Numerical Simulation on Flow and Heat Transfer Performance of Air-cooler for a Natural Gas Storage Compressor Unit

Liu Biyuan[1], Zhang Feng[2], Ma Zenghui[2], Zheng Zilong[2], Feng Jianmei[1]

[1] Xi'an Jiaotong University, People's Republic of China

[2] No1. Gas Production Plant of Xin Jiang Oilfield Company

Corresponding author: jmfeng@mail.xjtu.edu.cn

Abstract: Heat transfer efficiency has been a key issue for large size air coolers with the noise reducers used in natural gas storage compressor unit, especially operated in summer with cooling air at a high temperature. The 3-D numerical simulation model of the whole air cooler was established to study the flow field characteristic with different inlet and outlet structures by CFD software. The system pressure loss distributions were calculated. The relationship was obtained among heat exchange efficiency, resistance loss, and the structure of air cooler, the results presented some methods to improve cooling air flow rate and heat exchange efficiency. Based on the results, some effective measures were proposed to improve heat exchanger efficiency and were implemented in the actual operation unit.

Key words: Natural gas compressor, exhaust temperature, air cooler, heat transfer efficiency

1. Introduction

This Natural Gas storage station consists of 11 sets of compressor units and their corresponding 11 air cooler units. After the actual production, the noise value of the gas storage station exceeds the national standard [1]. Therefore, actions were taken to reduce the noise of compressor plant and air cooler unit. The exhaust temperature of natural gas compressors aims to be not higher than 65 ℃, however, the exhaust temperature of compressor working is up to 72 ℃, has exceeded the original design requirements. And during the period of gas injection, the units are going to experience the condition of high temperature in summer. With the increase of discharge pressure, the exhaust temperature of compressor will further increase and affect the stable operation of compressor. At the same

IOP Conf. Series: Materials Science and Engineering **232** (2017) 012074 doi:10.1088/1757-899X/232/1/012074

time, the working condition range of the compressor of the gas storage is very wide, the designing exhaust pressure of the compressor is 16.0~32.0MPa, and the wide load range of the working condition also affects the stable operation of the compressor and the air cooler unit. In the prior technology, the important measure for reducing the exhaust temperature of the natural gas compressor is to adopt an air cooler. The air cooler not only needs low maintenance cost, but also does not have the scaling problem compared with water cooling system, and also has a longer service life. Therefore, there is great significance to study the main factors that affect the heat transfer efficiency of the air cooler unit and to improve the structure of the air cooler for improving heat transfer efficiency.

Air cooling technology has been widely used in many compressor units. The research published by scholars Bergles [2] is the most representative. The software ANSYS CFD was used to simulate the heat transfer of air cooler and the flow field of compressor plant. Duvenhage [3] simulated 3-D flow field of air cooler, considering the horizontal wind along the long axis direction and the longitudinal direction. Staden [4] established mathematical model to study the effects of ambient crosswind on the performance of direct air cooling system. The domestic researchers of air cooler in distribution and heat transfer enhancement carried out a large number of numerical simulations and experimental researches. Liu Zhanbin [5] studied heat transfer process of fin tube numerically and experimentally. Structures of fin tubes were optimized to obtain the best heat transfer performance. Cheng Yuanda [6] studied fin tubes for direct air-cooler numerically, using porous model to replace the concept of the actual fin. Different structure parameters affected heat transfer efficiency and flow field of air cooling unit were obtained.

Now CFD technology is the main means to study performance of air cooler. Based on the establishment of a 3-D model of air cooler unit working in gas storage system, the flow field and temperature field under different conditions of air cooler were used to analyze the key factors affecting the air cooling heat exchanger efficiency [7]. Measures were proposed for improving performance of air cooler and bring down gas injection temperature lower than 65 ℃.

