Journal of Mechanical Engineering *Vol 18(2), 125-144, 2021*

Pressure Drop and Flow Characteristics in a Diffuser with a Dimpled Tube

*Ehan Sabah Shukri Askari**
Institute of Technology Baghdad, Middle Technical University (MTU),
Baghdad, Iraq,
**ehansabah@yahoo.com*

Wirachman Wisnoe
Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM),
Shah Alam, Selangor, Malaysia

ABSTRACT

The pressure drops and flow characteristics in a diffuser fitted with a semi-dimpled tube are numerically studied. Air is selected as a working fluid and its physical properties are modelled using pressure and velocity distribution. The study reveals that every semi-dimple act as a vortex generator. They provide the flow with intensive vortices between the dimpled surface and the diffuser wall. Therefore, they cause an enhancement in the pressure drop inside the diffuser. The performance of the dimples consisting of sphere type dimples with 5 mm in diameter. Three Reynolds number operated in the range of 25000 <Re< 50000 that is based on the hydraulic diameter of the diffuser Dh. The variation in Reynolds number is examined to further investigations of the pressure drop and flow characteristics of the diffuser. The numerical simulations are conducted using incompressible steady-state Reynolds Averaged Navier Stokes equations and the turbulence model RNG k-e is utilized in the current study. The flow characteristics of the diffuser with semi-dimpled tube are analysed and compared in terms of pressure contour, velocity profile, and velocity vectors, at the operating range of Reynolds numbers. The results are discussed to point out the flow structure mechanisms. It is found that equipping the diffuser with a semi-dimpled tube leads to an increase in the recirculation and vortices inside and near the dimples area. Therefore, they have a considerable influence on the flow field. For the diffuser equipped with a semi-dimpled tube, it is noted that the flow characteristics depend on the Reynolds number. The pressure drop in the flow direction becomes lower with increasing Reynolds number. The findings

ISSN 1823-5514, eISSN 2550-164X
Received for review: 2020-08-28
Accepted for publication: 2021-01-21
Published: 2021-04-15

indicate that for the semi-dimples, increasing Reynolds number increases the velocity distribution significantly due to better mixing of the air.

Keywords: *CFD; annular diffuser; dimpled tube; pressure drop; velocity distribution*

Nomenclature

D_i	Inlet diameter of the diffuser, mm		*Greek Symbols*
D_o	Outlet diameter of the diffuser, mm		
L	Length of the diffuser, mm	k	Turbulent kinetic energy
d_t	The diameter of the tube, mm	ε	Turbulence. dissipation
L_t	Length of the tube, mm	l	Turbulence. length scale
d_m	The diameter of the dimple, mm	I	Initial turbulence intensity
P	The pressure at the inlet, bar	$C\mu$	k-ε model parameter
u	The velocity at the inlet, $m.s^{-1}$	λ	Thermal conductivity
ρ	Air density, $kg.m^{-3}$	C_p	Specific heat capacity, $kJ.kg^{-1}$ K^{-1}
Re	Reynolds Number		

Introduction

There are many engineering applications where flow mixing, and recirculation is a necessary part. The classic types of flow characteristics enhancements such as fins, helical screw tape, dimples, etc., always increase turbulence and recirculation. Gas turbine engines are one of the most popular kinds of applications. In such engines, the product of power and efficiency increases as these types of enhancement is added. One of the important gas turbine engine components is the combustor. It directly affects engine performance. The most serious problems of the combustors are the high-pressure air coming from the compressor and the other highlighted problem how good is the mixing between the fuel and the compressed air. These two facts may cause serious problems on combustion uniformity and total pressure loss coefficient. Utilizing pre-diffuser can reduce the effects of previous problems. It is used to decelerate the flow and distribute the air around the combustor uniformly and stably. This process should be completed with a minimum total pressure loss.

To enhance efficiency and save energy with less loss, enhancement techniques are required to improve mixing and increase turbulence. Dimples are selected to generate turbulence and surfaces with dimples are easy to fabricate. The existence of dimples enhances the character of the flow. In recent years, numerous studies with numerical and experimental methods have been carried out to improve the mixing and the performance of the heat transfer by using different types of turbulence generators systems such as helical screw tapes [1]-[6], pin-fins [7]-[9], and dimples [10]-[18]. In the recent decade, many researchers used pimples as a mixing method and it could enhance the characteristics of the flow including temperature, velocity profile, etc. Zhou et al. [18] pointed out that the performance of the heat transfer in a dimpled surface is affected by the dimple geometries, including dimple shape, dimple depth ratio, and dimple diameter. They completed an experimental enquiry to search about the turbulent boundary layer characteristics when it flows over a dimpled surface and compares the results with a plain flat plate. The results found that the flow characteristics of the turbulent boundary layer flow over the dimpled surface were more complicated with much stronger near-wall Reynolds stress and higher turbulence kinetic energy, especially in the region near the back rims of the dimples.

Aroonrat and Wongwises [19] investigated the advantage of the refrigerant (R-134a) on the heat transfer and pressure drop characteristics. In a double-tube heat exchanger, the refrigerant (R-134a) was flowing into the inner tube and in the annulus, the cold water was flowing. The study focused on the inner tube with a dimpled tube and a plain tube. At the results, they found that with the dimpled tube, the heat transfer coefficient and frictional pressure drop were higher than that of the plain tube and as the Reynolds number increases, it increased Nusselt number.

To improve heat transfer by using dimples and protrusions, Xie et al. [20] suggested a modern design of an enhanced tube with dimples and protrusions to induce swirl flow. From this investigation, it is found that the geometries parameters of the dimples and the protrusions play an important role to improve the thermal-hydraulic characterizes and heat transfer rate compared with the plain tube, due to improved flow mixing. Xie et al. [21] carried out numerical research using cross-ellipsoidal dimples inside an enhanced tube to construe the mechanisms of the flow field method and heat transfer. In their research, they used longitudinal and transverse dimples. The effects of the dimple geometries (depth pitch and axis ratio) on thermal-hydraulic performance were discussed and the simulations were employed with the Re range from 5000 to 30,000. The results got that the flow mixing was improved by the downward flow of the transverse and longitudinal dimples. And the geometric parameter had been affected by the thermal-hydraulic performance significantly.

