International Refrigeration and Air Conditioning Conference

School of Mechanical Engineering

2010

CFD Simulation and Experimental Study on Air-side Performance for MCHX

Tu Xiaoping
Gree Electric Appliances

Lin Huahe
Gree Electric Appliances

Liang Xiangfei
Gree Electric Appliances

Follow this and additional works at: http://docs.lib.purdue.edu/iracc

CFD Simulation and Experimental Study on Air-side Performance of MCHX

Xiaoping TU[1], Huahe LIN[1], Xiangfei LIANG[1*]

[1]Refrigeration Institute of Gree Electric Appliances, Inc. of Zhuhai,
Jinji West Rd., Zhuhai City, 519070, P. R. China
Phone: +86-756-8668924, Fax: +86-756-8668982
E-mail: liangxf@gree.com.cn

* Corresponding Author

ABSTRACT

The air-side heat transfer and pressure drop characteristics of micro-channel heat exchanger (MCHX) with flat-tube louver fin were investigated using both computational fluid dynamics (CFD) simulation and experiment methods. The test samples were not enough to fully investigate air-side performance, so the main aim of experimental investigation was to offer effective data to validate CFD model. The 3-dimension (3-D) CFD model was established to predict air-side flow and heat transfer characteristics. The air-side experimental data were reduced using effectiveness-NTU method. The comparison between CFD simulations and experimental data showed that established 3-D CFD model had very good prediction ability. Several important geometry configuration parameters of MCHX louver fin, involving fin pith, louver angle, louver pitch, louver length and flow depth, which have great influence on MCHX performance, have been analyzed with CFD simulation method under commonly used residential air conditioner condenser condition. The variation tendencies of air-side performance with those parameters were obtained. Based on CFD simulation results, some optional configuration parameters for flat-tube louver fin were proposed.

1. INTRODUCTION

The MCHX, also called parallel flow heat exchanger, is a kind of high performance and compact heat exchanger, which mainly consists of multi-port brazed aluminum flat-tubes and louver fins. It has been commonly used in the automotive air-conditioning for years. Recently，the MCHX has been paid more and more attention by HVAC&R (heating, ventilating, air-conditioning and refrigeration) industries and used in some commercial and residential air-conditioning system. However, the geometry configuration parameters of available MCHX and louver fin are usually designed for automotive and commercial air-conditioning and work conditions are different with that of residential air-conditioning. If the MCHX is directly used in residential air-conditioning, the high performance may not be representative. Therefore, the redesign of configuration parameters is very necessary.

Many studies on louver fin have been reported Davenport (1980), through flow visualization study with smoke traces or dye injection techniques, showed that the air flow had two directions depended on Reynolds number based on the louver pitch and maximum air velocity. The air did not pass through louvers called duct directed flow at low Reynolds numbers. However, at high Reynolds numbers, the air flow direction became nearly parallel to louvers called louver directed flows. Achaichia and Cowell (1988) further confirmed the phenomena. Based on their researches, some air-side heat transfer coefficient and pressure drop correlations for louver fin have been proposed based on a huge database (Park and Jacobi, 2009), also including some results of their own experimental studies on heat transfer characteristics of different louver fin geometries by Chang and Wang (1997), Kim and Bullard (2002). However, due to different experimental and data reduction methods, these correlations have significant discrepancy with experimental results. Recently, numerical investigations of the louvered fin array have been widely performed to reduce investigation period. Atkinson et al. (1998) and Perrotin and Clodic (2004) numerically studied the 2-D and 3-D flow and heat transfer over compact louvered heat exchangers. Hsieh and Jang (2006) carried out 3-D

numerical analysis on heat and flow for louver fin heat exchangers with successively increased or decreased louver angle patterns. However, these investigations were based on geometry configuration parameters of louver fin used in the automotive air-conditioning, and couldn't directly used for optimizing MCHX used as residential air conditioner condenser.