2. **Physical model and simulation method**

2.1. Physical Model

The physical model of an air cooling unit of the gas storage compressor was established as shown in **Figure 1**. The model was established with a ratio of 1:1 based on the air cooler units are running. In order to ensure the precision of the numerical calculation, the outlet part of air cooler mathematical model was lengthened. The air cooler unit include the first exhaust gas cooler and the second exhaust gas cooler. The finned-tube of the first stage cooler is arranged staggered with three rows and of the second stage cooler is with four rows.

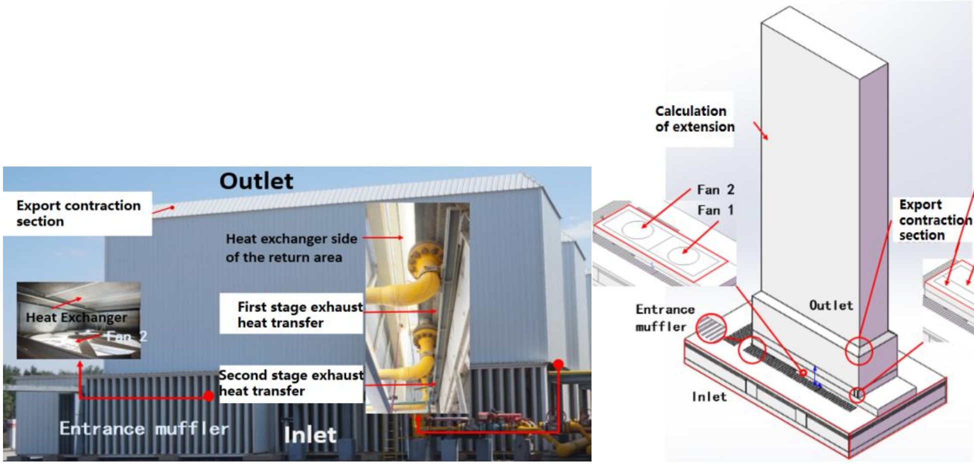


Figure 1: Air cooler unit for natural gas compressor of gas storage

In this paper, several models were established to compare with the basic model. Because of the limitation of this article, the specific model structures were not presented one by one, and the structural differences between the models were listed in **Table 1**:

Table 1: Structure of air cooler and the corresponding model number

Type	Entrance muffler	Mixed flow region	Export contraction section
Model 1（Basic model）	√	√	×
Model 2（Entrance muffler）	×	√	×
Model 3（Mixed flow region）	√	×	×
Model 4（Export contraction section）	√	√	√

2.2. Air Cooler Model Mesh Generation and Grid Independence Verification

There are big differences among each part of the air cooling unit structure size, and some regions were in irregular shape. So, tetrahedral mesh and hexahedral mesh were used to mesh the model. Hexahedral mesh was adopted in relatively regular shape part and tetrahedral mesh was adopted in area with complex shape. In the entire process the calculate condition and the accuracy are taken into account to choose an appropriate grid size, at the same time to meet a good continuity when data transferring.

In order to verify the mesh independence, the air cooler models were divided into 2070000, 3300000, and 4640000 grids, respectively. The mass flow rate of the two fans, the mass flow of the cooling air inlet and the air pressure of the cooling air upstream of the finned tube bundle (y=3755mm) were monitored. The results were shown in **Table 2**. The 3300000 meshes size model was used for simulation, as shown in **Figure 2**.

Table 2: Grid independence verification results

Grids number and the cooling air flow rate（kg/s）	Fan 1	Fan 2	Inlet
2070000	116.36	117.15	230.5
3300000	115.82	116.17	229.88
4640000	115.71	116.41	229.62

