Effect of Dean Vortices on Heat Transfer Characteristics of Square Channel Spiral Coil Sub-cooled Condenser

Hardeep Singh 1) Junya Washiashi2) Jun Liu3) Mitsuhisa Ichiyanagi1) Takashi Suzuki1)

1)Sophia University, Graduate School of Science and Technology, 7-1 Kioi-cho, Chiyoda-ku Tokyo 102-8554 Japan (E-mail: hsingh85@eagle.sophia.ac.jp)  
2) Keihin Corporation  
2021-8 Hoshakuiji, Takeanawza Town, Shioya County, Tochigi Prefecture  
3) Sophia University, Department of Engineering and Applied Sciences 7-1 Kioi-cho, Chiyoda, Tokyo, 102-8554, Japan  

Received on December 11, 2018

ABSTRACT: Due to the excellent heat transfer characteristics, the spiral coil sub-cooled condenser(SCSCC) has a bright application for the prospective automotive air-conditioning system. The engine load and emissions can be diminished by increasing the coefficient of performance of the vehicular A/C unit because this load is the second power consumption unit after the engine. Experimentally, two different cross-sectional channel of SCSCC has been analyzed and then compared with CFD investigation. The CFD results revealed the generation of Dean vortices inside the refrigerant and illustrate the co-relationship among heat transfer rate and pressure loss inside SCSCC.

KEYWORDS: Computational fluid dynamics, Spiral coil, Sub-cooled condenser, Dean Number, Secondary flow (D1)

1. Introduction

No doubt that vehicles are the backbone of humankind. However, the high dependency of vehicles steered to a high carbon deposit. Thus vehicles have to create an adverse effect on mother nature. Also, the Air-condition (A/C) system is primarily used to enhance human comfort during transportation. This A/C unit increases the extra load on the engine and leads to more vehicular emissions. During the study, around 7.1 billion gallons of gasoline per year has been consumed by vehicles for providing the A/C system in developed countries like the US. This high consumption has made automobiles A/C units as the most power consumption of fossil fuels vehicles succeeding engine [1]. Nevertheless, the refrigerating cargo system not only enhanced the self-life of numerous consumable food products but also help to preserve various medicines during transportation. The negative impact of refrigeration cargo units leads to more usage of fossil fuel causes global warming and pollution. Therefore, vehicular emissions and improved fuel economy can be possible to reduce the A/C load on the engine. This load can be decreased by improving the coefficient of performance of the air condition unit. Undoubtedly, the compressor unit is the backbone of the A/C cycle. However, in this research, the author paid attention to a condenser in order to achieve the goal. The embryonic research of this paper is toward the passive system, i.e., geometrical change [2-4]. Although, the outcome of the spiral coil heat exchanger was identified about a century ago recently their attention is more paid towards automobile applications. From previous studies, the spiral coil sub-cooled condenser is providing a better heat transfer coefficient towards the flat type sub-cooled condenser [5] because of the induced secondary flow inside the refrigerant [6-7]. The effect of Dean vortices has been concluded during the studies which have been changed along the variation in the square cross-section of the spiral coil sub-cooled condenser [8-9].

2. Experimental setup & CFD Analysis

2.1 Experimental setup

Two different kinds of SCSCC with different square channel/fin have been utilized for experiment. After performed the analysis, these experiments were carried out inside the calorimeter, which is the backbone of the experimental setup and used to evaluate the coefficient of performance. This device is used to maintain the desired temperature along with the humidity in a control volume. This artificial environment is created by using Calorimeter. Figure 1 (a), depicts the schematic overview of the calorimeter as well as the various experimental equipment which are parts of the experimental setup. The measuring range of the A/C cycle tester is from 700W-1600W and manufactured by Chino Corp. Sheated type K-thermocouples were used to calculate the refrigerant as well as air temperature. The range of this type of thermocouple is varied from 0 °C to 200 °C with the accuracy of ±0.75%.

