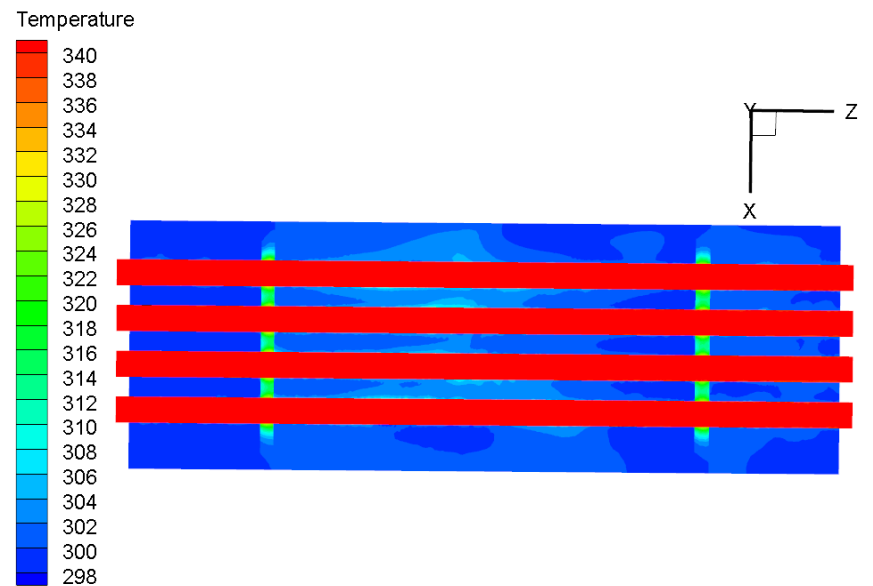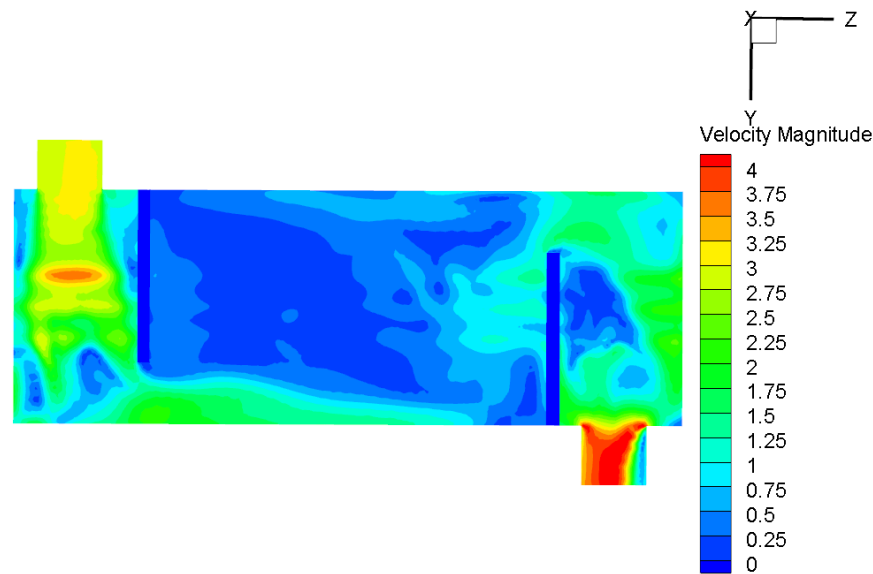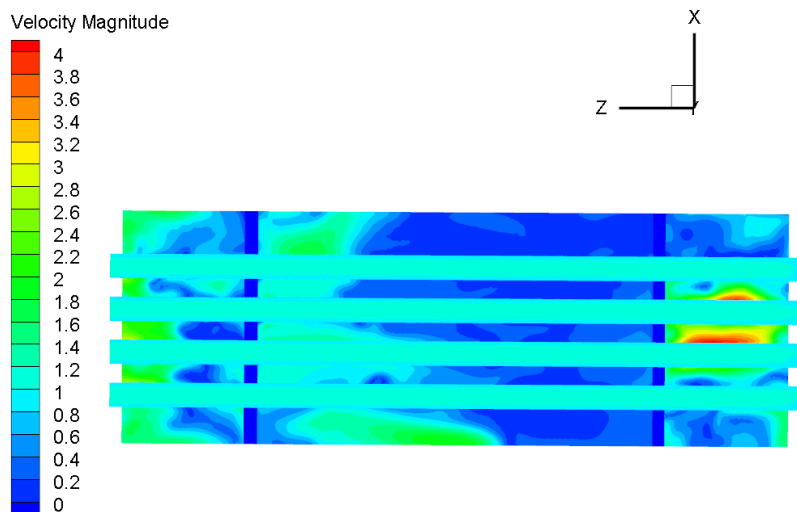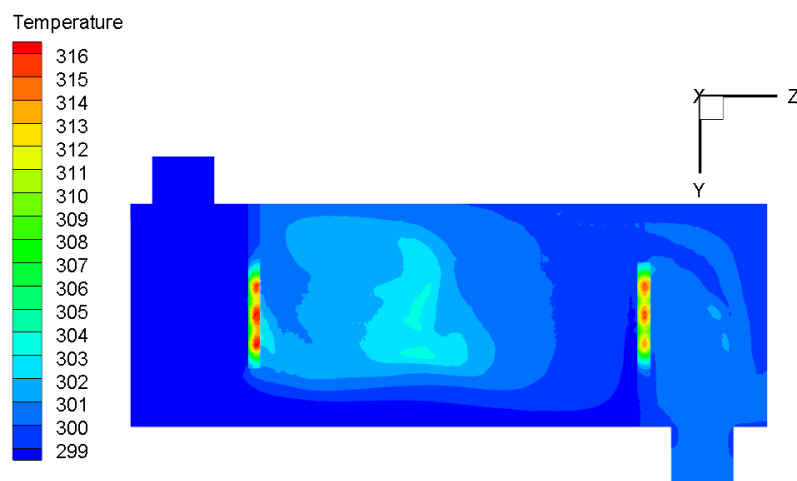