Journal of Southeast University (English Edition) Vol. 26, No. 2, pp. 222 – 226 June 2010 ISSN 1003—7985

Air interaction around outdoor air-cooled condensers

Wang Shugang Zhang Tengfei Zhang Jian

(School of Civil Engineering, Dalian University of Technology, Dalian 116024, China)

Abstract: In order to increase cooling or heating efficiency, a porous computational fluid dynamics (CFD) model is employed to predict the thermo-fluid status and optimize the placement of outdoor units. A full scale model is established to validate the accuracy of CFD simulation in terms of velocity and temperature distributions. The comparison between the measurement and the simulation shows a good agreement. By evaluating the condensers' sucked air temperature with CFD for three units installed in a row, it is found that the minimum separation distance among neighboring units is 0.2 m; a vertical wall should be apart from the unit line by at least 0.8 m; and large different operating pressures among units do not impact the flow rate and the heat transfer of the other units meaningfully.

Key words: air-cooled condensers; flow interaction; heat transfer optimization; computational fluid dynamics (CFD); measurement

With steadily increasing cooling demands in modern buildings, people turn to large air conditioning systems, whose outdoor air-cooled condensers are usually placed on the roof or the ground that aid to adequately dissipate heat to the ambient air. The cooling efficiency of such air conditioning systems depends much on the flow conditions through condensers. For example, if many air-cooled condensers are placed in a tiny space, the flows among the units may interact and degrade heat transfer performance; and even in the worst situation, hot exhaust air from a condenser may go directly into other units, which leads to a short-circuit of hot air. This will result in a poor performance of the air-conditioning systems and more energy consumption. Therefore, it is of great importance for outdoor air-cooled condensers in an optimal placement to enhance heat dissipation and realize high efficiency cooling.

Techniques including measurement and numerical modelling with CFD simulations can be applied to investigate the flow and heat transfer interactions among condensers. Researchers tend to choose CFD simulation due to its high efficiency, flexibility and relatively low costs. Chow and Lin[1] used CFD to predict the on-coil temperature of household air-conditioner condensers installed in building re-entrants, where heat released by the condensers induces a hot air plume that degrades heat dissipation. In addition, in 2000, Chow et al.[2] investigated different re-entrant shapes that may affect heat dissipation. Bojic et al.[3] also predicted temperature and flow conditions for four condensers per storey inside the building recessed spaces with CFD for exhaust air interaction. They found that the exhaust hot air induces a strong rising air stream, which benefits the condensers at low levels but degrades the high-level units. Moreover, in 2002, Bojic et al.[4] made a detailed study of air interaction around air-cooled condensers of window-type air-conditioners on a horizontal plane. Twenty configurations were considered and the results revealed that insufficient spaces can cause a significant increase in air suction temperature.

The above review of the literature reveals that the studies mainly focus on residential buildings and household air-conditioners; the primary tool used is CFD simulation. Few on-site measurements or laboratory experiments about the heat transfer between the condensers and ambient air have been reported; moreover, researchers are generally concerned about air interaction from condensers in vertical direction but ignore the cases of condensers on a horizontal plane. Therefore, in this paper, we focus on the air interaction among outdoor air-cooled condensers on the ground by both employing CFD simulation and laboratory experiments.

1 Mathematical Models

Air passage through an air-cooled condenser is very complicated. Generally, outside air is sucked into the condenser inside cavity by passing finned coils, which is essentially a kind of porous structure. Then the air is heated by finned coils carrying hot refrigerants inside and finally is drawn to the outside by an exhaust fan. Hence, the simulation of air movement through a condenser involves pure fluid motion and fluid motion through porous media.

The heat transfer phenomena induced by the condensers may include natural and forced convection, the flow pattern of which is governed by the conservation principles of mass, momentum, and energy. The Boussinesq approximation that is commonly used in natural convection can be applied to mimic the buoyancy forces. The two-equation RNG k-ε model is selected from among the Reynolds-averaged Navier-Stokes (RANS) equations[5]. Such a method has been the most general model for the turbulent convection flow[6].

