Flow Disturbing to Enhance Heat Transfer Inside a Duct: an Experimental Study

Z.J. Ibadi\textsuperscript{1}\textsuperscript{a}, H. A. N. Diabil\textsuperscript{2}\textsuperscript{a}

\textsuperscript{a} Mechanical Dept, Engineering College, Kufa University, Najaf, Iraq
\textsuperscript{1}\textsuperscript{a}zuheirj.alassadi@student.uokufa.edu.iq

Abstract. In the present experimental work, the effect of air circulation on increasing heat transfer rates within the duct was studied. Three air circulation speeds are implemented: 2400, 1800, and 1200 rpm. In addition, the effect of the distance between the heat source and the location of the circulating fan on heat transfer rates was investigated using three different distances: 20, 40, and 60 cm. The Exhaust fan, placed at the outlet of the duct, changed its speed to three values: 2850, 2140, and 1425 revolutions per minute. The Reynolds range ranged from 65,000 to 175,000. The results showed that the best thermal performance is achieved when the exhaust fan speed, air circulation speed, and the distance between the heat source are 1425 rpm, 2400 rpm, and 60 cm, respectively.

Keywords: Heat transfer, Air duct, Air circulation, Reynold number, Nusselt number, friction factor.

1. Introduction
Forced heat transfer by convection is either calculated and solved analytically and its relationship to fluid dynamics, with primary attention and focus on clarifying and understanding the physical and physical causes related, or the difficulty of obtaining analytical solutions to the problems of convection and pressure. Experimental use in solutions is an important technique for simplifying complexities and obtaining improved information in design and continuous development processes. In addition to securing the most accurate data, it increases the physical understanding of heat transfer processes. The results of empirical data are usually expressed either in the form of tables, graphs, or graphical formulas and equations for use with maximum generality [1]. In [2], the heat transfer performance of five different rectangular channels with sides of different angles was studied to examine the effects of Reynolds number, angle of attack, and aspect ratio in the channels and explain the effect on the pressure gradient in the channels. And heat transfer. Their results showed that the close aspect ratio channel gave much better heat transfer performance than the aspect ratio channels. High taking into account the aspect ratio of the channel. A raised air duct improves heat transfer and friction factor for short rectangular fins attached to the duct wall. The research analyzes the important heat transfer properties of the inner surface of the fins of the co-spin type. It was observed that the co-rotation pattern showed the most patterns and the best strengthening of the heat transfer between the four different fins patterns: heat transfer details and average heat transfer coefficient. The results of [3] showed the effect of the distribution on heat transfer using a trapezoidal channel consisting of two paths. Heat distribution is mainly influenced by turbulence patterns and secondary and complex flows generated by the channel geometry, ribs, and 180° rotation. Increased turbulence of flow enhances thermal performance. The experimental and numerical study in [4] presented the effect of flow circulation on heat transfer and pressure gradient in a two-way cooling duct near the engine. The rotating blade has been rotated 180 degrees. The resultant that results in a curvature of the surface geometry in the channel with the vane rotation has an important influence on the flow behaviour and the turbulence caused by the rotation. As shown in [5], numerical
investigation of a central air circulation system for the ventilation layer in the ducts and the roof of the phase change material for convective control. The central air circulation system balances the convection of forced heat exchange, collecting heat by the ventilation layer in the ceilings; Thus, improving efficiency in the air circulation path to store the heat energy of hot air by collecting solar heat. Measurement results for an experimental house were analyzed, and numerical simulations were performed to calculate the effective heat and high convection probes. Quantitative simulation and experimental results indicate that a significant reduction in energy consumption for temperature can be achieved by using the proposed system, which uses natural energies to reduce to control the internal temperature. In [6], the effect of the flow field on heat transfer was studied experimentally and numerically. Two internal paths of a cooling duct were made using a unique rotary and non-rotating motor geometry, which was experimentally and numerically examined in the comprehensive test of devices. Pressure drops and flow loss for the same channel. Geometry is checked in a small-scale model in non-rotation rotation. The study [7] presented that experimental procedure to investigate the effect of air duct circulation height on heat transfer enhancement. Deck affixed to arrays (7*7) of rectangular and short fin-type co-rotation in the channel. To measure heat transfer in the inner wall with fin base. An infrared camera from TVS 8000 was used. The pressure drop and heat transfer were calculated in the rotating joint fin Varied style duct height from 20 to 50 mm. The friction factor was calculated, and the pressure drop that relatively greater friction occurs in smaller air ducts and the friction factor decreases with increasing Reynolds number. Effect of airway height on average heat transfer in a region showed that heat transfer initially increases with air channel height and