The major object of present study is redesign configuration parameters of louver fin used in MCHX to apply in residential air-conditioner condenser. Because of limited test samples, experimental investigation on air-side performance is only to offer effective data to validate numerical model. The CFD simulation method is mainly used to study air-side performance for MCHX with flat-tube louver fin. The 3-D CFD models, taking into account tube effects, conjugated heat transfer and conduction through the fin, are simulated to investigate the effects of several important geometry parameters on air-side performance.

2. EXPERIMENT

2.1 Test Facility and Samples
The experiments were carried out in Air-conditioning and Heat Exchangers Laboratory directly under our incorporation, consisting of an unattached Air, water and R22 refrigerant Enthalpy Method Calorimeter. And the air enthalpy difference system contained a suction type wind tunnel and a set of air flow rate measurement device. The lab was equipped with automatic control devices and high precision metrical instruments, which was designed and constructed by two professional institutes.

The geometrical configuration and terminology of the test MCHX samples are expressed as fig.1. Six MCHX samples are used for the test. The louver fin of all test samples has the same fin thickness, fin length, louver length, louver angle and number of louvers, which are 0.1, 8, 6mm, 25 degree and 12, respectively. Every MCHX has fifteen flat-tubes and one pass. The other parameters of the test samples are given in table 1.
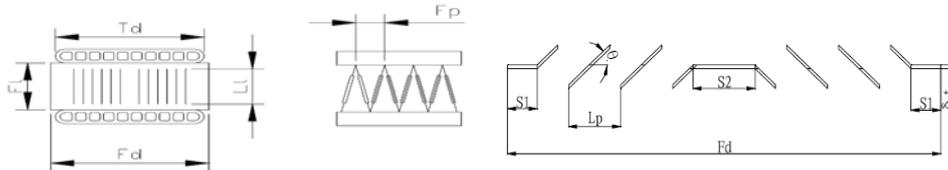

Fig. 1: Geometrical parameters of MCHX and louver fin

Table 1: The geometric parameters of the MCHX

No.	Tube length (mm)	Tube depth (mm)	Tube height (mm)	Port number	Fin pitch (mm)	Louver pitch (mm)
1	600	16	2	10	1.2	1
2	600	16	2	10	1.4	1
3	600	18	2	10	1.2	1
4	600	20	2	10	1.2	1.1
5	400	16	1.8	18	1.2	1
6	400	16	1.8	16	1.2	1

2.2 Experimental Results
The MCHX is installed in the inlet of wind tunnel with surrounded insulation to protect it from heat loss and air leakage. Tests are performed for face air velocity from 0.7 to 4 m/s, corresponding to the normal operating range of conditions in residential and commercial air-conditionings. The air inlet temperature is 10℃ for all tests. On the tube side, distilled water is used with constant 50℃ inlet temperature. The water flow rate is maintained at 1700L/h. The air-side thermal performance data are determined using the effectiveness-NTU method for cross flow heat exchangers with both fluids unmixed and the detailed reduction method can refer to Kim and Bullard's study paper (2002). Air-side pressure drop can be directly measured with static pressure device.

The fig. 2 shows the effects of fin pitch, tube depth or flow depth and tube height and face air velocity on air-side heat transfer and pressure drop performance.
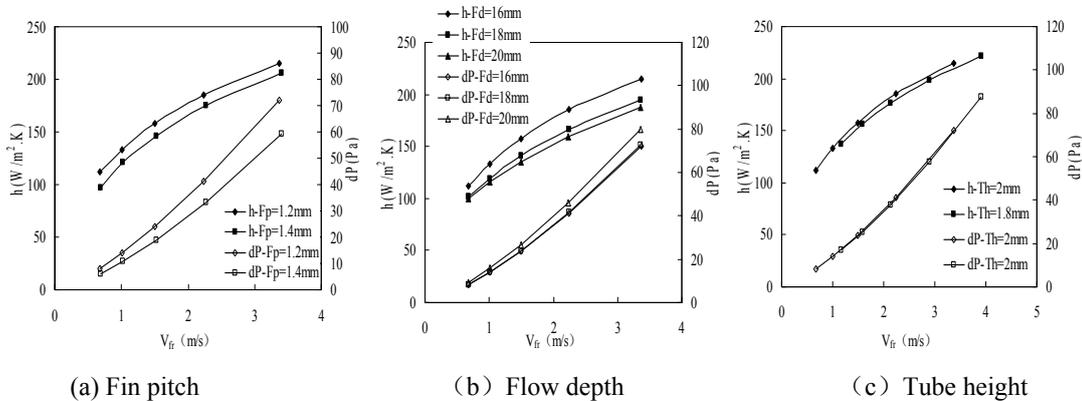