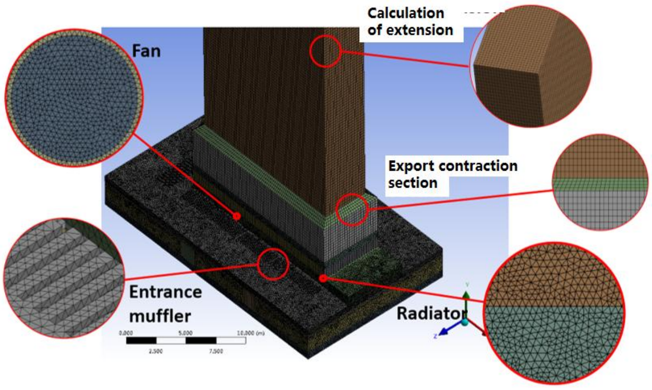


Figure 2: Air cooler model meshing

2.3. *Simulation methods*

The turbulence model has a significant influence on numerical results. Therefore, it is of great importance to select an appropriate turbulence model for computation. And there is a complex three-dimensional turbulent flow in the air cooler unit. As a turbulence model, the standard k-ε model has its own superiority in predictions of flow and heat transfer.

The air fluid has temperature-dependent physical properties. In order to obtain the heat exchange performance of the air cooler unit, energy equation was used.

2.4. *Boundary conditions*

In the calculation of the flow field and heat transfer in the air cooling unit, the boundary conditions should be set for the calculation area.

(1) Velocity entrance boundary conditions and pressure outlet boundary condition were set to simulate the resistance loss of each part of the air cooler unit.

(2) Positions of the fan and heat exchangers are treated as internal planes to analyze the resistance loss of air cooler internal flow field.

(3) The Fan model and the Radiator model are used to analyze heat transfer condition of air cooler. Meanwhile, pressure inlet boundary and pressure outlet boundary were set for the air cooler model.

The whole simulation of air cooling units is concerned with the overall heat transfer performance rather than the flow tube internal heat exchange. Therefore, the model is simplified reasonably. Finned tubes were treated as a radiator plane, in which the relationship of practical pressure loss and the heat exchanger coefficient with the velocity were arranged in the plane. The flow resistance and loss coefficient are related to the dynamic pressure of fluid. In this paper, the flow resistance loss curve of the first stage air cooler [6] is $P = 0.1624 + 12.505v + 0.9972v^2$, the flow resistance loss curve of the second stage air cooler [6] is $P = 0.17864 + 13.7555v + 1.09692v^2$. The total convective heat transfer coefficient of the heat exchanger [7] is $k = 276.23 + 39.028v - 8.5439v^2 + 0.7057v^3$.

Fan was treated as a performance curve using lumped parameter [8]. It is assumed as an infinite thin plane. The air pressure is going to increase when going through the fan. The discontinuous pressure rise is defined with the velocity of air. The fan performance curve [6] is $P = 360.875 + 11.013v - 1.84v^2 + 0.01993v^3$ in this paper.

Considering the effect of tangential velocity on the flow field, the rotating speed of the fan model was set up. The radial velocity was considered as zero, and there is a linear relationship [6] with tangential velocity and the radius $v_\theta = fr$, in which $f = 10.09$.

3. **Results and analysis**

3.1. Effects of local Structure on Flow and Heat Transfer

The influence of the local structure of air cooler units on the flow resistance loss was investigated, by comparing the cooling air flow rate of different models, The larger the cooling air flow, the smaller the resistance of the air cooler. The influence of the cooling air flow rate affects the heat transfer capacity when air goes through the fined tube with different velocity.

As results shown in Table 3, the cooling air flow rate increased 1.04% when there was no entrance mufflers. The flow rate of air decreasd 1.26% when there exists export contraction section. Therefore, the entrance muffler and the export contraction section of the air cooler would increase the resistance loss of the air cooler and reduce the flow rate of the cooling air.