eventually decreases with the opposite. Heat transfer and heat transfer coefficient for channel height 25mm is the optimum duct height for maximum thermal performance value. Also achieve significant thermal improvement, 2.8-3.8 times of smooth surface. In the low Reynolds region with a swivel, the fin is used in the channel. The numerical study was presented [8]. For a circular duct using a fixed fan once and rotating again at speeds of 500 rpm, 1500 rpm, and 3000 rpm. Reynolds numbers used 2.7 x 105 , 3.7 x 105 , and 4.9 x 105. The features of acceleration in the numerical way are good and more acceptable, and therefore Numerical method and its realization on different flow rates and the pressure rise corresponding to the speed of rotation of the three. Compare it with the available experimental results. The results obtained from Analysis of turbulent flow through the circular channel is somewhat consistent with rotating fan. Produced Experimental and Numerical study [9] in front of Centrifugal fan used to cool the thermal processor of laptops. The study presented a comparison of experimental and numerical results with a difference of no more than 5.4% by generating a small centrifuge using an integrated scheme consisting of a fan design. Optimum model fabrication and experimental and numerical validation in simulation. The models were first selected and then tested Experimental verifications. CFD numerical simulation software was used Experimental and numerical results showed little difference within the permissible limits of the design for all cases. The experimental study relied [10] on turbulent eddy flows in the tubes. The cross-section of the tubes had an important role in the occurrence and decay of the vortex. It depends mainly on the initial velocity distributions. Changes in pressure and strain are less effective than speed. Two different groups have been selected Speed distributions. The strength of the ‘turbulence effect’, deficiencies in periodic flows, the model gives much more realistic results. In the eddy flow studied, The rotation effect is little and insignificant compared to the thin shear flow. The experimental study [11] of several variables altering the aspect ratio of the air duct, the rotation of the force on the flow, and the thermal properties of the front and back surfaces of the two-pass rotor ducts. Because the main flow is very turbulent and has an increasing velocity in the center of the channel, The vortices tend to increase the heat in the circulation area and in the entrance area in The second flow. The rotation of the duct results in a clear effect on heat transfer and flow between the front and back surfaces, which is less than changing the aspect ratio. An experimental study [12-14] in which a smooth square channel was used by comparing the use of rotation and non-rotating flow conditions result from secondary flows On the most accurate heat transfer coefficient distributions. The heat transfer coefficient occurs at the back of the channel with the circulation of the coolant passage Increases up to 3.5 times the circular tube No rotation during heat transfer. The current study aims to know the effect of air circulation inside the duct on heat transfer and thermal performance rates. A fan was placed to circulate the air inside the duct, a capacity of (54) watts, and it
operates at three speeds (S3, S2, S1) (2400, 1800, 1200) RPM, respectively. The location of the fan’s effect on the heat source changes (X1, X2, X3) (20, 40, 60) cm, respectively. The design of the duct is made of galvanized iron, completely insulated from the outside with K flex. From the inside, the heat source area is isolated from the duct wall with thermal ceramics to prevent heat transfer by conduction. The Exhaust fan at the end of the duct has a capacity of 100 watts, and how can it affect the circulation of air inside the ducts? It can be predicted that air circulation increases the turbulence of the air inside the ducts, which contributes to enhancing heat transfer.

2. Experimental Module and Devices

Air duct made of galvanized iron, dimensions 40 * 40 cm square, length 270 cm, thickness 0.7 mm manufactured by CNC machines. It is insulated from the outside with a material such as (K FLEX) and has a very low thermal resistance of 0.04-0.03Wm-1K-1. Divide into three sections. The method of linking the sections using flanges with a gasket sealant between them and tightening the screws and lock type C between every two sections. The entrance is 150 cm on the left; the test section is 60 cm in the middle section, the exhaust section is 90 on the right, which includes are conical reducer shunt with a length of 50 cm from the square section (40 * 40) to the circular section with a diameter of (30) cm. It has a 100W suction fan installed on the circular part of it. It operates at three speeds (N3, N2, N1) (2800, 2140, 1425) rpm, respectively. The test section consists of a heat source uniform at the end of the test section of the duct. It is a thermal heater (heat tube with an inner diameter of 8 mm, length 190 cm, and a power of 3000 Watt surrounded by a heat sink from fins helical spiral disc extended along the length of the tube, and insulated with thermal rock for the electrodes conductive.) Coiled uniformly inside the perimeter of the duct and on, and insulated from the duct from the outside with refractory ceramics with a thickness of 0.8 cm, as shown in Figure 1. This section contains three locations for mounting a fan to circulate the air inside the duct with spaces (X1, X2, X3) (20cm, 40cm, 60cm) respectively from the uniform heat source. Three fans circulate air at speeds (S3, S2, S1) (2400, 1800, 1200) rpm, respectively.