It is difficult to know the precise details of the flow in the condenser, which is not our major consideration. So it is reasonable and possible to find a simple, finned coil alternative, for example, by treating it as a suitable porous medium. The simple homogeneous porous model[7] adds a momentum source, S_i, into the momentum conservation equation as follows:

$$S_i = -\left(\frac{\mu}{\alpha}v_i + C_2\frac{1}{2}\rho|v|v_i\right) \tag{1}$$

where S_i is the source term for the i-th (x, y, or z) momentum equation; μ is the fluid dynamic viscosity; α is the permeability; C_2 is the inertial resistance factor; ρ is the density and $|v|$ is the magnitude of the velocity. On the right hand side of Eq. (1), the first term is viscosity resistance and the sec-

Received 2009-11-23.

Biography: Wang Shugang (1963—), male, doctor, professor, sgwangln@yahoo.com.cn.

Citation: Wang Shugang, Zhang Tengfei, Zhang Jian. Air interaction around outdoor air-cooled condensers[J]. Journal of Southeast University (English Edition), 2010, 26(2): 222 – 226.

ond term is inertia resistance. This momentum source contributes to the pressure gradient by imposing a pressure drop that is proportional to the fluid velocity(or velocity squared) in the porous cell.

In order to determine inertial resistance and viscous resistance[8], permeability α and inertial resistance coefficient C_2 should be known and they can be expressed as

$$\alpha = \frac{d_b^2 \phi^3}{150(1-\phi)^2}, \quad C_2 = \frac{1.75}{\sqrt{150}} \varphi^{-3/2} \tag{2}$$

where d_b is the equivalent diameter of the solid phase, and ϕ is the porosity.

It is possible to calculate the flow and temperature fields if the viscosity resistance $1/\alpha$ and the inertia resistance $C_2/\alpha^{1/2}$ are known. Modeling the finned coil as the porous medium decreases the difficulty and accelerates the calculation speed.

2 Validation of the CFD Model

Validation of the porous CFD model becomes a necessity to test whether it can give reasonable and accurate results. To simplify test conditions and make boundary conditions controllable, we study three condenser mockups instead of realistic condensers.

2.1 Configurations of the condenser mockups

Three condensers of two types, as shown in Fig. 1, are considered with their prototypes used in an office building. The studied condenser looks like a box with finned coils on four sides and an exhaust fan at the top(see Fig. 1(a)). Inside each condenser, there are two boxes as shown in Fig. 1 (b), which are the compressor and other control components. The compressors do not run in the test but they are still kept inside to account for their impacts on flow motion. Two of them are of the same type with the dimensions of 0.877 m (length) × 0.877 m (width) × 1.370 m (height); for the middle one, the dimensions are 0.736 m (length) × 0.877 m (width) × 1.37 m(height). These units are assembled in a row and the smaller one is located in the middle with a separation distance of 0.45 m from the other two units.

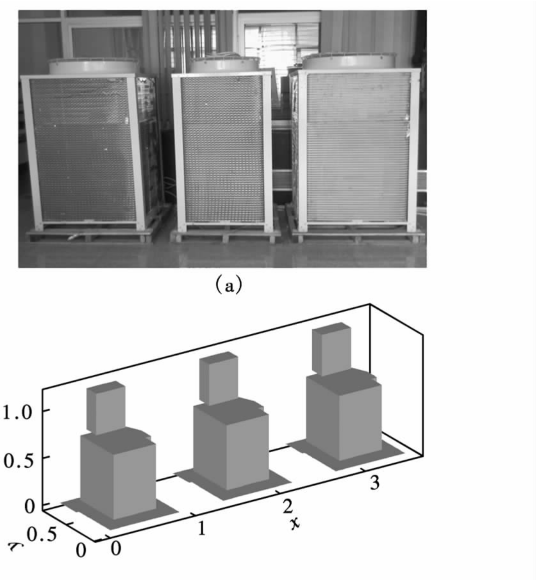

Fig. 1 The condenser mockup models. (a) The outside view; (b) Schematics of the inside configuration

Measurement variables involve flow velocity and air temperature between each two neighboring units, and the measuring instruments for velocity and temperature are an ultrasonic anemometer from KAIJO Sonic Corporation (accuracy: ±1% of readings) and many temperature recorders(accuracy: ±0.3 ℃). Fig. 2 illustrates the measuring positions, where vertical lines P_1, P_2 and P_3(in the middle of each unit with 0.20 m away from the air suction side in the y-direction) are for velocities, and other vertical lines P_{12} and P_{23}(in the middle of the interspace between the neighboring units) are for temperatures.