Recently, many industrial and thermal applications required new techniques to enhance heat transfer and increased thermal efficiency Wang et al. [22] applied a numerical study in a square channel with dimples. The aim was to test the bleed hole installation angle effect on the heat transfer and the flow structure. They found that utilizing a bleed hole in this channel significantly improved the thermal performance of the flow. Recently, dimples were utilized as one kind of concavity that has a significant effect on energy conservation augmentations. Qu et al. [23] investigated heat transfer and flow features in arrays of dimples inside a rectangular channel at a transitional Reynolds number. In this investigation, they used the SST turbulent model coupled with the Gamma-Theta transition model to indicate the effect of different dimple depth and different Reynolds numbers on the heat transfer performance and Nusselt number. The results displayed that under the laminar condition the average Nusselt number decreases in the flow direction while it increased when the flow became turbulent. Moreover, the study obtained heat transfer enhancement during the turbulent flow and became worse during the laminar flow. The numerical findings showed that the heat transfer performance increased with the dimples depth increases because of the induction of the turbulence kinetic energy into the main flow.

Abdulwahid et al. [24] studied the effect of an oval cross-section tube with an oval dimpled surface in a laminar flow experimentally. The non-Newtonian fluid was assumed with a constant heat flux condition and water as a working fluid. This kind of passive technique showed good results in terms of Nusselt number enhancement.

Most of the above works investigated flow features and heat transfer improvement in channels and pipes with dimples. The present research conducts numerical simulations to study pressure drop and flow characteristics in a diffuser with a dimpled tube. The study is employed sphere type dimples of 5 mm diameter and different Reynolds numbers based on the diffuser hydraulic diameter in the range of 25000 <Re< 50000. The pressure contour, velocity streamlines, and velocity vectors are numerically investigated to indicate the flow mixing mechanisms.

Setup Design Methods

Model mechanism descriptions

In the present research, a numerical method is used to be carried out to learn about the impact of dimples on the flow attribute and pressure drop in a diffuser geared up with a dimpled tube. Air is used as the working fluid in this investigation to show the effect of dimples insert. The study considers all the numerical simulations under the same inlet condition. In this research, (L=80 mm) is the diffuser length, (Di=20 mm) is the diffuser inlet diameter, and (Do=40 mm) is the diffuser outlet diameter. The tube diameter (dt) is (10 mm) and a length (Lt) is (60 mm), [25]. The spherical dimples have a diameter (ds=5 mm). The model geometries are illustrated in Figure 1.

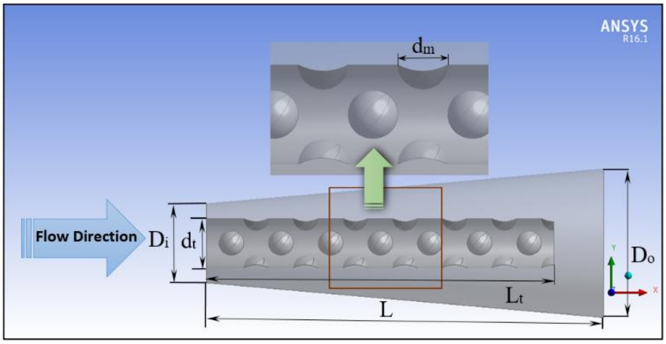

Figure 1: Model design geometries.

Boundary conditions arrangements

Some assumptions are made to design the pressure drop and flow structure of air flowing around a dimpled tube in a diffuser. They are as follows:

i. Turbulent flow with Reynolds number ranged from 25,000 to 50,000, in a steady and incompressible flow.
ii. The hydraulic diameter that is used in the Reynolds number is defined as the inlet diameter of the diffuser. The hydraulic diameter is calculated as follows, [26].

$$D_h = \frac{4\pi\left(D_i^{2} - d_t^{2}\right)/4}{\pi(D_i + d_t)} = D_i - d_t \tag{1}$$

iii. Air physical properties are considered as the dynamic viscosity (1.849e − 05 kg/m.s) and the density (1.184 kg/m^3), [27].
iv. At the inlet, the temperature is 870 K^o. The Reynolds number is ranged from 25,000 to 50,000 according to the changes of the air specified mass flow rate. The study presumes the temperature of the wall constant.
v. At the inlet of the diffuser, all the physical properties of the air are given constant.
vi. In the outlet, the static pressure is set as zero.

Computational setup

Turbulence model

The pressure drops and the flow structure for a dimpled tube and a diffuser are studied in the current investigation. The research used the

RNG k-ε as a turbulence model. It is significantly important to pick a turbulence model for a particular simulation. The turbulence model is based on the computational demands and how carefully predicted flow phenomenon. Then, the RNG k-ε turbulence model is adopted in the present study because it can provide excellent performance for flow, including recirculation and rotation [27]. The finite quantity technique is used to solve the time-independent incompressible Navier Stokes equations. The finite modeling is achieved utilizing the commercial CFD (Computational fluid dynamics) software ANSYS 16.1. The simulations are completed in three-dimensional domains employing the RNG k-ε model as a turbulence model for studying the pressure and velocity fields. Moreover, the commercial software ICEM version 16.1 is used to generate the grid method for the dimpled tube and the diffuser. This meshing tool was utilized to mesh the fluid computational domain with the unstructured tetrahedral grid as shown in Figure 2. The predictable velocity profile for five extraordinary grid independence has been compared to estimate the effect of the grid sizes on the accuracy of the numerical options results. It has been determined that 21599 factors grid is convenient for the simulation. The following transport equations were used to create the turbulence kinetic energy k and its rate of dissipation ε, [28].

