Numerical investigation of swirl flow effect on heat exchanger efficiency according to different inlet position and Reynolds number

A B Atalar¹ and M Cakan²

¹ M.Sc., Department of Mechanical Engineering, Istanbul Technical University, Istanbul, 34437, Turkey
² Ph.D., Department of Mechanical Engineering, Istanbul Technical University, Istanbul, 34437, Turkey

atalara@itu.edu.tr

Abstract. The aim of this investigation is to determine the configuration that will enable a micro heat exchanger system to function more efficiently. Heat exchangers are devices that can be used for exchanging energy between flows, which can have a wide variety of uses and different configurations; therefore, studies are carried out in order to increase the efficiency of heat exchangers. In this study, heat transfer capacity and system efficiency of the swirl flow were investigated using CFD as compared to the parallel flow heat exchanger, which is another configuration. The air drawn by the fan is sucked into the holes and enters the system, thereby air entering the system impinges on the heat exchanger tube. The flow character has a major role on the Nusselt number and heat transfer coefficient. The main flow variables are Reynolds number and the orientation of the inlet holes. The boundary conditions were determined according to the fan capabilities and power of resistance. The positions and types of the holes were adjusted to allow for axial and tangential impact on the heat exchanger tube, and the holes were also located in order to obtain the best impingement angle. CFD simulations were carried out with ANSYS Fluent and k-epsilon turbulent model was used with coupled algorithm. The effect of laminar flow on the heat transfer were observed at different Reynolds numbers. Also the effect of swirl flow characteristics on system efficiency is clearly observed as compared to parallel flow.

1. Introduction

Energy efficiency is one of the significant factors for developing electronic devices. Especially heat exchangers are used commonly worldwide in many electronic devices, whose sizes are becoming smaller day by day.

Swirl flows in pipes; can be defined as a flow type in which the fluid particles make both a translational movement along the pipe axis and a rotation about the pipe axis. For this reason, cylindrical coordinates are used in the analysis of rotational flows. The effect of the radial velocity component is neglected in most studies, while the influence of the axial and angular velocity components is primarily investigated in the cylindrical coordinates (Scott 1973). Rotational flows enable to increase the heat transfer and swirl flows are used in heat and mass exchangers, cooling of turbine blades, combustion chambers, rockets, centrifugal separators and pneumatic conveying...
technology (Zaherzadeh 1975). Generation of swirl flows in heat exchanger configurations can be achieved in two ways. The first way is to form helical grooves to induce continuously swirl motion. This configuration creates a non-decaying swirl flow motion. The other method, which creates a decaying swirl motion is obtained through tangential flow inlets (Neto et al. 1998).

The induced rotational flows can be examined in two different subgroups. According to the geometry of the inlet type: there are tangential inlet types that may be rectangular or circular. Or according to the flow regime: laminar or turbulent (Hay et al. 1975). In addition to this, heat transfer and impingement effect were investigated according to various number of tangential inlets configurations and Reynolds number (Rao et al. 2017).

The present study investigates the role of decaying laminar swirl flow on heat transfer enhancement from the inner surface of a coaxial heat exchanger as compared to laminar flow that flows parallel to the pipe axis. The Reynolds number based on flow passage hydraulic diameter is kept below 2300 in order to ensure laminar flow regime.

2. Methodology

The study uses two different types of flow entrances to induce laminar flow in between the inner boundaries of the co-axial tube arrangement.

2.1. Assembly of Experimental Setup

The heat exchanger system consists of an axial fan, different types of inlet holes, and co-axially arranged inner and outer pipes. The inner aluminium pipe which is heated internally simulates the “warm” heat exchanger fluid, while the unheated outer Plexiglass pipe bounds the heat exchanger fluid. Axial fan aspirates the air which enters from inlet openings and the flow can be adjusted for several Reynolds numbers by means of axial fan frequency converter.