(a) Fin pitch （b）Flow depth （c）Tube height

Fig. 2: Variations of heat transfer coefficient and pressure drop with fin pitch, flow depth, tube height and face air velocity.

As expected, from the fig.2, it can be seen that air-side heat transfer coefficient and pressure drop increase with face air velocity, while the variation amplitude of heat transfer coefficient decreases and pressure drop increases. So, the face air velocity should not be too large in order to keep high integrated performance. For the same air velocity, the heat transfer coefficient decreases with increasing fin pitch and flow depth, and increases with increasing tube height, while the pressure drop decreases with increasing fin pitch and increases with increasing flow depth and tube height. However, the variations of air-side pressure drop with flow depth and tube height are not obvious. In addition, the effects of fin pitch on air-side performance are not always developing to a same direction. When considering the varied louver angle, there is a turning point. That would be discussed in following CFD simulation section.

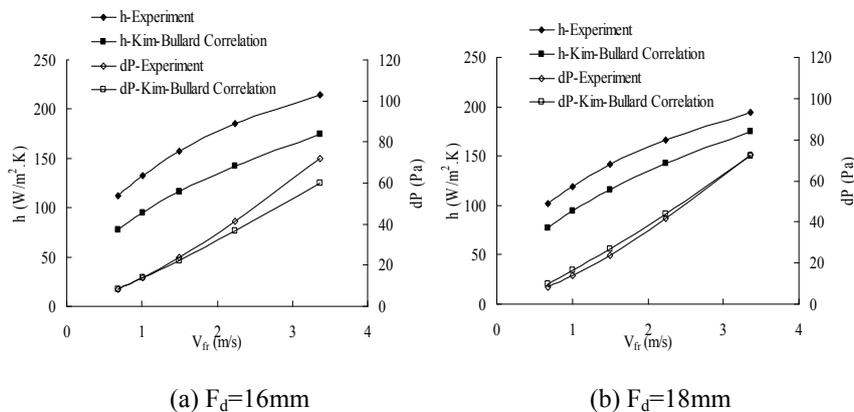(a) F_d=16mm (b) F_d=18mm

Fig. 3: Comparison of heat transfer coefficient and pressure drop with experimental results and Kim-Bullard correlation calculation results

Fig.3 presents comparison of experimental results and Kim-Bullard (2002) correlation calculation results on sample 1# and 3# in table 1. It is noticed that the variation tendencies with the air velocity of heat transfer coefficient are well consistent with the Kim-Bullard correlation calculation results, although the heat transfer coefficient has some discrepancy. That is because in present study, the louver pitch of the sample 1# and 3# is 1.0 mm, while Kim-Bullard correlation was developed using the test samples with 1.7 mm louver pitch. When flow depth is constant, larger louver pitch means fewer numbers of louvers, so the effect of louver on boundary layers becomes poor. Accordingly, the air-side heat transfer coefficient is decreasing. The discrepancy of pressure drop is within ±15%. Therefore, the experimental data in this study can be regarded as reliable and can be used to validate the CFD simulation results.