$$K = \frac{3}{2}(VI)^2 \tag{2}$$

where I is the initial turbulence intensity, and V is the inlet velocity magnitude, [28].

$$\varepsilon = \left(C_\mu^{\frac{3}{4}} . K^{\frac{3}{2}}\right) l^{-1} \tag{3}$$

Here l is the turbulence length scale, and $C\mu$ is a k–ε model parameter. Its value is given as 0.09, [28]. Then the initial turbulence intensity is given as below, [28]:

$$I = 0.16(Re)^{-\frac{1}{8}} \tag{4}$$

And the turbulence length scale, [28]:

$$l = 0.07\,L \tag{5}$$

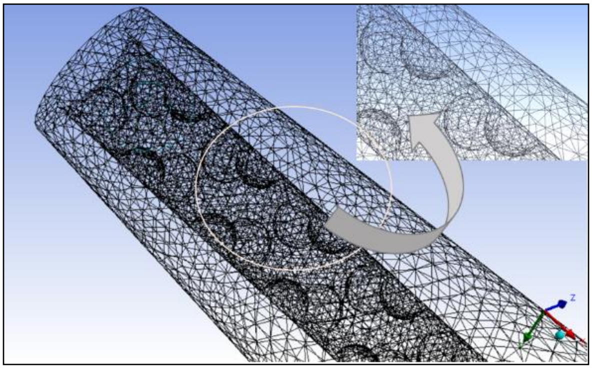

Figure 2: Schematic of the meshing generation.

Governing equations

The incompressible steady Navier-Stokes equations were solved by conservation Equations of mass, momentum, and energy. These equations mainly depict the flow characteristics and they are given by [29]:

- Continuity Equation

$$\frac{\partial}{\partial x_i}(r\rho u_i) = 0 \tag{6}$$

- Momentum Equation:

$$\frac{\partial}{\partial x}\left(r\rho u_i u_j\right) = -r\frac{\partial \mathrm{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[r\left(\mu\frac{\partial u_i}{\partial x_j} - \overline{\rho \acute{u}_i \acute{u}_j}\right)\right] \tag{7}$$

- Energy Equation:

$$\frac{\partial(\rho\bar{T})}{\partial x_i} + \frac{\partial(\rho\bar{u}_i\bar{T})}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\lambda}{C_p}\frac{\partial(\bar{T})}{\partial x_i}\right) \tag{8}$$

Where the quantity $-\overline{\rho \acute{u}_i \acute{u}_j}$ in Equation (3) is symbolized the turbulent Reynolds stresses generated by velocity fluctuations, the thermal conductivity is λ, and the specific heat at constant pressure is *Cp* [29].

Validation methods

The numerical work results accuracy is an important aim. The numerical findings of the turbulent flow through the diffuser with semi-dimpled tube were validated with the experimental data of Arora [30] and Djebedjian [28] as shown in Figure 3 and Figure 4, respectively. The validation is performed by two methods to support the reliability of the numerical work. Both the static pressure coefficient Cp and flow velocity are compared with the experiment results of the previous researchers. From Figure 3, it can be displayed that the numerical results of the static pressure coefficient Cp are in good agreement with the experimental of Arora [30], and the percentage error is within ± 2%. While Figure 4 displays a comparison of the velocity profile in the flow direction. The validation confirms the agreement of the numerical results with the experimental work of the Djebedjian [28] that presents the reliability of the adopted method within percentage error ± 4%.

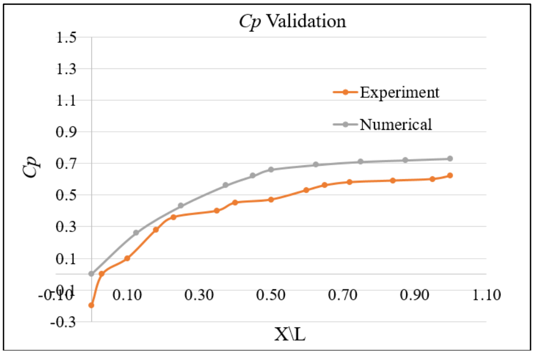

Figure 3: Numerical results validation of the static pressure loss coefficient *Cp* with the experimental data from Arora [30].

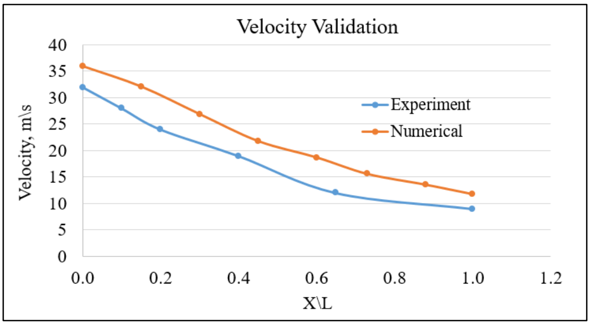

Figure 4: Comparison validation of numerical results with experimental work of Djebedjian et al. [28].

Results and Discussion

Pressure drop characteristics discussions

Based on the CFD simulations completed in the current study, the pressure drop characteristics for the airflow around the dimpled tube have been measured within the Reynolds Number range of 25000 <Re< 50000. Due to the different Reynolds numbers that are studied, the pressure contours differed clearly. Figures 5 and Figure 6 show comparison of pressures in contour and in profile, respectively versus flow direction in the dimpled tube with three various Reynolds number (Re).

By comparison, it is found that in (a) with Re (2.5 x 10^4), the pressure is the highest. Besides (b) with Re (3.8 x 10^4), the pressure will be less. Higher Reynolds Number like in (c) with Re (5.0 x 10^4), the pressure is decreased. The flow characteristics related to pressure is better with Re (2.5 x 10^4), where the pressure is highest which means the pressure drop is lowest. The figures show the increase of pressure following the increase of passage area of the diffuser.