![Figure 1. Assembly of heat exchanger](image)

Parts are shown in Figure 1 for heat exchanger assembly. The fan is located above the resistance and air is aspirated along the resistance. The resistance starts from the fixed plate and continues until the fan section.
2.2. Tested Configurations
There are radial and tangential impingement possibilities on the outer surface of the inner (heated) pipe. This makes 12 different configurations because of two different types of inlet holes and various Reynolds numbers that are chosen to be investigated.

Radial and tangential entrance hole arrangements are shown in Figure 2 and it was defined respective to the axis of the heated tube. Table 1 summarizes the configurations that are examined in this investigation.

![Figure 2. Top view (z-axis) of impingement configuration for (a) Radial (b) Tangential](image)

| Reynolds no. | Hole orientation | Number of holes |
|--------------|------------------|-----------------|
| 775          | Radial           | 4               |
|              | Radial           | 2               |
|              | Tangential       | 4               |
|              | Tangential       | 2               |
|              | Radial           | 4               |
| 1160         | Radial           | 2               |
|              | Tangential       | 4               |
|              | Tangential       | 2               |
|              | Radial           | 4               |
| 1550         | Radial           | 2               |
|              | Tangential       | 4               |
|              | Tangential       | 2               |

Mass flow rate, which is adjusted by the frequency control of the axial fan, is kept constant for 4-sided and double sided inlet configuration.

3. Computational Model
Simulations were carried out using ANSYS Fluent software with computer which have 8 core-3.6GHz processor and 16 GB of RAM.
3.1. Computational Geometry and Grid
The actual geometry (Fig. 3) consists of two co-axially arranged pipes and the gap between these pipes is set as the computational domain. The aspirated air enters through the entrance holes and flow in between the pipes. The radii of inner pipe (heated) and outer pipe are respectively 11 mm and 20 mm whereas the length of both pipes is 600 mm. The diameter of inlets is 6 mm and it extends along 5 mm because of the thickness of the Plexiglass.

![Image of 4-sided configuration](image_url)

Figure 3. Isometric view of 4-sided configuration of actual geometry for (a) Radial (b) Tangential entrances

The isometric view of the computational domain is shown in Figure 4. Computational grid is unstructured and it consists of average 1,885,000 elements. There are about 15,000 elements difference between the radial and tangential grid models; however, this is not a significant value for the accuracy of the calculations.

Figure 4 shows grid model for radial configuration and there are 6 inflation layers, which have 0,15 mm thickness with 1,05 growth rate, in order to obtain the required y plus value.

![Image of computational grid](image_url)

Figure 4. Computational grid for (a) Horizontal cross section (b) Inlet section

3.2. Boundary Conditions
The heat flux of resistance is held constant for all simulations; so that the inlet hole type and Reynolds numbers stay as main parameters. All boundary conditions which is used in CFD simulations are listed
in Table 2. Reynolds numbers are respectively 775, 1160 and 1550 and the heat flux which is applied on resistance is 150 W/m².

| Reynolds Number | Heat Flux of Resistance | Inlet Configuration Type |
|-----------------|-------------------------|-------------------------|
| 775             | 150 W/m²                | 4-Sided & Double Sided  |
| 1160            | 150 W/m²                | 4-Sided & Double Sided  |
| 1550            | 150 W/m²                | 4-Sided & Double Sided  |

Simulations were carried out with 4-sided and double sided, radial and tangential inlet types and all configurations were investigated for each Reynolds numbers.

In order to apply the fan aspiration as the boundary condition, outlet is defined as a mass flow outlet. The characteristic length (D) is defined as the difference in radius between the two pipes.

Reynolds number depends on characteristic length, velocity and kinematic viscosity for internal flow. D represents characteristic length in [m] and U denotes the velocity of air in [m/s], υ is kinematic viscosity in [m²/s]. If Re is under the 2300 for internal flow, flow is designated as laminar. In this study, the air, which is aspirated from inlet, is 20°C. Also the boundary condition type for inlets is defined as pressure inlet. With all these assumptions, the boundary conditions for heat exchanger’s experimental setup are most closely matched for CFD simulations.