The liquid refrigerant is condensed from the gaseous state by using the conventional flat type condenser. After condensation, the liquid refrigerant circulates towards the receiver. The function of the receiver is to separate the liquid refrigerant from the gas refrigerant. After the receiver, the only liquid refrigerant
Fig. 1 (a) Schematic overview of the calorimeter (b) Sub-cooled condenser with refrigerant flow line sensors

flows towards the spiral coil sub-cooled condenser. The square cross-sectional channel is provided in the spiral coil sub-cooled condenser. Inside the spiral channel, there is a generation of centrifugal forces that provided scrolled flow and led to the secondary flow generation. Although to understand the effect of the fins on the spiral coil condenser, two different fins, i.e., flat and wave fins with the same area is provided outside the coil.

The SCSCC is connected with various inlet and outlet pressure gauge, temperature gauge along with the mass flow rate device. Figure 1(b) illustrates the SCSCC along with various sensors/gauges. Moreover, the two different types of spiral coil sub-cooled condenser are representing in figure 2(a) and 3(a). The square channel with a different cross-sectional area is illustrating in figure 2(b) and 3(b). Moreover, a fluid volume that is running inside the two different square channels having spiral shape has been shown in figure 2(c) and 3(c). These wind tunnels are dedicated to the flat tube/sub-cooled condensers individually. For providing the variable air velocities, two DC variable speed propeller fans were individually used inside the wind tunnel. The

The accuracy of this sensor is 2% at full scale along with operating temperature conditions from -40 °C to 93 °C. The make and model number of this velocity sensor is Omega and FMA900A respectively. The refrigerant mass flow rate is calculated by using a Coriolis type flow meter which is installed just after the receiver in the liquid refrigerant pipe. The mass flow rate is of Keyence Corp with model number FD-SS 20 ASO (3033). The range of this sensor is from 0 to 20L/min along the accuracy of ±1% full scale.
which are submitted in equation (1) and (2), where \( \dot{q} \) represents the energy of the fluid, \( T_1 \) is the temperature of the air, \( \mu \) is the fluid viscosity, and \( g \) is the body accelerations acting on the continuum and \( \nabla \) is the vector differential operator. For calculating the temperature as well as heat flux, the energy equation is utilized and narrates below.

\[
\frac{\partial (\rho E)}{\partial t} + \nabla \cdot (\rho \mathbf{v} (E + p)) = \nabla \cdot (\mathbf{K} \nabla T)
\]  

(3)

where \( E \) [J] is the energy of the fluid, \( T \) [K] is the temperature of the fluid, \( k_{eff} \) [W/mK] is the effective conductivity.

Air/Refrigeration side CFD results are based upon the calculation model and iterative solution method, which are shown in Tables 2 and 3. In obtaining the refrigerant heat transfer characteristics and physical parameters, the value of the air heat transfer coefficient needs to insert in the boundary condition. Because the geometry is very complex, hence the real and CFD geometry is different from each. So a unique kind of relationship is developed among the real as shown in figure 2(a) and figure 2(c), i.e., CFD geometry. That relationship is known as area ratio and defined as the ratio of an area of a condenser with fins to the condenser area without the fin. The evaluation of heat transfer from the condenser by CFD without fin geometry as shown in figure 2(c). \( \dot{Q}_{air} \) [W], is expressing as below.

\[
\dot{Q}_{air} = A_{air} h_{air} (T_{inr} - T_{a})
\]  

(4)

where \( A_{air} \) [m²] is the area without a fin, \( h_{air} \) [W/m²K] is the heat transfer coefficient without the fin, \( T_{inr} \) [K] is the temperature of the wall, \( T_{a} \) [K] is the temperature of the air. Similarly, the heat transfer from the condenser by CFD for real geometry with the fin as shown in figure 2(a). \( \dot{Q}_{air} \) [W], is termed as following.

\[
\dot{Q}_{air} = A_{air} h_{air} (T_{inr} - T_{a})
\]  

(5)

where \( A_{air} \) [m²] is the area with a fin, \( h_{air} \) [W/m²K] is the heat transfer coefficient with fin. Since \( \dot{Q}_{air} \), it is equal to \( \dot{Q}_{air} \), the heat transfer coefficient of the airside is represented as follows.