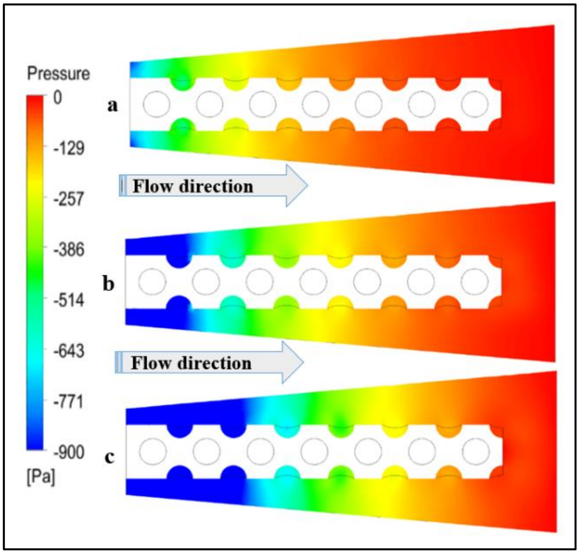

Figure 5: Comparison of pressure drop contours versus flow direction in the dimpled tube with various Reynolds number (a) Re = 2.5 x 10^4, (b). Re = 3.8 x 10^4 and (c) Re = 5.0 x 10^4.

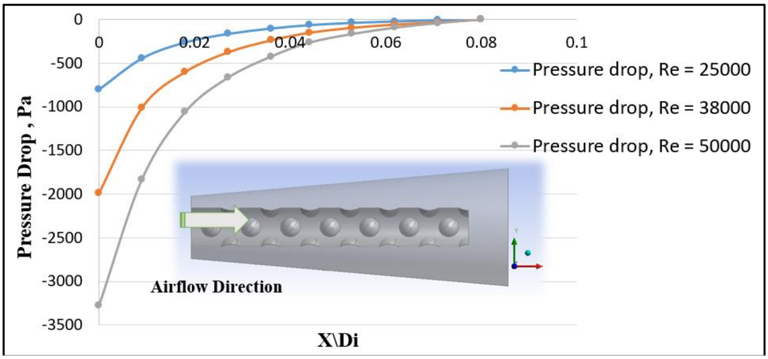

Figure 6: Comparison of pressure drop profile versus flow direction in a dimpled tube with various Reynolds number (Re = 2.5 x 10^4, 3.8 x 10^4 and 5.0 x 10^4).

Velocity distribution characteristics discussions

The following simulations are utilized to study the velocity profile characteristics in a dimpled tube during turbulence flow. The effect of Reynolds number on the velocity profile through the dimpled tube is examined. Figures 7, 8 and 9 show the velocity vectors for three different Reynolds numbers (2.5 x 10^4, 3.8 x $10^{4,}$ and 5.0 x 10^4) along the flow direction.

In terms of the effect of Reynolds number on the velocity feature, the recirculation and the tangential velocity component near the dimples area increases as the Reynolds number increased. The figures reveal that the air near the dimples area is mixed with the additional air that flowing around the dimpled tube and enhances the distributions of the velocity.

More explanations for the effect of dimples on the flow characteristics can be depicted in Figure 10, Figure 11, and Figure 12. The figures show clearly the presence of flow recirculation at the dimples area. The colour of the blue balls shows the velocity magnitude. For the three tested Reynolds numbers, recirculation and vortices are conducted at the dimples area but in different manners. The recirculation remains at the three tests but became weak with Reynolds number (5.0 x 10^4). This means that dimples can produce tangential velocities and provide the flow with high distribution intensity. However, the high distribution intensity also increases the pressure drop. It is obvious that the dimple hole cause to change the direction of the velocity inside the dimple. Thus, the cross-sectional area of the flow is increased. Songtao Wang explained this fact, as when the main flow accesses the dimple, it will be divided into two parts. The flow in some parts of the dimple has a high velocity and it makes a low velocity recirculation-zone in other parts. When the flow impinged the dimples area, the flow streamlines separate from the dimpled surface and reattach on the end wall downstream of the dimple. Then, the turbulent kinetic energy in the free shear layer is created between these two parts of the flow (the main and the reverse flow) [31].

The variation of velocity distribution in the flow direction around the dimpled tube is compared in Figure 13 for three various Reynolds numbers (Re = 2.5 x 10^4, 3.8 x 10^4 and 5.0 x 10^4). In this direction, it can be noticed that the velocity distribution decreases as the Reynolds number increased.

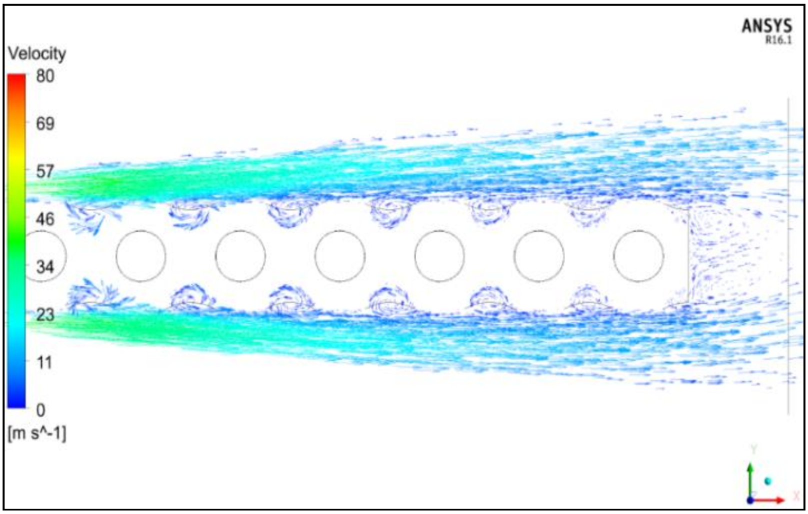

Figure 7: Velocity vectors around a dimpled tube with Reynolds number Re = 2.5 x 10^4.