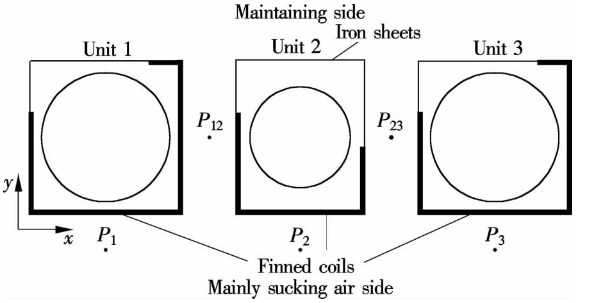

Fig. 2 Top view for positions of lines on which the measurements are conducted

The simulation ignores the wind, solar heating, and conduction heat transfer through the slab base for simplicity.

2.2 Comparison of CFD results with measurement

Fig. 3 shows the comparisons of the y-velocity and temperature profiles between CFD and measurement. Only velocity on line P_2 and temperature on lines P_{12} and P_{23} are selected for illustration due to limited space available in this paper, although we have compared results on other positions.

It can be seen in Fig. 3(a) that the y-velocity magnitude increases steadily with the vertical height due to induced air motion by the exhaust fan. The fan plays a greater role at the upper region near the fan; hence, the y-velocity magnitude increases at the higher part. Figs. 3(b) and (c) show that the temperature profiles have a trend to rise first and then to fall, and they reach a maximum at about 0.6 m from the ground. Near to the ground, cooler air enters the interspace, which causes a relatively lower temperature, and then the temperature reaches a maximum at about 0.6 m on account of the inside box shell(see Fig. 1 (b)), which results in a block effect for air circulation. Subsequently, the temperature goes down because the fan extracts a large amount of air entering the condenser in the upper region. More finned coils as heat sources in the left interspace lead to a higher temperature along line P_{12} than that along P_{23}.

Fig. 4 gives the comparison of exhaust air temperature for unit 1 on a plane 2 cm above of the fan outlet. Generally, higher temperature appears in the northwest corner of the plane, because the iron sheet instead of the finned coils is in the northwest sides(see Fig. 2) and the inside boxes(see Fig. 1(b)) hinder the entrance of ambient cool air. The simulated temperature distribution is in a more or less good agreement

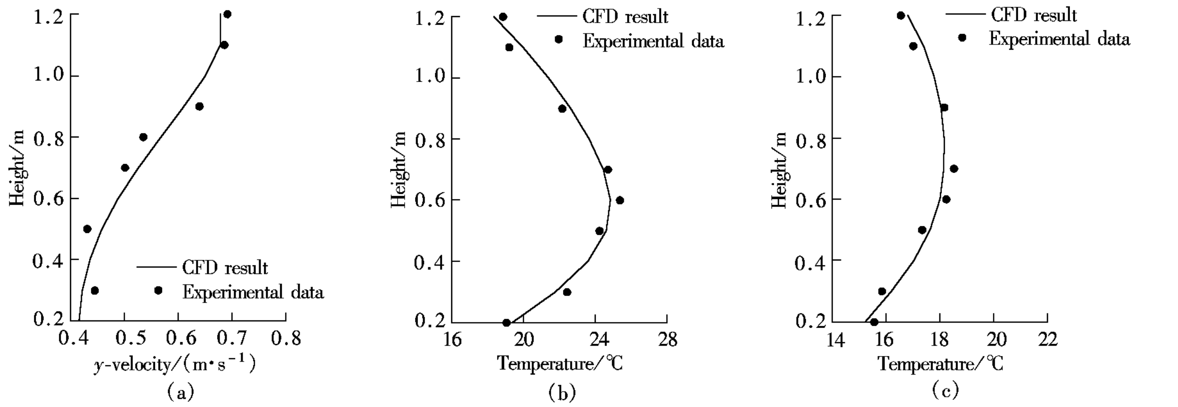

Fig. 3 Comparison between CFD results and experimental data. (a) y-velocity magnitude along P_2; (b) Temperature along P_{12}; (c) Temperature along P_{23}