\[
h_{air} = \frac{A_{air}}{A_{air}} h_{air}
\]  

(6)

For refrigerant CFD analysis, air side heat transfer coefficient, \( h_{air} \), is required. For finding \( h_{air} \), the \( h_{air} \) is firstly computed by airside calculation, and then this value is submitted in equation (6) to finding out the heat transfer coefficient \( h_{ref} \). In tables 1 and 2, \( V_{ref} \) [m/s] is the velocity of air, \( \dot{m} \) [kg/s] is the mass flow rate of refrigerant, \( T_{in} \) [K] is the refrigerant inlet temperature and \( P_{in} \) [MPa] refrigerant inlet pressure.

**Table 1 Experimental condition with variable air velocity**

| S.No | 1 | 2 | 3 | 4 | 5 |
|------|---|---|---|---|---|
| \( V_{ref} \) | 1.5 | 2 | 2.5 | 3 | 3.5 |
| \( \dot{m} \) | | | 0.007 | | |
| \( T_{in} \) | | | 327.6 | | |
| \( P_{in} \) | | | | 1.5 | |
Table 2 Airside viscous model and approach for the iterative solution method

| Viscous Model       | Laminar     |
|---------------------|-------------|
| Solution Method     | Simple      |
| Gradient            | Least Squares Cell-Based |
| Pressure            | Second Order |
| Momentum            | Second order Upwind |
| Turbulent Kinetic Energy | First Order Upwind |
| Gradient            | Least Squares Cell-Based |

Table 3 Calculation model for refrigerant side and iterative solution method

| Viscous Model     | RNG k-ε     |
|-------------------|-------------|
| Near-wall Treatment | Non-equilibrium wall function |
| Solution Method   | Simple      |
| Gradient          | Least Squares Cell-Based |
| Pressure          | Second Order |
| Cell zone conditions | Fluid air |
| Momentum          | Second order Upwind |
| Turbulent Kinetic Energy | First Order Upwind |
| Turbulent dissipation rate | First order upwind |

The 3D axisymmetric model is developed using volume-based computational fluid dynamics (CFD) software ANSYS FLUENT -15. Various grid sizes with cells 6278378(low), 6683783(medium) and 6756798(high) have been used to check their effect on simulation results. Grid independence check suggested a final grid size 6683783(medium), as it does not show any variation with 6756788(high) grid size. Furthermore, previous work evaluated the numerical error (Discretization error) of 2.6%. Moreover, convergence criteria for continuity, momentum, and energy equation is taken as $10^{-3}$, $10^{-3}$ and $10^{-3}$, respectively.

2.3 Evaluation of Experimental Result

The flow arrangement of heat exchanger plays a crucial role in the performance of the heat exchanger. In actual practice, a cross-flow type heat exchanger is used for experimental and CFD analysis. Through which, the two fluids (refrigerant and air) are directed perpendicular to each other. Figure 4 (a) illustrates the flow directions of refrigerant and the air. Moreover, along with the length of the sub-cooled condenser, the temperature distribution is shown in Fig 4 (b). The mode of heat transfers between the refrigerant and the air is through the convection and exchanged heat through the cross-flow heat exchanger. The equation for heat exchange can be expressed as:

$$\dot{Q}_c = A_c U \theta_m$$

(7)

where $\dot{Q}_c$ [W] is the heat transfer rate, $A_c$ [m$^2$] is the surface area of airside, $U$ [W/m$^2$K] is the overall heat transfer coefficient and $\theta_m$ [K] is the logarithmic mean temperature which is expressed as

$$\theta_m = \frac{\Delta \theta_1 - \Delta \theta_2}{\ln \frac{\Delta \theta_1}{\Delta \theta_2}}$$

(8)

The equation for the refrigerant side can be written as:

$$\dot{Q}_r = m_r C_p(T_o - T_i)$$

(9)

where $T_o$ [K] and $T_i$ [K] are the outlet, an inlet temperature of the refrigerant, and $T_{out}$ [K] and $T_{in}$ [K] are the outlet and inlet temperature of the air.