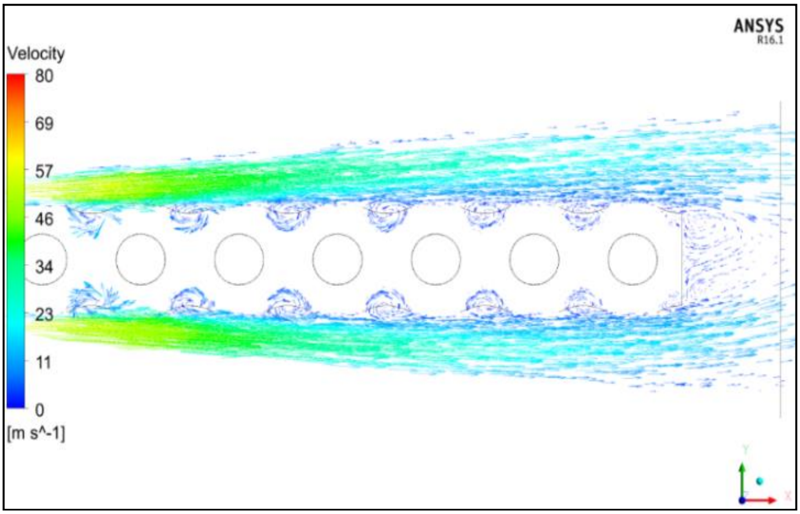

Figure 8: Velocity vectors around a dimpled tube with Reynolds number Re = 3.8 x 10^4.

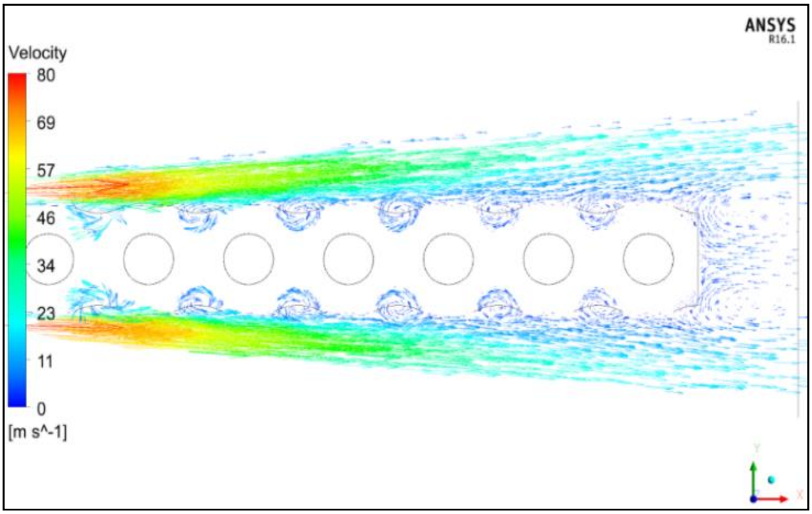

Figure 9: Velocity vectors around a dimpled tube with Reynolds number Re = 5.0 x 10^4.

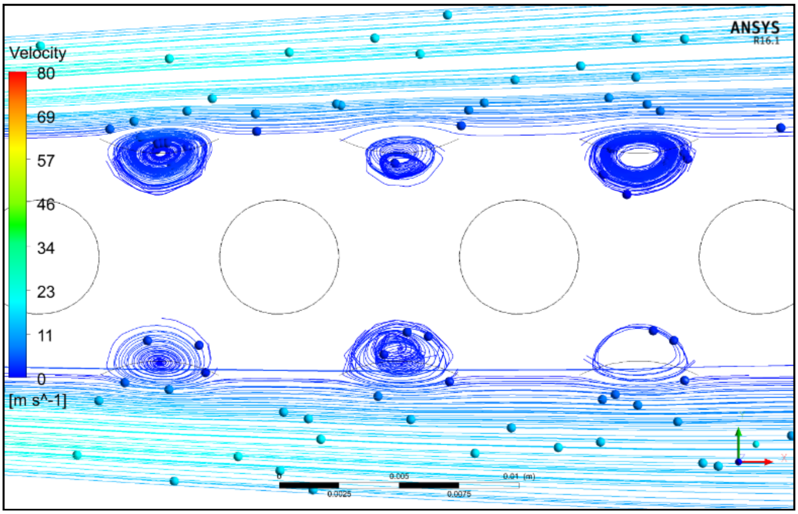

Figure 10: Velocity streamlines around a dimpled tube with Re = 2.5 x 10^4.

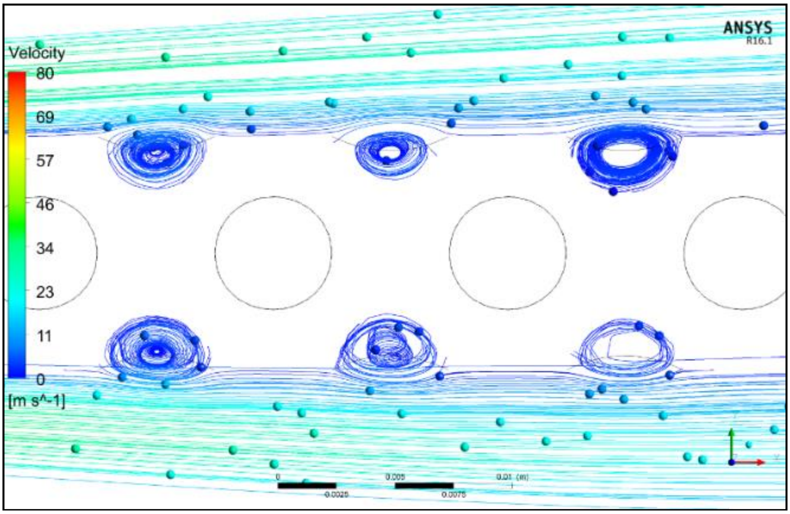

Figure 11: Velocity streamlines around a dimpled tube with Re = 3.8 x 10^4.