3.3. Solver Settings
Numerical computations were carried out by solving the 3D model, incompressible flow and steady state. There are also laminar and turbulent flow, so turbulent form of the Reynolds-Averaged Navier–Stokes (RANS) equations were performed. The flow is turbulent at the inlet section and impingement surface while the flow is laminar at the exit of the heat exchanger. Navier–Stokes equations including energy terms were performed by finite volume discretization method using ANSYS Fluent 18.0 software package.

Two equation RNG k-epsilon turbulence model is used for computations. The most significant difference between standard and RNG k-epsilon turbulent models is the additional strain rate term in epsilon equation. There is an option to modify turbulent viscosity to account for swirl. Pressure based coupled algorithm is used for pressure-velocity coupling. For the discretization of momentum, k and epsilon equations, a second order upwind scheme is used. Moreover, enhanced wall treatment is used and it is recommended to keep y⁺ values between 1 and 11 for enhanced RNG k-epsilon turbulence model.

4. Results and Discussions
In order to obtain the best heat transfer enhancement performance, different configurations were simulated using CFD software. Distributions of Nusselt number and surface temperature on the outer surface of the inner pipe is the main parameter, which indicates cooling efficiency.

The method was tested by investigating y⁺ value especially on the inner pipe surface. The highest Reynolds number configuration (Re=1550) was investigated because of the fact that higher velocity enables to higher y⁺ values. Therefore, if y⁺ is in the standard interval, the lower Reynolds numbers will not exceed interval of ideal y⁺.

Figure 5 shows that the maximum y⁺ occurs around the inlet section and the y⁺ value decreases as the air velocity decreases. In general, y⁺ value is less than 1 along the heated wall; however, in the impingement zone this value stays between 2 and 3.
Figure 5. $y^+$ distributions for (a) 4-Sided Radial (b) 4-Sided Tangential

Distribution of Nusselt number on the heated wall for different Reynolds numbers and inlet configurations are shown respectively in Figure 6, Figure 7 and Figure 8. Nusselt number is scaled between 0 and 40 in the figures and it is decreasing from inlet to outlet.

Figure 6. Distributions of Nusselt number on the heated wall at Re=775 for (a) 4-Sided radial (b) Double sided radial (c) 4-Sided tangential (d) Double sided tangential configuration.

Figure 7. Distributions of Nusselt number on the resistance at Re=1160 for (a) 4-Sided radial (b) Double sided radial (c) 4-Sided tangential (d) Double sided tangential configuration.

Figure 8. Distributions of Nusselt number on the resistance at Re=1550 for (a) 4-Sided radial (b) Double sided radial (c) 4-Sided tangential (d) Double sided tangential configuration.

While the Nusselt number exhibits values up to respectively 120 for tangential and 230 for radial at the impingement surface, it goes down to 10 towards the end of the heat exchanger. Also, these distributions are more homogenous in tangential configurations in which swirl flow dominates.

Distribution of temperature on the heated wall for different Reynolds numbers and inlet configurations are shown respectively in Figure 9, Figure 10 and Figure 11. Temperature is scaled between 293 [K] and 310 [K] in the figures and it increases as the flow picks heat and warms up from inlet to outlet.
Temperature and Nusselt number distributions are inversely proportional to each other. Where the Nusselt number is high, the heat convection is higher and the surface of the inner heated pipe is cooled more effectively. The relatively colder air strikes the hot surface and reduces the temperature by heat transfer through the heat exchanger.

Particularly when the temperature distributions of Figure 9 are examined, the temperature distributions occur along the streamlines. For all inlet configurations, streamlines are shown in Figure 12 and temperature distributions on resistance is related to the formation of streamlines.

Radial and tangential air inlet types exhibit different cooling performances from each other. Moreover, 4-Sided inlet hole configurations show worse performance than double-sided in all cases. The average heated wall temperature drop difference is 2°C between tangential and radial
impingement configurations. However, there is also 0.5-1°C difference between double and 4-sided model in the same locations.