However, the heat transfer for the refrigerant side $\dot{Q}_r$ can be equal to the refrigerant heat transfer $\dot{Q}_r$ given as:

$$\dot{Q}_r = A_r U \theta_m$$

(10)

where $m_r$ [kg/s] is the mass flow rate of refrigerant and $C_p$ [kJ/kg K] is the specific heat of refrigerant. In the case of steady-state condition, the air side heat transfer $\dot{Q}_a$ is equal to the refrigerant heat transfer $\dot{Q}_r$.

$$\dot{Q}_a = A_r U \theta_m$$

(11)

The overall heat transfer coefficient, $U$, is derived by using equations (7) - (10).

$$U = \frac{\dot{Q}_r}{\dot{m}_r C_p(T_o - T_i)}$$

(12)
On the other hand, the evaluation of the heat transfer equation in term of log mean temperature difference from the refrigerant can also be expressed $\dot{Q}_r$, as given below.

$$\dot{Q}_r = AU \theta_s$$  \hspace{1cm} (13)$$

where $A_s$ [m²] is the surface area of a refrigerant side, and $U$ [W/m²K] is the overall heat transfer coefficient, which is given by using equations (10) and (12).

$$U = \frac{m_c(T_{in} - T_{out})}{A_s}$$  \hspace{1cm} (14)$$

Finally, the heat fluxes for air and refrigerant side can be expressed as:

$$q_a = \frac{m_c(T_{in} - T_{out})}{A_s}$$  \hspace{1cm} (15)$$

$$q_r = \frac{m_c(T_{in} - T_{out})}{A_s}$$  \hspace{1cm} (16)$$

3. Results and Discussions

3.1 Evaluation of measurement uncertainty

The measurement uncertainties at 95% confidence intervals, $U_{RSS}$ of the temperature measurement of air, temperature measurement of refrigerant and pressure measurement were evaluated using the following equations.

$$B_m = (B_b^2 + B_t^2)^{1/2}$$  \hspace{1cm} (17)$$

$$U_{RSS} = (B_m^2 + t_\alpha \sigma_m^2)^{1/2}$$  \hspace{1cm} (18)$$

where $B_{B,T,T_m} = 1,2,3,4$ representing the bias limit. The value of $t_\alpha$ based upon the student’s t-test which is 1.96 under the present experimental conditions. The uncertainty due to total measurement errors, $U_{RSS}$, is given by the summation of the bias index, $B_m$, and the precision index, $\sigma_m$. $U_{RSS}$ of temperature measurement of air, refrigerant and pressure measurement are 0.79, 0.65 and 0.23, respectively.

Table 4 Measurement uncertainties of (a) temperature measurement of air (b) temperature measurement of refrigerant and (c) pressure measurement.

| S.No | Error factor | Units | Symbol | Accuracy |
|------|--------------|-------|--------|----------|
| 1 | Thermo-couple error | [K] | $B_1$ | ±0.75% |
| 2 | Airflow tester error | [K] | $B_2$ | 2.215 (2%) |
| 3 | Bias index | [K] | $B_3$ | 0.33 |
| 4 | Standard deviation | [K] | $\sigma_1$ | 0.367 |
| 5 | Precision index | [K] | $\sigma_m$ | 0.367 |

3.2 Comparison of experimental and CFD results

For CFD calculation, the experimental values are used as the boundary condition. Tables 5 and 7 summarize the experimental results as well as CFD boundary conditions for channel 2.7*2.7 at variable air velocity. Similarly, tables 6 and 8 summarize the experimental results as well as CFD boundary conditions for channel 4.1*4.1 at variable air velocity.