with the measurement although there are discrepancies on some specific points. Also, we would like to point out that since the test runs in a little-chilled ambient air environment (average temperature of 14 ℃), the outside air can be directly supplied to a building without running the condensers to conserve energy. Nevertheless, this test is for the purpose of obtaining data to validate the CFD modeling.

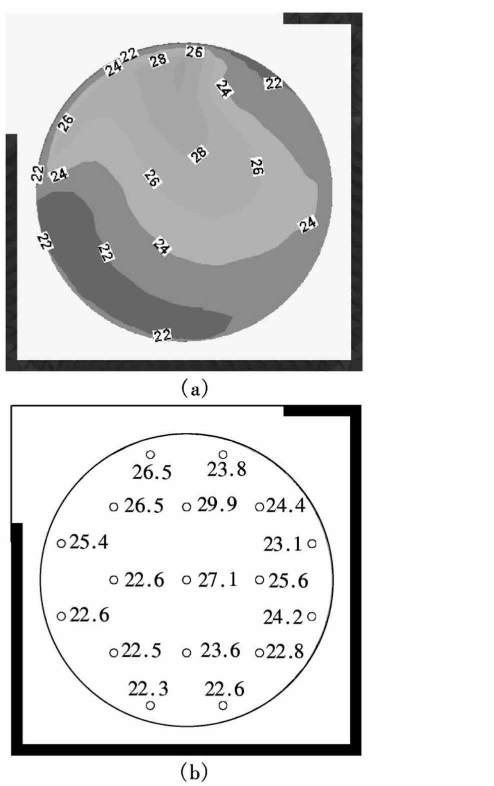

Fig. 4 Comparison of exhaust air temperature on the plane above 2 cm of the fan outlet (unit: ℃). (a) CFD simulation; (b) Measurement

Based on the above results, the tendency of the CFD results and the experimental data are nearly the same and the agreement is more or less satisfactory. Therefore, to some extent, the CFD method by modeling the finned coil as a porous medium is acceptable.

3 Simulation of Air Interaction among the Units

For simplicity, three condensers of the same type with dimensions of 0.877 m × 0.877 m × 1.370 m are selected to study air interaction. In simulation, the imposed pressure jump of the fan is assumed to be 55 Pa and the heat release rate to the ambient air is assumed to be constant at 29 kW in a calm ambient environment averaged at 28 ℃. These parameters are found to match the full load of the condenser, presuming the worst situation for condenser operation.

The distance to the maintaining side (see Fig. 2) of condensers should be greater than 0.6 m to have enough space for installation and maintenance, but other distances such as the distance between the neighboring units and the distance with a vertical wall in parallel with the array are not stipulated clearly or scientifically. The producer provides some reference distances, but he/she has never made theoretical analyses. The following will thus analyze the impacts of different separation distances. For comparison, particular attention is paid to the average sucked air temperature (t_e) to condensers and the mass flow rate passing through (m).

3.1 Different separation distances between every two neighboring units

When the separation distance between neighboring condensers is considered, there are no vertical walls around the units. With the distance decreasing gradually from 0.45 m, the sucked air temperature increases while the heat exchange rate decreases due to the air interaction among the units, in which the middle unit responds most evidently in terms of the temperature increase rate. So the air suction temperature of the middle unit is considered as the standard. The mass flow rates of the three units change little during the fixed pressure drop of the exhaust fan. From Fig. 5(a), 0.2 m can be considered as a critical distance for units placed in a small space; because if it continues reducing from 0.2 m, the curve slope becomes bigger, and the air inlet temperature rises more noticeably. The higher temperature of the sucked air, the lower COP of the air-conditioners, and hence more electricity is consumed. To guarantee a targeted COP, the units should be seperated by at least 0.2 m.