Figure 13. Distributions of Nusselt number(left) and temperature(right) along z-direction on the heated wall surfaces at Re=1550 for hole section.

Figure 13 was plotted on the heated wall surface and along the length of the pipe. The temperature on the resistance goes up to 31°C for radial configurations; and up to 29°C for tangential configurations. In addition, Nusselt numbers were determined as a 9 at the outlet for radial type and 13 for tangential inlet types. All results clearly show that the tangential configurations are more effective than the radial ones on average.

5. Conclusion
Energy efficiency is one of the most significant factors for electronic devices and optimization studies are becoming indispensable to obtain an effective design. As heat exchangers are reduced in size and efficiency is increased, areas of use will become even more widespread.

In this study, the best configuration was obtained according to cooling performance. There are two main design options: one of them is radial air entrance and the other is tangential entrance type. These definitions are made according to the axis of the heated pipe. The variable speed fan which is located at the top of the heat exchanger aspirates air from inlet holes. A total of 12 different simulations were carried out for 3 different Reynolds number and 4 different types of designs. Reynolds numbers are respectively 775, 1160 and 15550. Moreover, there are 4-sided and double sided inlet types for tangentially and radial impingement cases. Heat flux of resistance is constant as 150 W/m^2.

Two equations RNG k-epsilon turbulence model was used in the computations. Moreover, pressure based coupled algorithm and second order upwind schemes were used in the solver. Distributions of Nusselt number and temperature were obtained for all configurations. The most effective cooling is double sided tangential type because flow streamlines are spreading independently and in a more homogenous manner in double sided configurations than 4-sided ones. When radial and tangential inlet types are compared, there are also some significant differences.

First of all, there is high Nusselt number on the impingement surface for radial inlet type because of the fact that the aspirated air which enters from inlet hits to heated surface directly. However, tangential inlet configurations enable a swirl flow and air is passing on the heated surface in a well-distributed fashion. Also the swirl flow effect can be seen clearly at the temperature distributions for tangential types.

Maximum Nusselt number reaches up to 230 at the impingement surface for radial inlet type because of the fact that the aspirated air which enters from inlet hits to heated surface directly. However, tangential inlet configurations enable a swirl flow and air is passing on the heated surface in a well-distributed fashion. Also the swirl flow effect can be seen clearly at the temperature distributions for tangential types.

A tangential model should be preferred for a well distributed cooling, but a radial inlet type model can be used if a sudden cooling around the entrance hole region is required. Also closed-bottom side
of the heat exchanger has higher temperatures in radial inlet configurations and it has negative effect on general for averaged cooling. The best cooling performance was obtained with double sided tangential configuration at the highest Reynolds number.

In future studies, optimization studies will be carried out by trying new inlet configurations to attain ideal design. CFD simulations will be experimentally verified after the test setup is established.

References
[1] Hay, N., & West, P. D. (1975). Heat Transfer in Free Swirling Flow in a Pipe. Journal of Heat Transfer, 97(3).
[2] Neto, S. R., Legentilhomme, P., & Legrand, J. (2001). Finite element simulation of mass transfer in laminar swirling decaying flow induced by means of a tangential inlet in an annulus. Computer Methods in Applied Mechanics and Engineering, 190(35-36), 4713-4731. doi:10.1016/s0045-7825(00)00341-8
[3] Scott, C. J. (1973). Turbulent annular swirl flows. Heat Transfer Lab., Univ. of Minnesota.
[4] Rao, Y., Biegger, C., & Weigand, B. (2017). Heat transfer and pressure loss in swirl tubes with one and multiple tangential jets pertinent to gas turbine internal cooling. International Journal of Heat and Mass Transfer, 106, 1356-1367. doi:10.1016/j.ijheatmasstransfer.2016.10.119
[5] Zaherzadeh, N., & Jagadish, B. (1975). Heat transfer in decaying swirl flows. International Journal of Heat and Mass Transfer, 18(7-8), 941-944. doi:10.1016/0017-9310(75)90187-8