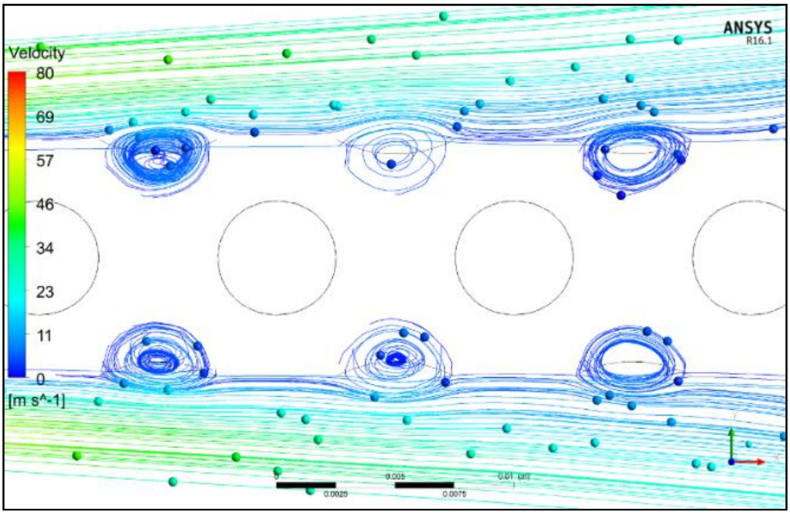

Figure 12: Velocity streamlines around a dimpled tube with Re = 5.0 x 10^4.

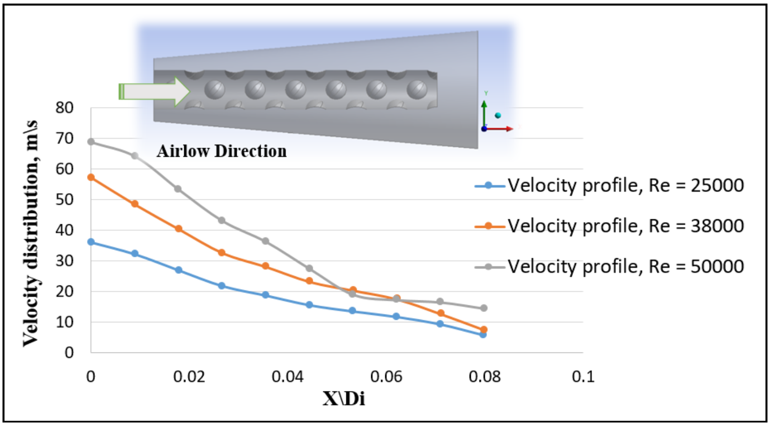

Figure 13: Comparison of velocity distribution versus flow direction in a dimpled tube with various Reynolds number (Re = 2.5 x 10^4, 3.8 x 10^4 and 5.0 x 10^4).

Velocity and pressure profile comparison in the outlet

To test the effect of dimples insertion on the outlet flow characteristics, an outlet plane is used.

The streamline velocity, the velocity profile and the pressure variation comparison with three different Reynolds numbers (Re = 2.5×10^4, 3.8×10^4, and 5.0×10^4) are depicted in Figure 14, 15 and 16, respectively. Figure 14 presents the velocity streamlines for different Reynolds numbers (Re) near the outlet of the diffuser. For a plane at the outlet of the annular diffuser behind the dimpled tube, the distribution of the velocity increased with higher Reynolds numbers. In Figure 15, it is displayed that the distribution of the velocity increases at the outlet as the Reynolds number increases because that increase disturbs the entire flow and cause more swirl flow and recirculation. At the same time, the pressure variation decreases with a high Reynolds number as shown in Figure 16.

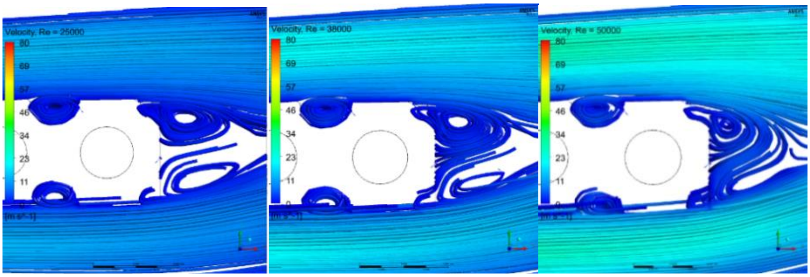

Figure 14: Streamline velocity comparison in the flow direction around a dimpled tube with various Reynolds number (Re = 2.5×10^4, 3.8×10^4 and 5.0×10^4).

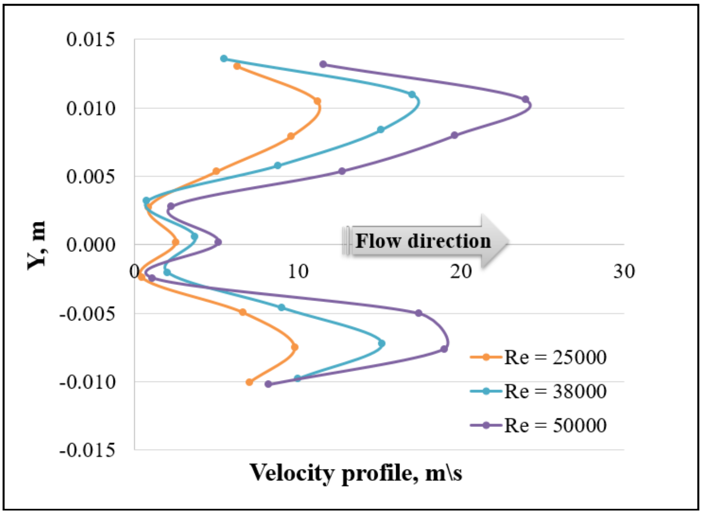

Figure 15: Comparison of velocity distribution versus radial flow direction around a dimpled tube with various Reynolds number (Re = 2.5 x 10^4, 3.8 x 10^4 and 5.0 x 10^4).