3. CFD SIMULATION

3.1 CFD Model and Boundary Conditions

Fig.4 shows 3-D CFD model computational domain, which is a complete louver fin geometric configuration used in MCHX. The tube and fin with the unlouvered zone have been modeled. To reduce the computational period, only half of the fin and half of the tube are represented. The symmetry conditions are assumed on both sides of the computational domain and the periodic boundary conditions are applied to at the top and bottom of the computational domain as depicted of fig.4. In order to avoid the effect of air circumfluence, computational domain has been extended some distance downward and upward along the flow direction.
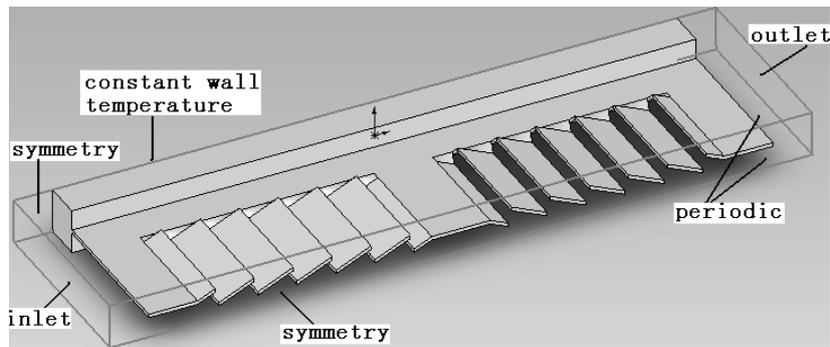

Fig. 4: Computational domain and boundary conditions

The air flow over the louvers is assumed to be laminar and steady. The model is governed by the conservation equations of mass, momentum and energy. To form a closed set of equations, ideal gas state equation can be used to relate pressure, density and temperature of air. The viscosity and thermal conductivity of the air are a function only of the temperature, the change of which is small in flow over the fin and can be assumed constant and evaluated at the mean air temperature of inlet and outlet. The viscosity is obtained from Sutherland's law and the thermal conductivity can be fitted as linear form of mean temperature.

The boundary conditions for the velocity and thermal fields need to be specified on the interface of the fin surface and fluid due to take into account conjugate heat transfer. At the upstream boundary, the flow direction velocity and temperature are assumed to be constant, with the other two direction velocities being set to zero and flow direction velocity being set to the inlet air velocity and the temperature being set to the ambient atmospheric temperature. The downstream boundary is assumed to be pressure-outlet condition. The symmetry conditions are assumed on both sides of the computational domain and the periodic boundary condition are applied to at the top and bottom of the computational domain. All solid surfaces including the louver surface and the tube surface are assumed as no-slip boundary conditions and constant wall temperature are specified.

3.2 CFD Simulation Conditions and Data Post-processing

The tube wall temperature of the CFD simulation is 45℃, the air inlet velocity is set as 2 m/s and the air inlet temperature is 35℃ . That is common condition of condenser for residential air conditioner.

The heat transfer coefficient and pressure drop can be used to characteristic air-side performance of louver fin for a given set of geometric and flow conditions. The heat transfer coefficient containing surface efficiency is defined in terms of the heat flux Q and the logarithmic mean temperature difference *LMTD*. The air-side pressure drop ΔP can be obtained directly from simulation results.

$$h = \frac{Q}{A \cdot LMTD} \tag{1}$$

$$LMTD = \frac{T_{a,out} - T_{a,in}}{\ln(\frac{T_w - T_{a,in}}{T_w - T_{a,out}})} \tag{2}$$

An integrated parameter K used to evaluate air-side performance is defined as:

$$K = \frac{(h / h_0)}{(\Delta p / \Delta p_0)^{1/3}}$$

(3)

where, h_o and ΔP_o are reference air-side heat transfer coefficient and pressure drop. The larger K value means higher air-side performance

3.3 CFD Model Validation

In this study, different zone use different meshes. At the fluid flow zone, the wedge mesh is used for larger velocity and thermal boundary layers near the wall surface excluding the fat tube. This kind of mesh is arranged from coarse to thick at perpendicular direction of wall surface. The mesh node direction is consistent with fluid flow direction. Outside fluid flow zone of the boundary layers, self-adaptive tetrahedron mesh is used. Considering the fin heat conduction and the effect of fin thickness, the solid zone also uses tetrahedron mesh. A careful check for the grid-independence of the numerical solutions has been made to ensure the accuracy and validity of the numerical results. The total number of cells in the 3-D meshes is about two million.