Table 3: Cooling air flow rate of different models

Type	Mass flow rate kg/s			Increase
	Inlet	Fan1	Fan2	（%）
Model 1（Basic model）	229.88	115.82	117.47	-
Model 2（Entrance muffler）	232.26	116.95	116.17	1.04
Model 3（Mixed flow region）	232.29	115.96	116.33	1.05
Model 4（Export contraction section）	226.98	114.76	116.22	-1.26

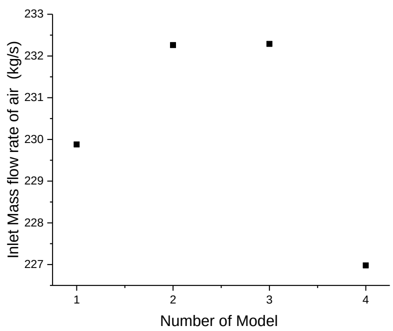
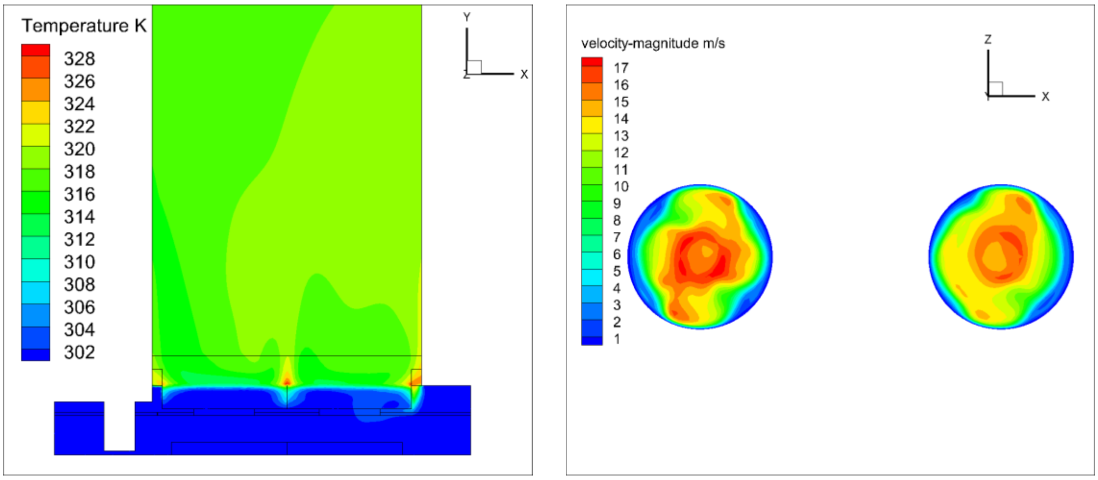


Figure 3: Inlet mass flow rate of cooling air of different models

The mixed flow region was found in the air cooling unit of the natural gas storage(as shown in Figure 1). To present the influence of mix flow region on the flow rate and temperature distribution, the temperature and velocity distribution of the surface under mixed flow region 50mm and the fan import surface were analyzed, by comparing with Model 1 (basic model) and Model 3 (mixed flow region).It can be seen in **Figure 3**4. When there existed the mixed flow region in air cooling unit, the average temperature of the surface under mixed flow region was 320.1K, which was 18.27℃ higher than the model when the mixed flow region was blocked. Fan 1 is closer to the mixed flow region, so higher temperature air flow will go through the fan and enter the air cooling system again. The inlet air temperature of fan 1 is 301.22K, which is 1.05K higher than that of no mixed flow region. At the same time, higher temperature air back into the fan 1 would decline the fan performance. The air flow rate of two fans were lower by 3.5% than that has no mixed flow region.