Table 5 Airside experimental results and boundary conditions for 2.7*2.7 channel for variable air velocity.

| S.No | $V_e$ | $T_{in}$ | $T_{out}$ | $T_{ref}$ | $P_e$ |
|------|-------|----------|-----------|-----------|------|
| 1 | 1.5 | 313.5 | 315.5 | 314.5 | 3.5 |
| 2 | 2.5 | 313.5 | 315.5 | 314.5 | 3.5 |
| 3 | 3 | 313.5 | 315.5 | 314.5 | 3.5 |
| 4 | 4 | 313.5 | 315.5 | 314.5 | 3.5 |

Table 6 Airside experimental results and boundary conditions for 4.1*4.1 channel for variable air velocity.

| S.No | $V_e$ | $T_{in}$ | $T_{out}$ | $T_{ref}$ | $P_e$ |
|------|-------|----------|-----------|-----------|------|
| 1 | 1.5 | 312.9 | 315.9 | 314.7 | 3.5 |
| 2 | 2 | 312.9 | 315.9 | 314.7 | 3.5 |
| 3 | 3 | 312.9 | 315.9 | 314.7 | 3.5 |
| 4 | 4 | 312.9 | 315.9 | 314.7 | 3.5 |

Table 7 Experimental results and boundary condition of 2.7mm square channel for refrigerant side CFD analysis with variable air velocity.

| S.No | $V_{r}$ | $T_{in}$ | $T_{out}$ | $P_{r}$ |
|------|---------|----------|-----------|--------|
| 1 | 1.5 | 313.5 | 313.5 | 1.70 |
| 2 | 2.5 | 313.5 | 313.5 | 1.57 |
| 3 | 3 | 313.5 | 313.5 | 1.49 |
| 4 | 4 | 313.5 | 313.5 | 1.44 |
| 5 | 5 | 313.5 | 313.5 | 1.39 |
Table 8: Experimental results and boundary condition of 4.1mm square channel for refrigerant side CFD analysis with variable air velocity.

| S.No | 1   | 2   | 3   | 4   | 5   |
|------|-----|-----|-----|-----|-----|
| $V_a$ [m/s] | 1.5 | 2.0 | 2.5 | 3.0 | 3.5 |
| $T_{in}$ [K] | 313.0 | 312.8 | 313.0 | 312.9 | 312.9 |
| $T_{out}$ [K] | 317.2 | 315.9 | 315.5 | 314.8 | 314.7 |
| $\dot{m}$ [kg/s] | 0.007 | 0.007 | 0.007 | 0.007 | 0.007 |
| $T_\in$ | 330.8 | 328.0 | 326.0 | 324.7 | 323.5 |
| $T_\out$ | 329.9 | 327.0 | 325.2 | 323.6 | 322.4 |
| $P_\in$ | 1.59 | 1.49 | 1.43 | 1.37 | 1.33 |
| $P_\out$ | 1.58 | 1.48 | 1.41 | 1.36 | 1.32 |

where, $T_{in}$ [K] and $T_{out}$ [K] is the inlet and outlet temperature of the air, respectively, and $V_a$ [m/s] represents air velocity. In tables 7 and 8, $\dot{m}$ [kg/s] is the mass flow rate of refrigerant, $T_{in}$ [K], $T_{out}$ [K] are the inlet and outlet temperature of the refrigerant, $P_\in$ [MPa] and $P_\out$ [MPa] represents the inlet and outlet Pressure of the refrigerant.

In the case of the square channel of side 2.7mm, figure 6 depicts the effect of air velocities on the heat transfer rate of the refrigerant in experimental as well as in CFD analysis. It has been visible from the figure that the approachability of CFD and experimental is the same. For the air velocity, the heat transfer rate is first decreased and then little increased. The equation 7 exemplifies that the rate of heat transfer is the function of temperature difference as well as the refrigerant flow rate. Also, in this case, it is clear from table 7 that the inlet temperature of the refrigerant is decreasing along with the change in air velocity.

Although, the maximum and average errors in between the CFD and experimental error is 9.5% and 7% respectively. As the CFD analysis is based upon the uniform temperature distribution on the fin of the spiral coil sub-cooled condenser, therefore CFD results depict higher value compared with the experimental.