3. Data Extracting

One of the problems that are solved by methods of mathematical analysis is the presence of fixed mathematical relationships to describe the variables. Empirical relationships were discovered through practical experiments because mathematics is a mixture of discoveries and inventions. It explains the changing cases. The constant relationships include surface area, Reynolds number, electrical potential, heat transfer, and pressure gradient. Among the empirical relationships whose applications are within specific ranges, such as the coefficient of friction and the Nusselt number, as in:

\[ T_{A_{outlet}} = \sum \frac{T_{outlet}}{S} \]  
(1)

\[ \Delta T = T_{A_{outlet}} - T_{inlet} \]  
(2)

\[ D_h = \frac{4 (H+H)}{4H} = H \]  
(3)

\[ D_h = \frac{4A}{p} \]  
(4)

\[ Re = \frac{\rho V_{re} D_h}{\mu} \]  
(5)

\[ f = \frac{2\Delta P + Dh}{4\rho LV^2} = 0.046 Re^{-0.2} \quad \text{Blasius} \quad 2 \times 10^4 < Re < 10^6 \]  
(6)
\[
Nu = \frac{h \Delta T}{\kappa_a} = 0.023Re^{0.8}Pr^{0.4} \quad \text{Dittus – Boelter} \quad 2 \times 10^4 < Re < 10^6
\] (7)

\[
Q_t = (I \times V)
\] (8)

\[
Q_i = m \cdot c_p \left( T_{\text{Outlet}} - T_{\text{Inlet}} \right)
\] (9)

\[
Q_{\text{conv}} = \dot{h} A_s \left( T_{\text{Outlet}} - T_1 \right)
\] (10)

\[
\varepsilon = \frac{h}{\bar{h}} \left| PP \right| = \left( \frac{Nu}{Nu_0} \right) \left( \frac{f_0}{f} \right)^{1/3}
\] (11)

4. Results and Discussion

4.1 Effect Distribution of Temperature

The results of the pilot study will be discussed. First: the effect of air circulation in the duct: all cases related to the circulation of air within the duct and the location of the circulating fan from the heat source used, and the speed change is shown by the frequency changer device. The results indicate that the fan is farther from the heat source and closer to the air Exhaust inlet X3 = 60 CM, the temperature change is greater at the higher speed of the 2400 RPM circulating fan. = S3 at the lowest air Exhausts speed N1 = 1425 RPM. Fig. 2 shows the temperature changes with the change of location and the change of rotational speed. This is due to the strong effect of turbulence at high speed of the rotating fan and the reduction of the influence of ambient temperature on the heat generated. The difference in the temperature withdrawn and entering the channel Eq. 1, 2

4.2 Effect Reynold Number on Nusselt Number

Changing the effect of air circulation inside ducts at different locations and velocities of flow, the relationship between the Nusselt number and the different Reynold numbers is presented under these cases. Reynold number. Depends on the equivalent diameter of the air duct, speed, and density. As the speed values increase, the Reynolds number increases, and the Nusselt number increases under the influence of air circulation and the best among the cases, respectively (FAN X3 S3 N3, fan X3 S2 N3, fan X1 S3 N3), (X fan site effect, Rotate S fan speed for air, N speed for Exhaust fan) (310.6, 303.8, 302.9) and below in FAN X1 S1 N1 the value of the Nusselt number (182.55) the position of the fan at X3 = 60 and the highest air circulation speed is S3 = 2400 RPM. And the Exhaust speed is N3 = 2850 RPM to 310.6. This increase in the Nusselt number values shows the effect of an increase in turbulence in the flow at X3 = 60 cm from the heat source and closer to the inlet as shown in (Fig 3, Fig 4, Fig 5) at (X1, X2, X3) respectively. Eq. 3, 4, 5, 7