Fig.5 shows comparison of CFD simulations and experimental results (1# sample in Table 1) of air-side heat transfer coefficient and pressure drop. As seen in the figure, the numerical predictions of the pressure drops are in excellent agreement with the experimental results. The tendency of the numerical results is similar to those of experimental results. The numerical results often overestimated the average heat transfer coefficient. There are some possible explanations about the overestimation. Firstly, the actual boundary condition on the tube surface is not constant during the experiment. Secondly, the contact resistance between the tube and fin, which is not accounted for in the numerical simulation, is absorbed to be measured heat transfer coefficients during the experiments. When air velocity is small, due to larger measurement uncertainties, numerical results of the heat transfer coefficient are smaller than experimental results. The maximum discrepancy between experimental heat transfer coefficients and that of CFD is within 10%, and that of the pressure drop is within 9%. Therefore, the CFD model built in this study can be considered as good and reliable for predicting air-side performance.
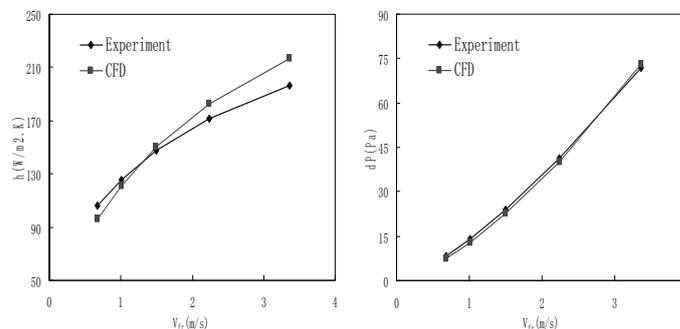

Fig. 5: Comparison of heat transfer coefficient and pressure drop with experimental results and CFD simulations

3.4 CFD Simulations and Discussions

The performance of MCHX is highly affected by the louver fin geometry. The present study mainly evaluates the influence of fin pith, louver angle, louver pitch, and louver length and flow depth on the flow pressure drop and heat transfer.

Fig.6 presents variations of heat transfer coefficient and pressure drop with fin pitch and louver angle. When louver angle increases, for small fin pitch (F_p=1.1mm), there is a maximum heat transfer coefficient. Moreover, the heat transfer coefficient for F_p=1.1mm is even smaller than that of F_p=1.6mm at 32deg. louver angle. That is because the thermal boundary layer on the louver surface for F_p=1.6mm and θ=32 deg. is relatively thinner. This phenomenon can be observed from the simulated temperature field. However, the simulated temperature field picture is not so clear to show here. The variations of pressure drop agree with conventional fin-and-tube heat exchanger. Fig.8 gives the integrated performance comparison of different fin pitch and louver angle combinations. From the fig.8, it can be seen that integrated performance is increasing as fin pitch for the same louver angle, but decreasing as louver angle for small fin pitch. For larger fin pitch (e.g. F_p=1.6mm), the integrated performance has a top point as increasing of louver angle. The optimal combination of fin pitch and louver angle is F_p=1.6mm and θ=26 deg. As seen from the