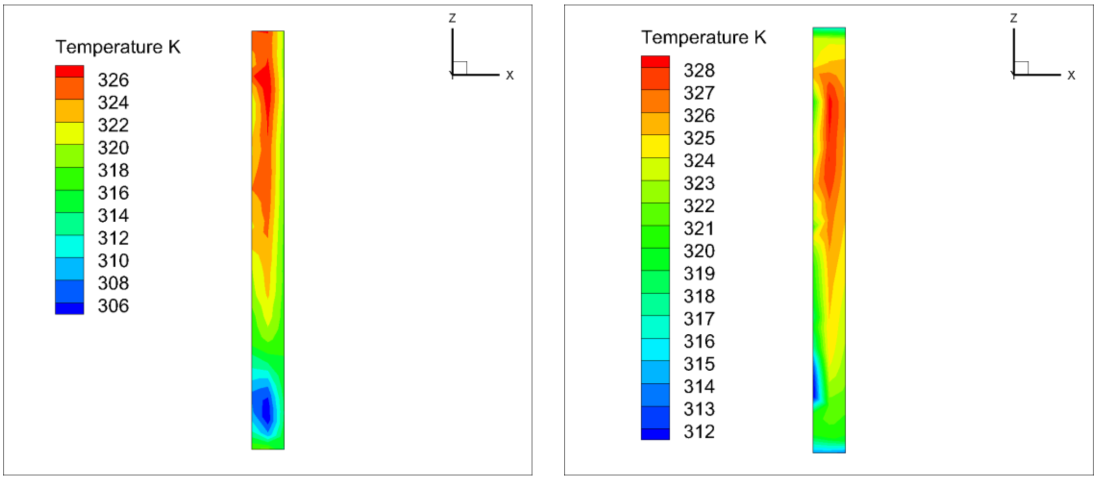
Figure 4 showed the temperature distribution on the plane z = 0mm of Model 1 (basic model) and the velocity distribution of the two fans plane.



（a）Temperature distribution of plane Z = 0mm （b）Velocity distribution of two fans

Figure 4: Model 1（Basic Model）

Figure 5 showed the temperature distribution on the plane of the mixed flow region of Model 1 (Basic model, with mixed flow region) and Model 3 (No mixed flow region) .



（a）Model 1（basic model） （b）Model 3（no mixed flow region）

Figure 5: Temperature distribution on the plane of the mixed flow region

3.2. Effects of finned tube fouling on flow and heat transfer performance

At site, it was found that there is heavy fouling on the outer surface of finned tubes in the working air cooler unit. Therefore, finned tubes with 0.1mm to 0.5mm thickness of fouling were simulated to study the effects of fouling on the flow field and heat transfer performance of air cooler unit, using the Model 1 as the studied model . The efficiency of heat exchange of Model 1 was simulated for the fouling thickness of 0.1mm, 0.2mm, 0.3mm, 0.4mm, 0.5mm,respectively.

According to the fan performance curve and the first-stage heat exchanger loss resistance curve, the performance curve was shown in Figure 6. When the fouling thickness of heat exchanger increased from 0.1mm to 0.5mm, the resistance loss increased by 4 times[9]. The results showed that the flow rate of cooling air reduced and the performance of fan decreased.

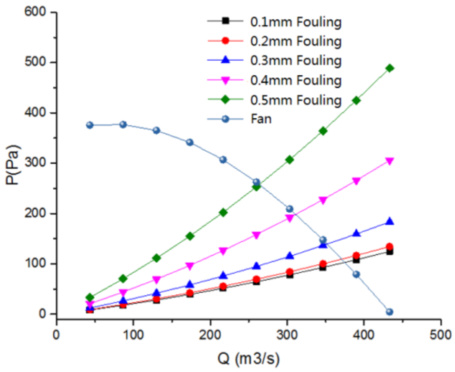
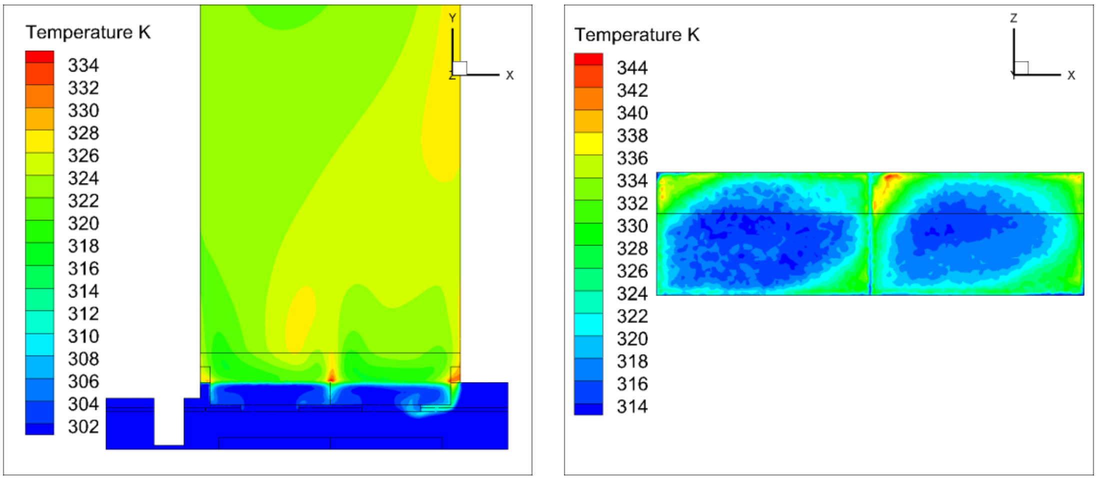


