International Journal of Engineering & Technology, 7 (4.25) (2018) 185-189

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Velocity and Pressure Analysis in a Divergent Area Fitted with Dimpled Hub

Ehan Sabah Shukri Askari[1]*, Wirachman Wisnoe[2]

[1]*Engineering Technical College-Baghdad, Middle Technical University (MTU), Baghdad, Iraq*
[2]*Flight Technology and Test Centre (FTTC), Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Selangor DE, Malaysia*
**Corresponding author E-mail: [1] ehansabah@itbaghdad.mtu.edu.iq.*

Abstract

A diffuser fitted with turbulators is able to enhance the distribution of both velocity and pressure. This research presents a numerical analysis for an annular diffuser type with dimpled hub. The diffuser is used to increase the static pressure by consuming kinetic energy. Three different hub models which are straight dimpled hub (SDH), divergent dimpled hub (DDH) and cone dimpled hub (CDH) are simulated to study their effects on the velocity and pressure profile for an annular diffuser. Results indicate that the insertion of the three dimpled hub models enhances velocity and pressure distribution through the diffuser. Numerical results recorded that the velocity disturbance enhanced up to 10%, 20% and 4% for the SDH, DDH, and CDH, respectively compared with the diffuser without dimples. Furthermore, the results show that the velocity distribution is the best with SDH model and the static pressure with CDH is the largest.

Keywords: *annular diffuser, dimpled hub, velocity distribution, CFD.*

1. Introduction

The annular diffuser has been used as a basic geometry to study flow behavior in several mechanical devices such as ducting, air conditioning, gas turbines, compressors, fans, pumps, wind tunnels, etc. The annular diffuser has very strong industrial significance. With an annular diffuser, good performance is possible since an inner surface is presented to guide the flow radially outward. It affords the possibility of introducing many different geometric combinations since there is now an inner surface that can be varied independently of the outer surface. Several studies have been performed to characterize the main features of the flow through a diffuser, especially using the turbulators to create a swirling flow. Swirling flow has been used as a good method to enhance velocity distribution and heat transfer in many thermal applications. This is because swirling flow is usually accompanied with high tangential velocity and turbulence intensity, which provides an additional mechanism to increase the heat transfer. Different methods of generating the swirling flow may lead to different characteristics of flow structure and heat transfer [1-6]. Swirling flow has been investigated for several numerical and experimental solutions [7-10].

Flow through annular diffuser has numerous industrial applications, and needs to be understood in more details. Anupriya and Jayanti [11] presented a study of annular flow through a vertical circular pipe experimentally and theoretically. The study provided a method for the prediction of the pressure variation across the diverging section. Experiments were carried out in air-water flows through a vertical diverging pipe section. Pressure profiles were recorded upstream, across and downstream of the diverging section. It showed for the cases of sudden expansion and gradual expansion with included half-angles of 8˚ and 15˚ for the diverging section. Findings showed that the pressure variation was characterized by a strong pressure recovery coefficient downstream of the expansion which was in turn influenced by the smoothness of the expansion.

Dimpled is used as an enhancement method in many research. Lu Zheng et al. [12] studied flow characteristics in a circular tube fitted with twisted tape inserts equipped with dimples, and effects of dimple were discussed. Al_2O_3-water nanofluid was employed to study its influence on flow and heat transfer. The results indicated that flow performance and recirculation were improved. Nares Chimres et al. [13] numerically studied the fin and tube heat exchanger with a semi-dimple pair. The studied parameters consist of the diameter, attack angle and the placed location of the semi-dimple. Numerical findings showed that the semi-dimples pair was a suitable candidate for plain fin replacement.

There are a large number of researches concerns the cooling technologies by using different cooling structures such as rib, pin, fin, and dimples. The essential purpose of these studies was to enhance the heat transfer coefficient with a low friction factor penalty. A dimple was a new technology that could be used to improve the heat transfer. A study of flow structure and heat transfer of a convergent channel with dimpled surfaces by using bleed holes was performed by Songtao Wang et al. [14]. It was found that the combination of the bleed hole inserts and the dimpled surface of the channel improves the thermal performance.

A new design of heat transfer enhancement tube by employing dimples and protrusions was carried out by Shuai Xie et al. [15]. The local streamlines, velocity contour, temperature contour and the Nusselt number was performed to describe heat transfer mechanisms and flow field characteristics.

Nomenclature			
D_i	Annular diffuser inlet diameter, mm		*Greek Symbols*
D_o	Annular diffuser outlet diameter, mm		
L	Annular diffuser length, mm	k	Turbulent kinetic energy
d_h	Hub diameter, mm	ε	Turbulence dissipation
d_L	Hub length, mm	l	Turbulence length scale
d_m	Dimple diameter, mm	I	Initial turbulence intensity
P	Inlet pressure, bar	$C\mu$	k-ε model parameter
u	Inlet velocity, m.s^{-1}	λ	Thermal conductivity
ρ	density, kg.m^{-3}	C_p	Specific heat capacity, kJ.kg^{-1} K^{-1}
Re	Reynolds Number		

The previous research works have reported that using dimples provide the flow with a good flow structure enhancement and heat transfer enhancement. However, a very small number of studies investigated the flow structure and heat transfer in an annular diffuser with dimples. The main aim of this research work is to study the effect of the dimpled hub in an annular diffuser on the flow structure and velocity distribution at Re = 2.7×10^4. In addition, three different dimpled hub models are proposed in this study.