fig.7, the heat transfer rate per unit face area is decreasing with increasing fin pitch, and the pressure drop is almost linearly increasing. Therefore, when heat transfer rate meets actual requirement, the combination of larger fin pitch and smaller louver angle is a better choice.
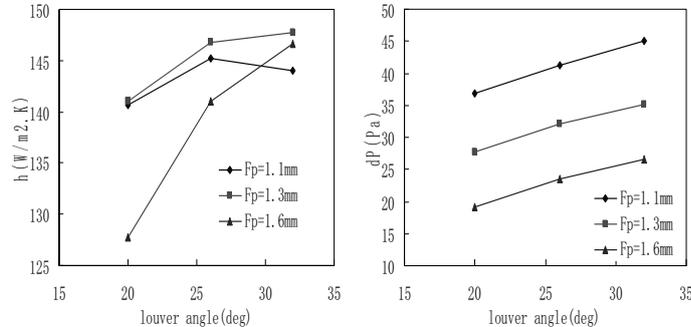

Fig. 6: Variations of heat transfer coefficient and pressure drop with fin pitch and louver angle
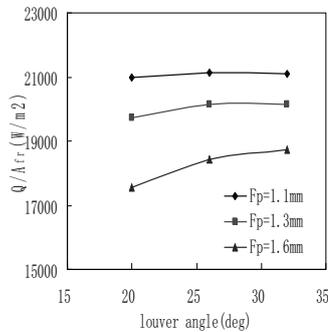

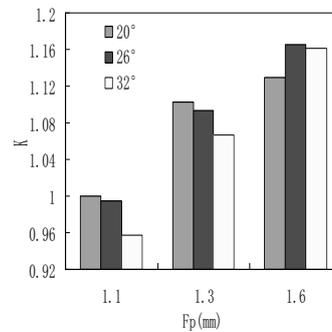Fig.7: Variations of heat transfer rate per unit face area with fin pitch and louver angle

Fig.8: Variations of integrated performance with fin pitch and louver angle

Fig.9 and fig.10 show the variations of heat transfer coefficient and pressure drop and heat transfer rate per unit face area with louver pitch and louver angle for F_p=1.3mm. The heat transfer coefficient decrease with increasing louver pitch. For small louver pitch, the heat transfer coefficient of large louver angle is higher. When louver pitch reaches 1.5mm, the heat transfer coefficient of large louver angle is lowered. The pressure drop decreases with increasing louver pitch and increases with increasing louver angle. For large louver angle, the heat transfer rate per unit face area decreases with louver pitch, while there is a maximum for small louver angle (fig.10). Fig.13 gives the comparison of the integrated performance for the combination of louver pitch and louver angle. Obviously, the integrated performance of small louver angle is better than that of large louver angle, and the former change with the louver pitch is much smaller. Hence, when heat transfer rate meets actual requirement, the combination of small louver pitch and small louver angle is a better choice.
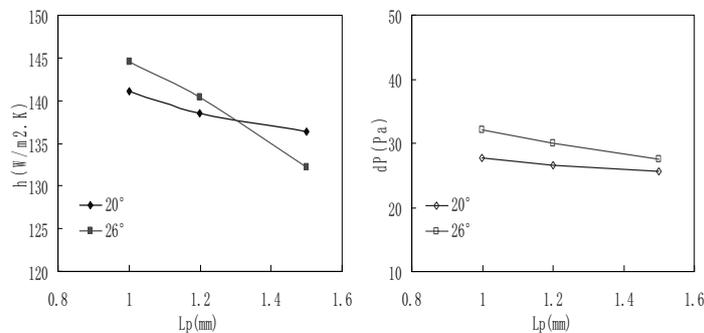