For channel 4.1mm x4.1mm, figure 5 illustrates the effect of air velocities on the heat transfer rate of the refrigerant in experimental as well as in CFD analysis. It has been visible from the figure that concerning air velocity, the heat transfer rate in CFD as well as in the experiment is increasing. The equation 10 exemplifies that the rate of heat transfer is the function of temperature difference as well as the refrigerant flow rate. Moreover, in this case, it is clear from table 8 that the mass flow rate is almost constant, but the inlet/outlet temperature difference of refrigerant is increasing along with the change in air velocity. Although, the maximum and average errors between the CFD and experimental error is 18% and 10.5% respectively. As the CFD analysis is based upon the uniform temperature distribution on the fin of the spiral coil sub-cooled condenser, therefore CFD results depict higher value compared with the experimental.
is the high fin efficiency. At high velocity, the experimental fins perform better and depict a higher heat transfer coefficient. The maximum error among CFD and experiment is about 14%, and the average is of 9%.

The relationship among the overall heat transfer coefficient to air velocity for the channel 2.7mm*2.7mm has been shown in figure 8. In both the cases, i.e., in experimental/CFD the overall heat transfer coefficient is increasing as varying with the air velocity. It has been clear from the figure that the approachability of the overall heat transfer coefficient in the case of CFD as well as in experimental is correlative. The errors among the CFD and experimental values are 13% maximum and 9.5% average. The errors between them can be reduced if the CFD analysis is rectified by connecting with the fin thermal distribution analysis as this CFD results are based upon the uniform fin temperature distribution which is responsible for providing higher value compared with the experimental analysis.

The refrigerant inlet/outlet temperature differences for the square cross-section of 2.7mm*2.7mm in between the CFD and experimental values have been demonstrated in figure 10. It is concluded from the figure that the CFD represents the little higher values as compared to the experimental. In this case, the maximum error is shown in the figure is about 11% and the average error of 8.3% respectively.

The drop in pressure is a significant loss for the cooling purpose. As the pressure is dropped, the cooling effect of the cycle decreased. Figure 11 represents the effects of refrigeration side pressure drop to the air velocity. It has been observed that in the case of experimental data, the pressure drop first increases and then it remains constant. However, in the case of CFD analyses, the pressure drop is constant throughout the different air velocity. Besides, it has been noted that the pressure difference in the case of experimental is more as compared to the CFD analysis. Moreover, the maximum error is about 80%. This is because of the lubricating oil that is used with refrigerant.

Figure 9, exemplifies the co-relationship amid refrigerant inlet/outlet temperatures and the air velocity for the channel section of 4.1mm*4.1mm. This figure narrates that more the temperature difference leads to more heat transfer. In this case, the CFD analysis computed the higher value as compared to the experimental value. This is because of the higher heat flux computed during the CFD calculation. The maximum and average error between the CFD and experimental is 73% and 36% respectively.
inside the refrigerant cycle. This lubricant oil has high viscosity and may provide the more pressure loss. Furthermore, during manufacturing, a special kind of sealant is used as an adhesive between the inner (spiral body) and outer body (body with fin) of SCSCCC. This sealant has a low melting temperature and gets melt when heat is given to the SCSCCC so that it makes a bond between the bodies. While melt down, the extra sealant moves towards the galleries and offered resistance to the fluid.

3.3 Effect of secondary flow

![Graph showing effect of secondary flow](image)

Fig.12 Inside the square channel of 2.7mm and 4.1mm located at cut-section of the spiral coil sub-cooled condenser as shown in figure 2 (b) and 3(b). Refrigerant velocity magnitude distribution along the X direction in both the cases.

The generation of secondary flow inside the refrigerant is the most advantageous point of the spiral coil heat exchanger. This secondary flow plays a crucial role in enhancing the heat transfer rate. The enhancement of heat transfer is because of the high velocity as well as higher kinetic energy towards the outer side of the coil. Figure 12 represents the velocity magnitude of the refrigerant along with the variation of 2.7mm square channel to 4.1 square channels at the same plane. It is clear from the figure that the secondary velocity magnitude of refrigerant is the function of the cross-sectional area of the channel. No doubt that with the increase in the cross-sectional channel area, the heat transfer rate is increased because of the higher surface area. On the other hand, the primary flow velocity magnitude decreases and leads to the weak generation of secondary flow. Other more effects of secondary flow generation like the turbulent kinetic energy are found and compared in between the 2.7mm and 4.1mm square channel of the SCSCC.