Figure 6 The performance of the fan and the heat exchanger under various fouling thickness

The fouling on the finned tube narrowed down the channel secton, which increased the cooling air velocity and made the resistance loss of air cooler unit increased. Meanwhile, the increased velocity would enhance the heat transfer coefficient. So, it's necessary to consider many factors in the finned tube outside to study how the heat transfer coefficient changed. The heat transfer coefficient with the finned tube fouling thickness increase showed a slightly increasing, but the fan performance reduced greatly. The decrease of flow rate of the cooling air weakened the heat transfer coefficient increase. The resultant effect behaved as the heat exchange capacity of finned tubes decreased.

Figure 7 showed the temperature distribution of plane z=0mm and the surface of finned tubes.



（a）The plane Z=0mm （b）The heat exchanger surface

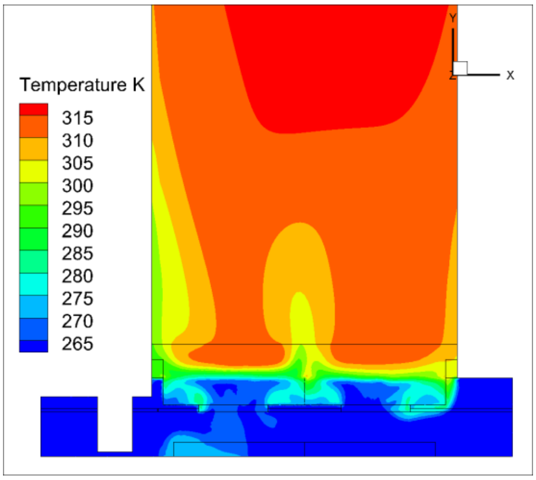
Figure 7: The temperature distribution using the model 1-0.5mm fouling

3.3. Effects of Ambient Temperature on Flow and Heat Transfer

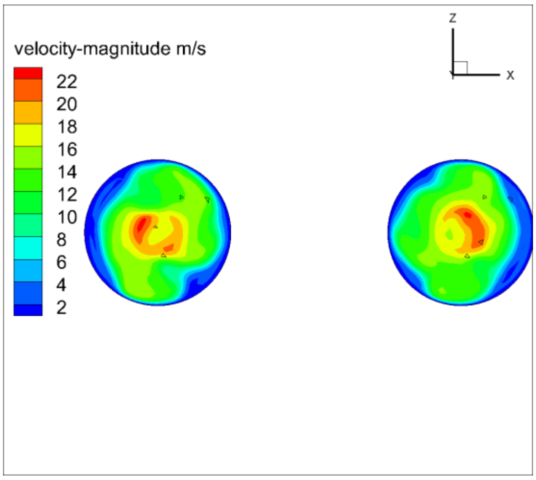
The physical properties of gas affected the resistance loss of system and fan performance. According to environment temperature range of the actual operating condition, this paper simulated the flow field and heat transfer performance of air cooling unit operating under different environment temperature from 243 K to 323 K every 10 K based on Model 1.