Fig. 9: Variations of heat transfer coefficient and pressure drop with louver pitch and louver angle
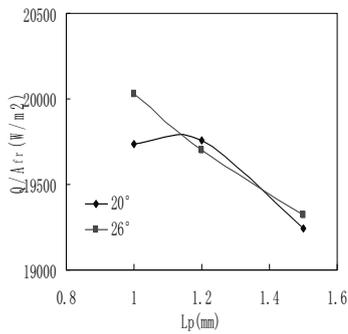
Fig.10: Variations of heat transfer rate per unit
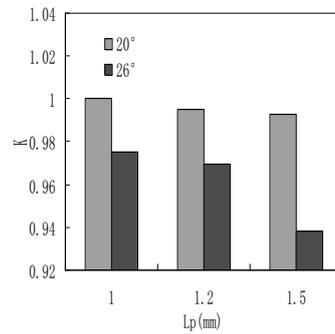face area with louver pitch and louver angle

Fig.11: Variations of integrated performance
with louver pitch and louver angle

Fig.12 and fig.13 depict the effect of louver length on the heat transfer and pressure drop characteristics for F_l=8mm and θ=20 deg. The heat transfer coefficient and heat transfer rate per unit face area increase with louver length, and the pressure drop is nearly linearly rising. The rising amplitudes of the heat transfer coefficient and heat transfer rate become smaller and smaller, and when louver length reaches over 7 mm, the heat transfer coefficient has a little decrease. From the fig.14, the integrated performance has also an optimal value for 6.53 mm louver length. In fact, the louver length can't go beyond the fin length (F_l=8mm).
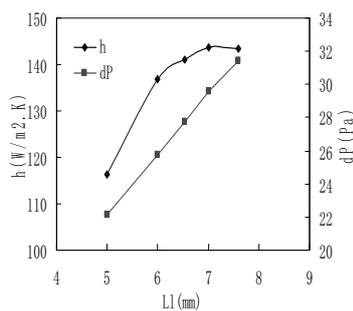

Fig. 12: Variations of heat
transfer coefficient and pressure
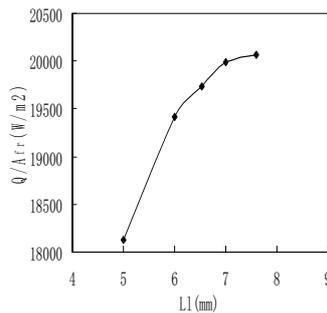drop with louver length

Fig.13: Variations of heat transfer
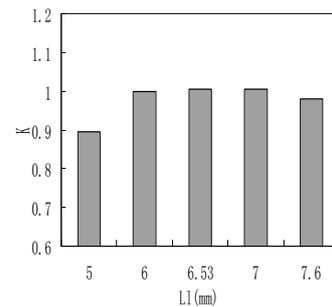rate per unit face area with louver
length

Fig.14: Variations of integrated
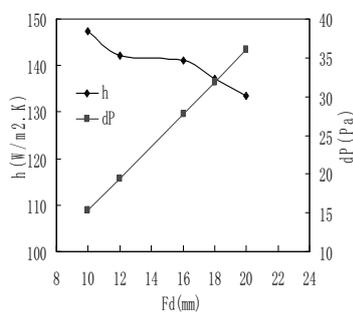performance with louver length

Fig. 15: Variations of heat
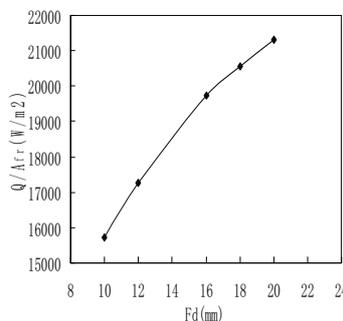transfer coefficient and pressure
drop with Flow depth

Fig.16: Variations of heat transfer
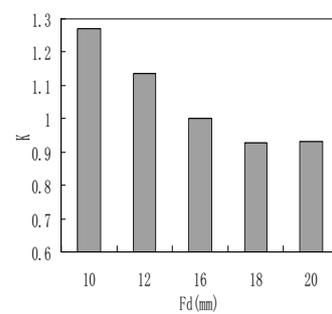rate per unit face area with flow
depth

Fig.17: Variations of integrated
performance with flow depth

It is shown in fig.15 that the heat transfer coefficient decreases with flow depth and the pressure drop linearly increases with flow depth. In addition, the integrated performance also decreases with flow depth as depicted fig.19. The heat transfer rate per unit face area increases with flow depth (fig.16). From the fig 15 and fig.16, the flow depth influences the pressure drop and heat transfer rate per unit face area significantly. So, if heat transfer rate meets actual requirement, smaller flow depth should be considered firstly. In addition, small flow depth suggests small MCHX volume.