Turbulent kinetic energy is one of the most crucial factors which is utilized to find out the effect of secondary flow generation. Higher the kinetic energy toward the outer side of the wall results in more secondary flow generation and leads to higher heat transfer. It is clear from the figure 13(a) and (b) that the in case of 4.1mm square channel the kinetic energy comparing to the 2.7mm square channel is very low from the inner side of the coil toward the outer side. This low kinetic energy leads to lower heat transfer rate by the Dean effect, i.e., secondary flow generation.

![Graph showing turbulent kinetic energy distribution](image)

Fig. 13 (a) and (b) inside the square channel of 2.7mm and 4.1mm respectively located at cut-section of the spiral coil sub-cooled condenser as shown in figure 2(b) and 3(b). Refrigerant kinetic energy distribution along the Y direction in both the cases.

4. Conclusion/outcome and future plan

The reason behind this study is to analyze the effect of the square cross-sectional channel of the sub-cooled condenser on the thermos-fluid parameters. Firstly, two unique kinds of sub-cooled prototype condenser with different square channels and fins were manufactured and then analyzed at different air velocities. During the examination various heat transfer characteristics and other parameters have been found and concluded as follows:

- Among the experiments, the CFD analysis has been carried out and then these results were validated experimentally. During the results, it has been found that the relation between the experimental and CFD data is almost comparable except the pressure loss.
- It has been investigated from the CFD analysis, that the cross-sectional area of the channel directly affects the phenomena of subsequent flow generation. A higher cross-sectional area leads to more heat transfer as an increase in the Reynolds number but decreases the effect of secondary flow generation. It means that the spiral channel is behaving...
like the conventional type straight tube sub-cooled condenser.

- Besides, the proposed work will more be focused on the pressure difference. In the case of 4.1mm square channel, the CFD analysis is failed to measure the refrigerant inlet/outlet pressure difference. So, future work will be more focused on the pressure drop as it is the most critical factor for increasing the coefficient of performance. Moreover, for neglecting the effect of lubricating oil, a new kind of bench test is under construction and will be used for further investigation.

This paper is written based on a proceeding presented at the 2018 JSAE Congress (Autumn) held at Nagoya from 17 October to 19 October 2018.

References

[1] Johnson VH: Fuel used for vehicle air conditioning: a state-by-state thermal comfort-based approach (National Renewable Energy Laboratory, Society of Automotive Engineers Inc.), SAE paper, 01-1957, (2002), doi:10.4271/2002-01-1957.

[2] Maradiya C, Vadher J, Agarwal R: The heat transfer enhancement techniques and their Thermal Performance Factor, Beni-Suef University Journal of Basic and Applied Sciences, 01, pp. 1-21(2017). doi: 10.1016/j.bjbas.2017.10.001.

[3] Bergles A.E, Heat: Transfer Enhancement of Heat Exchangers (Kluwer Academic Publishers, Dordrecht (1998)).

[4] N. Sahiti, F, Durst, A. Dewan: Int J Heat Mass Tran, 48, pp. 23-24, (2005), doi: 10.1016/j.ijheatmasstransfer.2005.07.001.

[5] W.M. Rosenhow, J.P. Hartnett, E.N. Ganic: Handbook of Heat Transfer Applications (McGraw-Hill, New York, 1985).

[6] Kumar V.K.R., et al.: CFD Analysis of a Helically Coiled Tube in Tube Heat Exchanger. 5th International Conference of Materials Processing and Characterization (ICMPC 2016) Materials Today: Proceedings, 4, pp. 2341-2349, (2017)

[7] Dean W R: Note on the motion of fluid in a curved pipe. Philosophical Magazine, 4(20), pp. 208-233, (1927).

[8] Tawat S, Tanong K, Atipoang N: Enhancement of the fin efficiency of a solid wire fin by oscillating heat pipe under forced convection. Case studies in thermal engineering,2, pp. 36-41, (2014), doi:org/10.1016/j.csite.2013.10.003.

[9] S.V. Patankar, V.S. Pratap, D.B. Spalding.: Prediction of turbulent flow in curved pipes, 3, pp. 583-595, (2006). doi:org/10.1017/S0022112075000481.