3.2 Distance between units and opposite wall in parallel with the array

When evaluating the distance between a parallel vertical wall with the condenser array, the separation distance of the neighboring units is assumed to be 0.2 m. As the distance

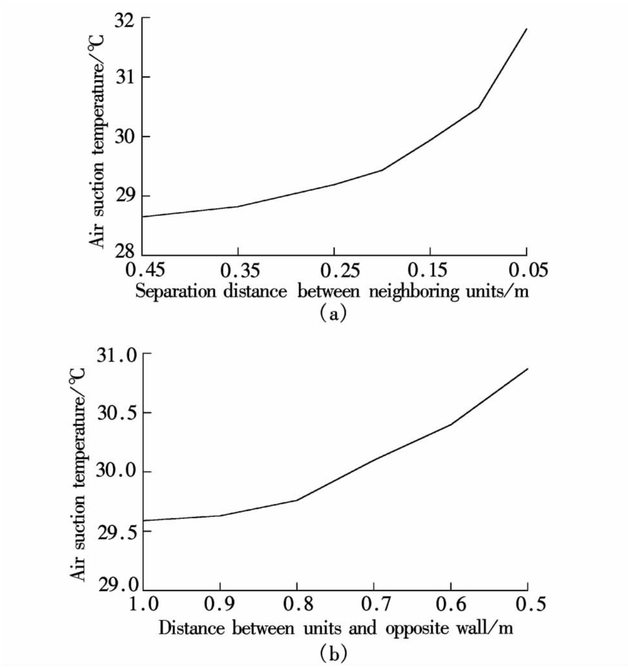

Fig. 5 Air suction temperature of the middle condenser vs. distance reduction. (a) Distance between every two neighboring condensers; (b) Distance between condensers and opposite wall

decreases from 1. 0 to 0. 5 m, air mass flow rates remain nearly stable, while sucked air temperature increases. The middle unit again responds most evidently. The results in Fig. 5(b) show that 0. 8 m can be considered as a reasonably critical distance for a small space. Because if it reduces continuously, the sucked air temperature rise of the middle unit will be faster, which results in a deteriorated performance of the system. The impact of a vertical wall is not so evident as the separation distance because the vertical wall exerts an influence on the only sucking side of each condenser.

3. 3 Fan pressure jumps

In this part, condenser exhaust fans operating at highly different pressure jumps are investigated to observe whether the highly different pressures affect air flow and heat dissipation. The separation distance between the units is 0. 2 m; the fan pressure jump of the middle unit is maintained at 55 Pa; and then the pressure jumps of the other two units increase from 55 to 1 000 Pa. The mass flow rates of the three units together with the detailed values on the four sucking faces of the middle unit(see Fig. 6) are illustrated in Tab. 1.

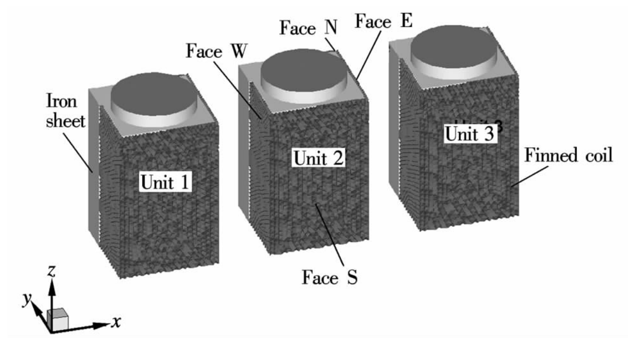

Fig. 6 A schematic illustration of the outdoor condensers

Tab. 1 Mass flow rates at different fan pressure jumps

Item	Fan pressure/Pa		Mass flow rate/(kg·s^{-1})						
	Units 1 and 3	Unit 2	Unit 1	Unit 2					Unit 3
				Total	Face N	Face E	Face S	Face W	
1	55	55	3. 03	2. 99	0. 32	0. 57	1. 44	0. 57	3. 02
2	100	55	4. 10	2. 97	0. 34	0. 47	1. 56	0. 50	4. 09
3	200	55	4. 89	2. 46	0. 32	0. 27	1. 47	0. 32	4. 88
4	400	55	8. 24	2. 85	0. 44	0. 08	2. 02	0. 20	8. 23
5	1 000	55	13. 05	2. 63	0. 55	0. 40	2. 56	0. 20	13. 03

From Tab. 1, fans whose pressure jumps are very great do not reduce the mass flow rate of the other one(the middle unit) obviously; but contrarily, the strong fan strengthens the whole flow field, which leads to the mass flow rates of some suction faces (faces N and S) increasing in a global aspect. Therefore, different fan pressure jumps should not dominate flow interruption.