4.3 Effect Of Pressure Drop and Friction Factor

Friction loss resistance to airflow is caused by duct size, the roughness of duct walls, and air velocity. Pressure loss resistance to airflow is caused by components such as diffusers, coils, and filters. The pressure drops for the experimentally studied cases depend on the Reynolds number and the mass flow rate in the full range. As expected, the pressure drops increase with increasing Reynolds number. This is the same behaviour when the mass flow rate is used instead of the Reynolds number. Higher pressure drop coefficient values (FAN X1 S1, FAN X2 S1, FAN X2 S2) (131.991, 131.884, 131.668) % at N3=2850 RPM and the lowest results after the absence of air circulation, which is equal to one is (FAN X1 S3 FAN X3 S2 FAN X3 S3) (126.659, 126.745, 126.821) % at N=1425 RPM Through the results;
it is clear to us that the effect of the pressure drop has a small effect when the air is circulated inside the channel due to the clear convergence in the results. This increase with the increase in the Reynolds number. This increase in the pressure drop results from an increase in flow turbulence, as in Fig. 6. Eq.6

4.4 Effect of Effectiveness

Effectiveness is a measure of the performance of heat transfer used in the heat exchanger or ducts. The models characterize the thermal performance coefficient to enhance the improvement of heat transfer. It represents the ratio of the Nusselt number with the model to the Nusselt number without the model to the ratio of the coefficient of friction with the model to the ratio of the coefficient of friction without the model raised. The Nusselt number is the main criterion for heat transfer. The best performance for the case (FAN X3 S3 N1, FAN X3 S2 N1, FAN X1 S3 N1 (1.1967672058788, 1.200884723, 1.205587393) While the lowest thermal performance in cases (1.0114989061, 1.000814747, 1.000212372) (FAN X1 S1 N2, FAN X2 S1 N3, FAN X1 S1 N3) Whereas the lowest thermal performance in the cases. Fig. 7 shows the effect of air circulation in the duct in the changing locations and distances, a clear improvement in the thermal performance. The reason is due to the increase in turbulence in the flow as a result of the high movement of air molecules, which creates an increase in randomness and the convergence of particles, which contributes to enhancing heat transfer Eq.1

Fig. 1 Experimental model and devices

1. manometer 2. Thermostats 3. clamp meter 4. Data logger 5. Laptop 6. thermocouples k 7. frequency converter 8. Anemometer 9. K FLEX insulation 10. Exhaust fan 11. circulation fan 12. Heat source 13. Thermal ceramic insulator 14. Exhaust fan
Fig 2 Effect Distribution Temperature with location

Fig. 3. Effect pressure drop & ratio friction factor with Reynold number
4.4.1 Nomenclature

| Symbol | Description           | units  | Symbol | Description | units   |
|--------|-----------------------|--------|--------|-------------|---------|
| As     | Cross-section area    | cm²    | P      | circumference | cm     |
| D_h    | hydraulic diameter    | cm     | Pr     | Prandtl Number |        |
| H      | Height of duct        | cm     | ΔP     | Pressure drops | Nm⁻²   |
| h      | Heat transfer coefficient | Wm⁻²°C⁻¹ | Q_i     | Power source | Watt    |
| f      | Friction factor with model |         | Q_i     | Internal heat transfer | Watt    |
| f_o    | Friction factor without model |         | Q_conv | Heat transfer convection | Watt    |
| f/f_o  | Friction factor ratio | Nu_o   | Nusselt Number without model |        |
| I      | current               | Amber  | Re     | Reynolds Number |        |
| K_a    | Thermal conductivity  | Wm⁻¹°C⁻¹ | S      | Speed fan circulation air | rpm    |
| L      | Length of duct        | cm     | ΔT     | change temperature | Co     |
| m'     | Flow rate             | Kg s⁻¹ | TOutlet | Average temperature outlet | Co     |
| N      | Air Exhaust fan speed | rpm    | V      | Voltage | volt |
| Nu     | Nusselt Number with model |         | X      | Distance effect circulation fan from heat source | cm  |

4.4.2. Greek symbol

| Greek symbol | Description          | units  | Greek symbol | Description | units |
|--------------|----------------------|--------|--------------|-------------|-------|
| ε            | Effectiveness        |        | μ            | viscosity   | kg/m.s|
| ρ            | Density              | kg/m³  |              |             |       |
4.4.3 List Of Abbreviations

| Abbreviation | Meaning |
|--------------|---------|
| FAN X\(_1\) S\(_1\) N\(_1\) | Fan location 20cm, circulation speed fan 1200 rpm, N\(_1\) Exhaust fan speed 1425 rpm |
| FAN X\(_2\) S\(_2\) N\(_2\) | Fan location 40cm, circulation speed fan 1800 rpm, N\(_2\) Exhaust fan speed 2140 rpm |
| FAN X\(_3\) S\(_3\) N\(_3\) | Fan location 60cm, circulation speed fan 2400 rpm, N\(_3\) Exhaust fan speed 2850 rpm |