With the temperature of inlet cooling air increasing, the cooling air density decreased. According to the density effects of fan[10], when the density of the gas flowing through the system decreased, the fan power and fan pressure would be reduced. When the inlet temperature increased from 243 K to 323 K, the cooling air flow rate reduced by 13.47%.

Figure 8 showed the temperature distribution in the z=0mm plane and the velocity distribution in the surface of two fans. When the ambient temperature in winter is 243K, compared with air cooler working in summer at 323 K, the cooling air flow into the air cooler unit increased by 21.87%. Because of the high temperature in summer and the low temperature in winter, there is a big difference in the physical properties of the air. When the air cooler operates in high temperature, the cooling efficiency of the air cooler will be reduced. Therefore, optimizing the structure of the air cooler and improving the performance of the fan is more important to improve the heat transfer efficiency of the air cooler in the summer with high ambient temperature.



（a）Temperature distribution of plane Z = 0mm



（b）Velocity distribution of fans

Figure 8: Model 1- the inlet temperature of air is 263K

3.4. The effects of fan blades number on flow and heat transfer performance

When the number of fan blade increases, the pressure-rise of fan usually improved and the running noise reduced. This paper simulated the fan with 8 blades and 10 blades to study the flow field and heat transfer performance of the air cooling unit.

Refer to different conditions in Section 3.2, simulation of the 10-blade fan was done under different fouling thickness to study the flow and heat transfer performance of air cooling unit. The same fouling thickness was considered with numbers in Section 3.2. The simulation results are shown in **Figure 9**. When the fouling was 0.5mm, the air flow rate of 10-blade fan was 8.72% higher than that of 8 blade fan. **Figure 10** showed the velocity distribution of the fan with 8 blades and the 10 blades in the fan plane.

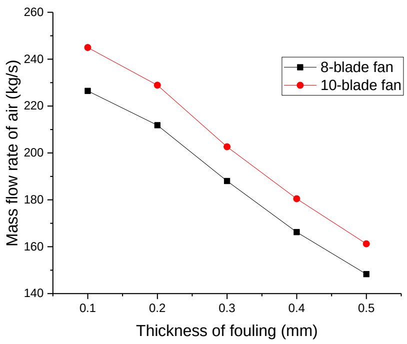
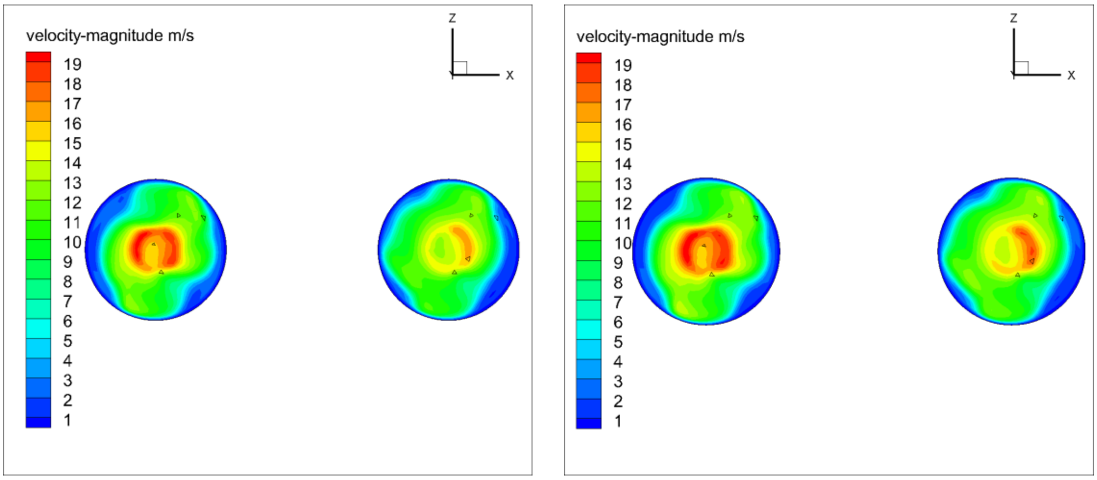


Figure 9: Flow rate of air cooler with 8 blades and 10 blades fan



（a） 8-blade fan （b） 10-blade fan

Figure 10: Velocity distribution on fan plane

The fan of 2# air cooler unit at site is the fan with 8 blades, and the 1# air cooler unit is the fan with 10 blades. The outlet temperature of 1# and 2# air cooler unit were measured at the same time. As shown in **Table 4**, the outlet temperature of the air cooler unit of the 2#unit is 5-8℃ lower than that of the 1# unit.