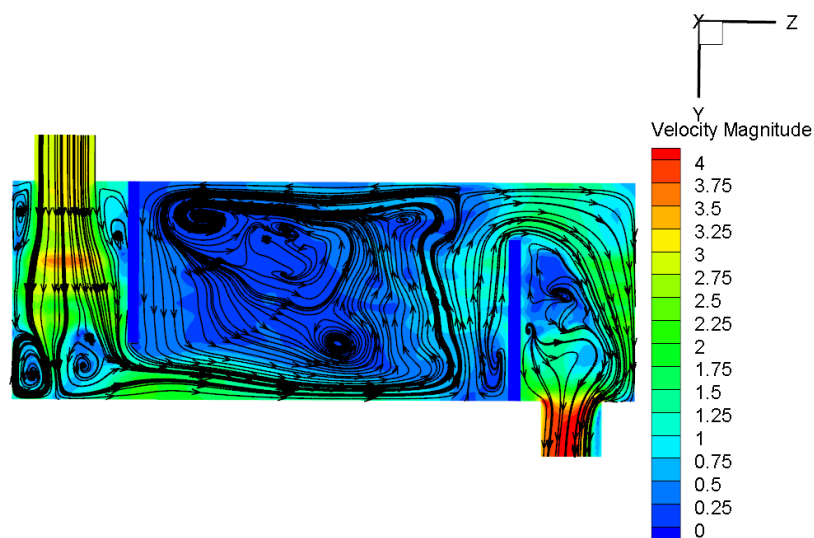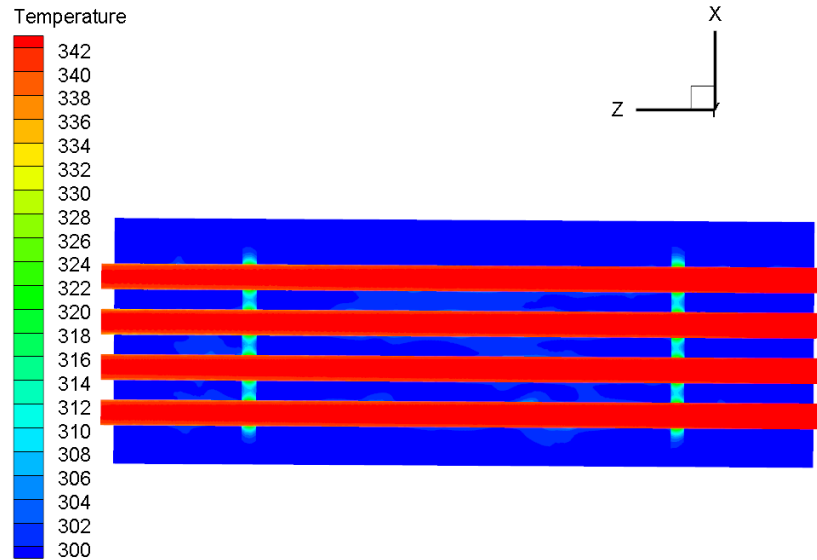
Figure 14: Inlet velocity of cold water = 1 m/s