5. Conclusions

This paper presents a different method from previous studies. Obstructing the flow of air and increasing turbulence in the flow by circulating air within the airway at different speeds and different impact sites. The study showed the effect of the location of the rotating fan on the rate of temperature change inside the air duct. The best location for the rotation effect from the heat source is 60 cm, then 40 cm, and finally 20 cm. The effect of changing the speed of air circulation in the airway. The optimum speed for optimization is the highest air circulation speed (2400) rpm. Changing the speed of the Exhaust fan located at the end of the tube depends on the change in temperature distribution, Reynolds number, Nusselt number, and thermal performance (efficiency). The results showed that the lowest air Exhaust speed was 1425 rpm, and the best distance was 20 closers to the start of the test segment. The pressure drops and the ratio of the friction coefficient with Reynolds number, and the results showed that it increases with increasing Reynolds number, but the effect is minimal. Thermal performance coefficient decreases with increasing Reynolds number, best at the lowest Exhaust speed of 1425 rpm in the position closest to the start of the test site and the highest air circulating speed of 2400 rpm.

References

[1] R. Razgaitis, & J. ~P Holman,. A survey of heat transfer in confined swirl flows. in Future energy production systems: Heat and mass transfer processes vol. 2 831 (1976).

[2] J.S. Park J.C. Han, Y. Huang, S. Ou and, R.J. Boyle, Heat transfer performance comparisons of five different rectangular channels with parallel angled ribs, Int. J. Heat Mass Tran. 35 (11) (1992) 2891–2903

[3] Y.W. Lee, et al./Information & Management 40 (2002) 133–146 135. values, but only for those values used or needed by information consumers. In summary, the academic research included several types of studies

[4] L.Rathjen, D. K Hennecke. Elfert, M., Bock, S., and Kleinstück, R.,2001, “Detailed Heat/Mass Transfer Distributions in a Rotating Two-Pass Coolant Channel with Engine-Near Cross Section and Smooth Walls,” Heat Transfer in Gas Turbine Systems _Annals of the New York Academy of Sciences_  , R. J.

Goldstein, ed., New York Academy of Sciences, New York, Vol. 934, pp.432–439.
[5] H. Lee, A. Ozaki, & M. Lee, (2017). Energy-saving effect of air circulation heat storage system using natural energy. Building and Environment, 124,104,117. doi: 10.1016/j.buildenv.2017.08.00

[6] S. Schubert, S. O. Neumann, and, B. Weigand, 2004, “Heat Transfer Characteristics in a Rib Roughened Two-Pass Cooling Channel With Engine Near Cross-Sections: An Experimental and Numerical Study,” The Tenth International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, HI, Paper No. ISROMAC10-2004-018, Mar. 7–11.

[7] M.D. Islam a. Oyakawa b,1, M. Yaga b,1, I. Kubo The Effects of duct height on heat transfer enhancement of a co-rotating type rectangular finned surface in duct Experimental Thermal and Fluid Science 33 (2009) 348–356.

[8] Rajwade, G. & Cui, J. Numerical study of turbulent flows through a circular duct with a rotating fan.IMECE2009-11269 November 13-19, Lake Buena Vista, Florida, USA.

[9] Lin, S. C., and C. L. Huang. 2002. An integrated experimental and numerical study of forward-curved centrifugal fan. Experimental Thermal and Fluid Science, Vol.26, pp. 421-434.

[10] Parchen, R. R., and W. Steenbergen. 1998. An experimental and numerical study of turbulent swirling pipe flows. Journals of Fluids Engineering, Vol. 120, pp. 54-61.

[11] Wang, C., Gao, P., Wang, S., Li, X. & Fang, C. Experimental study of single-phase forced circulation heat transfer in circular pipe under rolling motion. Nucl. Eng. Des. 265, 348–355 (2013).

[12] Wagner, J. H., Johnson, B. V., and Hajeck, T. J., 1991a, "Heat Transfer in Rotating Passages With Smooth Walls and Radial Outward Flow," ASME Journal of Turbomachinery, Vol. 113, pp. 42-51.

[13] Sharma, S., Singh, J., Obaid, A. J., & Patyal, V. (2021). Tool-condition Monitoring in turning process of Fe-0.75Mn-0.51C steel with coated metal carbide inserts using multi-Sensor fusion strategy: A statistical analysis based ingenious approach. Journal of Green Engineering, 2998-3013.

[14] A. Khechechhouche, N. Elsharif, I. Kermerchou, and A. Sadoun, “Construction and performance evaluation of a conventional solar distiller”, Heritage and Sustainable Development, vol. 1, no. 2, pp. 72-77, 2019.