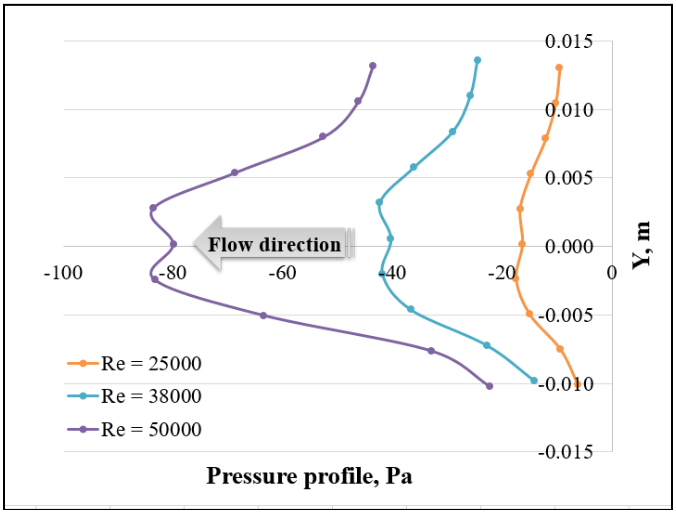

Figure 16: Comparison of pressure variation versus radial flow direction around a dimpled tube with various Reynolds number (Re = 2.5 x 10^4, 3.8 x 10^4 and 5.0 x 10^4).

Conclusions

In this paper, a numerical analysis was performed to investigate the pressure drop and flow structure of air flowing around a dimpled tube in an annular diffuser. The research is considered the boundary condition as a steady and incompressible flow. The results were obtained by using different Reynolds numbers ranging as 25000 <Re< 50000. Based on these assumptions, the following conclusion is made.

1. The numerical analyses were performed to differ the pressure drop around the dimpled tube. From the comparative analysis of pressure drop contours versus flow direction, the air pressure decreases with the increase of Reynolds number because of the increases in the flow passage area in the diffuser and the pressure losses increases due to the existence of the dimples that create recirculation and vortices causes to increase the losses.
2. Velocity simulations are utilized to study the velocity profile characteristics in a dimpled tube during the turbulent flow. It is found that the air near the dimples area is mixed with the additional air that flowing around the dimpled tube and enhances the distributions of the velocity that means more tangential velocity components. This fact leads to enhance the flow mixing through the diffuser area.
3. The recirculation and the tangential velocity component intensity near the dimples area increase as the Reynolds number increase.
4. In the flow direction, it can be noticed that with the increase of Reynolds number, the velocity profile decreases.
5. At the outlet, the velocity distribution increases as the Reynolds number increases and the pressure rate decreases with the increase of Reynolds number. The increase in Reynolds number disturbs the flow and creates more swirl flow and recirculation.

References

[1] P. R. Ingole and S. L. Kolhe, “Analysis of coiled-tube heat exchangers to improve heat transfer rate with spirally corrugated wall,” *International Journal of Modern Engineering Research (IJMER)*, vol. 4, pp. 196-175, 2014.

[2] S. Naga Sarada, A.V. Sita Rama Raju, K. Kalyani Radha and L. Shyam Sunder, “Enhancement of heat transfer using varying width twisted tape inserts,” *International Journal of Engineering, Science and Technology*, vol. 2, pp. 107-118, 2007.

[3] F. A. Saleh, "Computational and experimental investigations on thermal and fluid flow characteristics for different models of tapes inserts with TiO2/water nanofluid under turbulent conditions," *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS*, vol. 18, pp. 1-14, 2018.

[4] E. S. Shukri and W. Wisnoe, "Temperature distribution enhancement in a conical diffuser fitted with helical screw-tape with and without center-rod," *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering*, vol. 10, pp. 439-444, 2016.

[5] E. S. Shukri, "Numerical comparison of temperature distribution in an annular diffuser fitted with helical screw-tape hub and pimpled hub," *4th International Conference on Power and Energy Systems Engineering, CPESE 2017, 25-29 September 2017, Berlin, Germany, Procedia Engineering*, vol. 141, pp. 625-629, 2017.

[6] V. Arunprasad, P. Murugesan and S. Jaisankar, "Experimental studies on natural circulation solar water heating system fitted with helical screw tape inserts," *Asian Journal of Research in Social Sciences and Humanities*, vol. 6, pp. 1703-1716, 2016.

[7] A. Soni, "Study of thermal performance between plate-fin, pin-fin and elliptical fin heat sinks in closed enclosure under natural convection," *International Advanced Research Journal in Science, Engineering and Technology, ISO 3297:2007 Certified, ISSN 2394-1588*, vol. 3, no. 11, pp. 133-139, 2016.

[8] Y. S. Tomar and M. M. Sahu, "Comparative study of performance of pin fin under forced convection heat transfer," *International Journal of Engineering Research & Technology (IJERT), ISSN: 2278 – 0181*, vol. 2 no. 9, pp. 364-374, 2013.

[9] R. J. Moitsheki and C. Harley, "Steady thermal analysis of two-dimensional cylindrical pin fin with a nonconstant base temperature," *Hindawi Publishing Corporation Mathematical Problems in Engineering*, 2011, Article ID 132457, DOI:10.1155/2011/132457, 2011, pp. 1-17.