4. CONCLUSION

The air-side heat transfer and pressure drop characteristic for MCHX with flat-tube louver fin were studied experimentally and numerically. The 3-D CFD simulation model was established and validated to predict air-side flow and heat transfer characteristics. Several important geometry configuration parameters of MCHX louver fin, involving fin pith, louver angle, louver pitch, louver length and flow depth, have been analyzed with CFD simulation method under commonly used residential air conditioner condenser conditions. Major conclusions are summarized as follows:

When heat transfer rate meets actual requirement, the combination of larger fin pitch and smaller louver angle is a better choice and small louver angle should be considered firstly for small fin pitch. Only taking into account integrated performance, the combination of 1.6 mm fin pitch with 26 degree louver angle is optimal for present study.

The integrated performance is highest for small louver pitch and small louver angle. Meanwhile, heat transfer rate per unit face area is the biggest. Therefore, small louver pitch and small louver angle are best choice.

The heat transfer coefficient and heat transfer rate per unit face area increase with louver length, and the pressure drop is nearly linearly rising. The rising amplitudes of the heat transfer coefficient and heat transfer rate become smaller and smaller. When louver length reaches over 7 mm, the heat transfer coefficient has a little decrease. The integrated performance reaches an optimal value for 6.53 mm louver length for 8 mm fin length.

The flow depth has greatly influence on pressure drop and heat transfer rate per unit face area relative to heat transfer coefficient. With increasing flow depth, the pressure drop linearly increases and the heat transfer coefficient has a little decrease. So, if heat transfer rate meets actual requirement, smaller flow depth should be considered firstly, which suggests small MCHX volume.

REFERENCES

Achaichia A., Cowell T. A., 1988, Heat Transfer and Pressure Drop Characteristics of Flat Tube and Louvered Plate Fin Surfaces. *Experimental Thermal and Fluid Science*, vol. 1: p. 147-157.

Atkinson K. N., Drakulin R., Heikal M. R., Cowell T. A., 1998, Two- and Three-Dimensional Numerical Models of Flow and Heat Transfer over Louver Fin Arrays in Compact Heat Exchangers, *International Journal of Heat Mass Transfer*, vol.41: p. 4063-4080.

Chang Y. J., Wang C. C., 1997, A Generalized Heat Transfer Correlation for Louver Fin Geometry, *International Journal of Heat Mass Transfer*, vol. 40: p. 533-544.

Davenport C. J., 1980, Heat Transfer and Fluid Flow in Louvered Triangular Ducts: PhD thesis. CNAA, Lanchester Polytechnic.

Hsieh C. T., Jang J. Y., 2006, 3-D Thermal-hydraulic Analysis for Louver Fin Heat Exchangers with Variable Louver Angle, *Applied thermal engineering*, vol. 26: p. 1629-1639.

Kim M. H., Bullard C. W., 2002, Air-side Thermal Hydraulic Performance of Multi-louvered Fin Aluminum Heat Exchangers, *International Journal of Refrigeration*, vol. 25: p. 390-400.

Park Y. G., Jacobi A. M., 2009, Air-side Heat Transfer and Friction Correlations for Flat-tube Louver-fin Heat Exchangers, *Transactions of the ASME*, vol. 134: p. 1-14.

Perrotin T., Clodic D., 2004, Thermal-Hydraulic CFD Study in Louvered Fin-and- flat-tube Heat Exchangers, *International Journal of Refrigeration*, vol. 27: p. 422-432.