Figure 15: Inlet velocity of cold water = 3 m/s

When the inlet velocity of hot water remains unchanged and the inlet velocity of cold water increases, the flow field and temperature field inside the heat exchanger will change. Due to the increases of cold water inlet velocity, cooling water cooling effect is increasing. According to the convective heat transfer theory, the heat taken away by cold water per unit time increases significantly, therefore, internal temperature of the pipe can be cooled.

While the internal temperature of the heat exchanger is relatively low, the temperature of baffles would increase. Since the baffle plate is made of stainless-steel metal, and its thermal conductivity is much larger than water, according to the heat conduction theory, the baffle plates can absorb a part of the heat more quickly than the water, leading to a more obviously increasing temperature.

- ## **Discussion**

Set-out problem and Answers

The project goal is to define and find out the heat exchanging states, specifically in velocities and temperature for both water after 3 seconds. According to the previously mentioned results and images, this set-out problem can be well solved.

Error and Uncertainty

A huge amount of cold water flow around the area at the exit, while the distance between the inlet and outlet is only 1.08m, which may cause certain error to the calculation results using the pressure as the outlet boundary conditions.

During the simulation, the temperature near the shell of heat exchanger is relatively low, under an ideal and simplified settings, if the thickness of heat exchanger shell and the water pipe are ignored. However, if the wall thickness is taken into consideration, the stainless-steel walls

(shell and pipes) will heat the water, and the temperature close to the shell and pipes will be relatively high in the reality.

Additionally, it is not recommended to simplify the structure of the heat exchanger in the real situation. Considering the thickness and use the original structure to simulate will make the calculation results more accurate.

Effects of simplification

When setting the heat exchanger shell as the adiabatic boundary condition, the heat released by the heat exchanger and the heat radiation to the external environment are ignored, a higher internal temperature of the heat exchanger will achieve.

Moreover, during the calculation process, the water is treated as an incompressible fluid with constant density and constant specific heat capacity, and the influence of factors such as evaporation and condensation of water is not considered, which will make the inner temperature of the heat exchanger relatively lower than the real situation. In fact, the density and specific heat capacity of water will decrease as the temperature increase, which is inversely proportional to each other.

- **Conclusions**

Before the simulation, the number of mesh element sizing is set to 3.84 million, the transient calculation time step is set as 0.02s, and the simplified geometric model is used. All three conditions can meet the requirements of convergence.

During the simulation, as the cold water inlet flow rate increases, which directly help the convection of heat transfer increases, the heat dissipation effect inside the heat exchanger is strengthened, resulting to reach a lower temperature. As the thermal conductivity of the baffles are stronger than the water, the more obvious effect of temperature rise is resulted on them. Since the baffle plate is made of stainless-steel metal, and its thermal conductivity is much larger than water, according to the heat conduction theory, the baffle plates can absorb a part of the heat more quickly than the water, leading to a more obviously increasing temperature.

There are several improvements could made for the total simulation:

1. Use the outflow velocities of both water for the boundary conditions at the outlet in order to avoid error of calculation resulted by the measured pressure values.
2. Dig into more computing resources, take the wall thickness of heat exchanger shell and hot water pipe into the consideration, and try to use the original model instead of the simplified one to calculate and simulate, leading to more accurate results.
3. Consider the facts about water density, specific heat capacity and other physical parameters that would change with the temperature. Each physical parameter should be fitted into a function related to temperature for calculation.

- ## **References**

[1] Joel H. Ferziger, Milovan Perić, Robert L. Street. Computational Methods for Fluid Dynamics[M]. Springer, 2002.

[2] Tao Wenquan. Numerical Heat Transfer[M]. Xi'an: Xi'an Jiaotong University, 2001.

[3] Wikipedia

Final Paper Rubric		
Criteria	**Rating**	**Pts**
Abstract		3.0
Introduction		3.0
Methods		3.0
Results		5.0
Discussion		10.0
Conclusions		3.0
References		3.0
		Total Points _____/30