[10] F. Sangtarash and H. Shokuhmand, "Experimental and numerical investigation of the heat transfer augmentation and pressure drop in simple dimpled and perforated dimpled louver fin banks with an in-line or staggered arrangement," *Applied Thermal Engineering*, vol. 82, pp. 194-205, 2015.

[11] A. A. Tsynaeva and M. N. Nikitin, "Study of the flow in a channel with dumbbell-shaped dimples," *International Conference on Industrial Engineering, ICIE 2016, Procedia Engineering*, vol. 150, pp. 2340-2344, 2016.

[12] J. Abraham and R. Maki, “Hydrodynamics of laminar flow through dimpled pipes,” *MOJ Civil Engineering*, vol. 4, no. 3, pp. 150-154, 2018.

[13] K. A. Amsha, “Turbulence modelling of the flow and heat transfer in dimpled channels,” *A thesis submitted to The University of Manchester for the degree of Doctor of Philosophy (Ph.D.) in the Faculty of Science and Engineering, School of Mechanical, Aerospace and Civil Engineering*, 2017.

[14] S. Shin, K. S. Lee, S. D. Park and J. S. Kwak, “Measurement of the heat transfer coefficient in the dimpled channel: effects of dimple arrangement and channel height,” *Journal of Mechanical Science and Technology*, vol. 23, pp. 624- 630, 2009.

[15] E. S. Shukri, “Thermal flow analysis for annular diffuser equipped with pimpled hub,” *Journal of Mechanical Engineering, JMechE*, vol. 15, no. 1, pp. 115-134, 2018.

[16] J.Sathees Babu, “Analysis of heat transfer and fluid flow characteristics of concentric tube heat exchanger with dimpled tube,” *Journal of Xi'an University of Architecture & Technology*, vol. 13, pp. 508-516, 2020.

[17] E. S. Shukri, “Performance analysis of annulus diffusing area with divergent dimpled tube,” *The 6th International Conference on Power and Energy Systems Engineering (CPESE 2019), September 20 – 23, 2019, Okinawa, Japan*, vol. 6, pp. 11-18, 2020.

[18] W. Zhou, Y. Rao and H. Hu, “An experimental investigation on the characteristics of turbulent boundary layer flow over a dimpled surface,” *Journal of Fluids Engineering ASME*, vol. 138, pp. 021204-1-021204-13, 2016.

[19] K. Aroonrat and S. Wongwises, “Experimental study on two-phase condensation heat transfer and pressure drop of R-134a flowing in a dimpled tube,” *International Journal of Heat and Mass Transfer*, vol. 106, pp. 437-448, 2017.

[20] S. Xie, Z. Liang, L. Zhang, Y. Wang, H. Ding and J. Zhang, “Numerical investigation on heat transfer performance and flow characteristics in enhanced tube with dimples and protrusions,” *International Journal of Heat and Mass Transfer*, vol. 122, pp. 602- 613, 2018.

[21] S. Xie, Z. Liang, L. Zhang and Y. Wang, “A numerical study on heat transfer enhancement and flow structure in enhanced tube with cross ellipsoidal dimples,” *International Journal of Heat and Mass Transfer*, vol. 125, pp. 434- 444, 2018.

[22] S. Wang, W. Du, L. Luo, D. Qiu, X. Zhang and S. Li, “Flow structure and heat transfer characteristics of a dimpled wedge channel with a bleed hole in dimple at different orientations and locations,” *International Journal of Heat and Mass Transfer*, vol. 117, pp. 1216-1230, 2018.

[23] H. Qu, Z. Shen and Y. Xie, “Numerical investigation of flow and heat transfer in a dimpled channel among transitional Reynolds numbers,”

Hindawi Publishing Corporation Mathematical Problems in Engineering, 2013, pp. 1-10.

[24] Ammar F. Abdulwahid, Hyder H. Balla and Reyadh Ch. Al-Zuhairy, "Experimental heat transfer enhancement in oval dimpled tube," *Journal of Mechanical Engineering Research and Developments, ISSN: 1024-1752*, vol. 43, pp. 400-408, 2020.

[25] Y. Eldrainy, J. Jeffrie and M. Ridzwan, "Prediction of the flow inside a micro gas turbine combustor," *Jurnal Mecchanikal*, no. 25, pp. 50–63, 2008.

[26] P. Promvonge, S. Suwannapan, M. Pimsarn and C. Thianpong, "Experimental study on heat transfer in square duct with combined twisted-tape and winglet vortex generators," *International Communications in Heat and Mass Transfer*, vol. 59, pp. 158-165, 2014.

[27] E. S. Shukri, B. H. Abdullah, O. M. Ali, "Velocity distribution in an annular diffuser fitted with twisted hub for different area ratios," *4th International Conference on Power and Energy Systems Engineering, CPESE 2017, 25-29 September 2017, Berlin, Germany*, vol. 141, pp. 639-643, 2018.

[28] B. Djebedjian, M. Giat, J. P. Renaudeaux and M. A Rayan, "Numerical study of turbulent flow through equiangular annular diffusers," *Proceeding of Fifth International Conference of Fluid Mechanics, Cairo*, pp.719-729, 1995.

[29] D. Flórez-Orrego, W. Arias, D. López, and H. Velásquez, "Experimental and CFD study of a single-phase cone-shaped helical coiled heat exchanger: an empirical correlation," *Proceedings of the 25th International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems*, pp. 375-394, 2012.

[30] B. B. Arora and B. D. Pathak, "Effect of geometry on the performance of annular diffuser," *International Journal of Applied Engineering Research*, vol. 4, pp. 2639-2652, 2009.

[31] S. Wang, W. Du, L. Luo, D. Qiu, X. Zhang and S. Li, "Flow structure and heat transfer characteristics of a dimpled wedge channel with a bleed hole in dimple at different orientations and locations," *International Journal of Heat and Mass Transfer*, vol. 117, pp. 1216-1230, 2018.