Table 4: Temperature of out exhaust of model with 8 blades and 10 blades (unit ℃)

Set No.	20:00	23:00	2:00	5:00	8:00	11:00	14:00	17:00
1#	70.51	69.62	68.60	66.54	64.90	67.60	67.58	67.92
2#	65.49	63.42	62.79	60.20	59.08	62.46	62.51	62.72

According to the results, when the number of blades of the cooling fan changed from 8 to 10, the flow rate of cooling air increased by more than 8% under the same condition. The main reason is that when the number of blades increased, the performance curve of the fan improved and pressure rise easily overcame the resistance loss of the air cooler.

4. Conclusions

The numerical simulation model of the whole air cooling unit was established, and the influence of local structure and environment temperature on the performance of fan and heat exchanger was simulated and compared. The main conclusions were as follows.

1) Air cooler entrance muffler has little effect on the flow properties of air cooler. The cooling air flow was only reduced by 1.02% when there are mufflers at the entrance. The air flow rate can be increased by 1.28% when the outlet contraction section was removed. The air flow rate increased by 0.73% when the mixed flow region beside heat exchange was blocked.

2) When the fouling thickness of finned tubes were increased from 0.1mm to 0.5mm, the resistance loss of finned tubes was 3.94 times than that of no fouling, which led the cooling air flow rate reduce by 27.55%, and resulted in decreasing of efficiency of heat exchange. Therefore, cleaning fouling can effectively improve the heat transfer efficiency of air cooling unit.

3) When the ambient temperature increased from 243 K to 323 K, cooling air flow rate reduced by 17.94% and the flow performance of two fans reduced by about 17%, which resulted in lower efficiency of air cooling unit. Cooling air flow rate increased by about 8% when the blade number increased from 8 to 10, which could effectively improve the heat exchange capacity of air cooler.

References:

[1] Environmental Monitoring In China. Emission standard for industrial enterprises noise at boundary[S]. 2008.

[2] E B A. Heat transfer enhancement second generation heat transfer technology[C]. Proctor. 10th UIT National Heat Transfer Conference. 1992: 3-21.

[3] Duvenhage K K D G. The influence of wind on the performance of forced draught air-cooled heat exchangers[J]. Journal of Wind Engineering & Industrial Aerodynamics,1996,62(2-3):259-277.

[4] Staden M P V P L. Numerical modelling of the effects of ambient conditions on large power station air cooled steam condensers[J]. American Society of Mechanical Engineers Fluids Engineering Division Fed,1995,39:387.

[5] Liu Zhanbin. Numerical simulation and experimental research of heat transfer process of fin tube [D]. Xi'an University of Technology,2008.

[6] Cheng Yuanda. Numerical investigation on the heat transfer and fluid flow characteristic of finned-tube heat echanger and direct air cooled condenser unit [D].Beijing Jiaotong University,2008.

[7] Bu Yongdong. Flow and heat transfer characteristics and air-side flow field optimization of air-cooledcondenser cell [D]. North China Electric Power University,2013.

[8] Ding Liyuan. Numerical simulation and analysis on Internal Flow Field of Air-cooled Unit with Deflectors [D]. North China Electric Power University,2014.

[9] Chen Wenfei. Analysis on influence characteristic of dust accumulation for air-cooled condenser[D]. North China Electric Power University,2013.

[10] Xu Kuichang. The handbook of fan[M]. China Machine Press,2011.