4 Conclusions

Modeling the finned coils as porous media results in less labor and time in the simulation of the condensers. Although the discrepancies between CFD simulation and measurement exist, the similar tendencies of CFD and experimental results reveal the simulation to be feasible, rational and accurate. The measurement itself has also some uncertainty.

By evaluating the sucked air temperature of condensers that are assembled in a small space in a row with CFD analysis, it is found that:

1) 0. 2 m is a critical distance between neighboring condensers that does not cause significant air interaction and thus affects the cooling effects in a calm ambient environment.

2) A vertical opposite wall parallel to the units in a row that is seperated by more than 0. 8 m has a minimal effect on condenser heat dissipation.

3) One or two of the three condensers in large fan pressure jumps will not obviously affect others' sucked air flow rates or the whole heat transfer condition.

However, if in practice the above critical distances cannot be guaranteed, auxiliary measures should be taken by guiding the airflow through each condenser from being interacted with each other such as using flow separators.

References

[1] Chow T T, Lin Z. Prediction of on-coil temperature of condensers installed at tall building re-entrant [J]. *Applied Thermal Engineering*, 1999, **19**(2): 117 − 132.

[2] Chow T T, Lin Z, Wang Q W. Effect of building re-entrant shape on performance of air-cooled condensing units [J]. *Energy and Buildings*, 2000, **32**(2): 143 − 152.

[3] Bojic M, Lee M, Yik F. Flow and temperature outside a high-rise residential building due to heat rejection by its air-conditioners [J]. *Energy and Buildings*, 2001, **33**(7): 737 - 751.
[4] Bojic M, Lee M, Yik F, et al. Influence of clearances on the energy performance of window-type air-conditioners at the same level outside residential buildings [J]. *Building and Environment*, 2002, **37**(7): 713 - 726.
[5] Yakhot V, Orzag S, Thangam S, et al. Development of turbulence models for shear flows by a double expansion technique [J]. *Phys Fluids A*, 1992, **4**(7): 1510 - 1520.
[6] Zhang Z, Zhang W, Zhai Z, et al. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD. Part-2: comparison with experimental data from literature [J]. *HVAC& R Research*, 2007, **13**(6): 871 - 886.
[7] Fluent Inc. *Fluent user' s guide* [M]. Fluent Inc, 2003.
[8] Vafai K. Convective flow and heat transfer in variable porosity media [J]. *Journal of Fluid Mechanics*, 1984, **147**(1): 233 - 259.

多台室外空冷冷凝器间的气流扰动研究

王树刚　张腾飞　张　剑

（大连理工大学土木工程学院，大连 116024）

摘要：为了达到高效的制冷与制热效果，通过建立以多孔介质渗流力学为基础的计算流体动力学模型（CFD）来研究室外机与环境空气的换热过程，以获得室外机的合理布置方式；同时在实验室建立等尺度的室外机物理模型进行测试，获得了比较吻合的速度和温度分布结果，验证了 CFD 模拟方法的准确性. 然后应用数值模拟方法分析不同布置方案下室外机的进风温度，优化了实际工程中 3 台室外机成直线布置时的合理间距：相邻 2 台机组之间的最小距离须大于 0. 2 m，机组主要进风侧面与其周围高于机组的竖直墙面的极限间距为 0. 8 m；风机动力相差较大的室外机相邻布置时，其中风机压差较大的室外机不会影响其他机组的进风量和换热量.

关键词：空冷冷凝器；气流扰动；传热优化；计算流体动力学；测试

中图分类号：TU831