2. General Geometries of the Physical Models

2.1. Physical Models Geometries

This work was performed to study the influences of the dimpled hub on the flow field structure, velocity distribution, and pressure distribution through an annular diffuser with air as a working fluid. The study was conducted by utilizing a numerical method and all the simulations are solved under the similar inlet condition.

Furthermore, the annular diffuser has a length (L) 80 mm, inlet diameter (D_i) 20 mm, outlet diameter (D_o) 40 mm, hub inlet diameter (d_h) 10 mm, hub length (d_L) 60 mm and dimple diameter (d_m) 5 mm. The dimples are arranged helically and are spherical in shape. The geometries of the annular diffuser and the dimpled hub are displayed in Fig. 1.

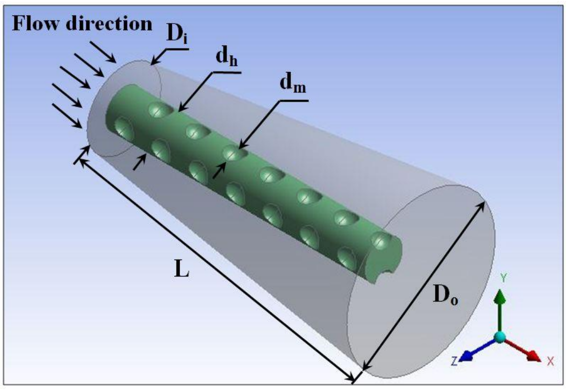

Fig. 1: Annular diffuser geometries

2.2. Boundary Conditions

Numerical investigations were analyzed with air as the working fluid. Assumptions were made to model the velocity profile in an annular diffuser fitted with the dimpled hub. The main assumptions to simulate this problem are:

1. The flow is steady and incompressible.
2. The velocity at the inlet boundary condition is 40 m/s.
3. The pressure at the diffuser outlet is set to Atmospheric.
4. The physical properties of the fluid at the annular diffuser inlet are constant.
5. Inlet Reynolds number is (Re = 2.7×10^4) based on the hydraulic diameter (D_h) as follows, [16]

$$D_h = \frac{4\pi\,(Di^2 - {d_h}^2)/4}{\pi(Di + d_h)} = Di - d_h \tag{1}$$

The physical properties of the working fluid (air) have been considered as the density (1.168 kg/m^3) and the dynamic viscosity (1.848×10^{-5} kg/m.s).

3. Numerical Methods

In this work, the governing equations are solved by using the commercial software ANSYS FLUENT 16.1. The time independent incompressible Navier-Stokes equations are solved using finite volume technique. The numerical analyses are performed in three-dimensional domains applying standard k-ε model as a turbulence model. The k-ε turbulence model is adopted in the current study because it can be efficiently captured in the three-dimensional flow field. Since that, it is assumed to be convenient for this research simulation which is three-dimensional in nature. [17].

3.1. Mesh Generation

In this work, the computational domain for the three tested models was created from the subtraction of the annular diffuser section from the dimpled hub section. Fig. 2 shows three different dimpled hub models which are generated by Auto CAD software.

The grids were generated using the structured grid generator, ANSYS ICEM software. It is structured hexahedral mesh generator software designed to automatically generate meshes in complex 2D and 3D geometries. Fig. 3 displays a schematic of the grid generation for three numerical models, namely, SDH, DDH, and CDH, respectively.

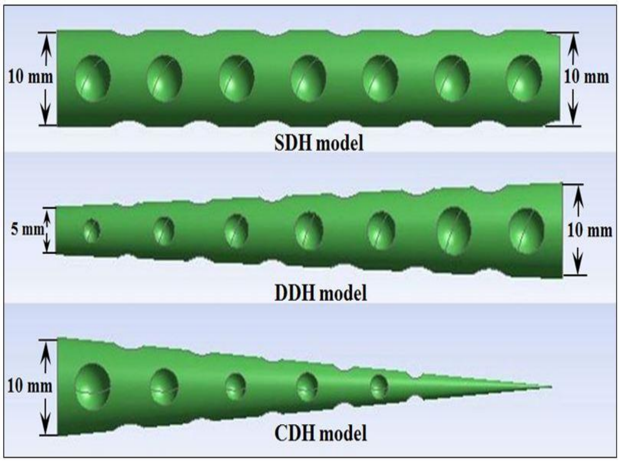

Fig.2: Different dimpled hub models

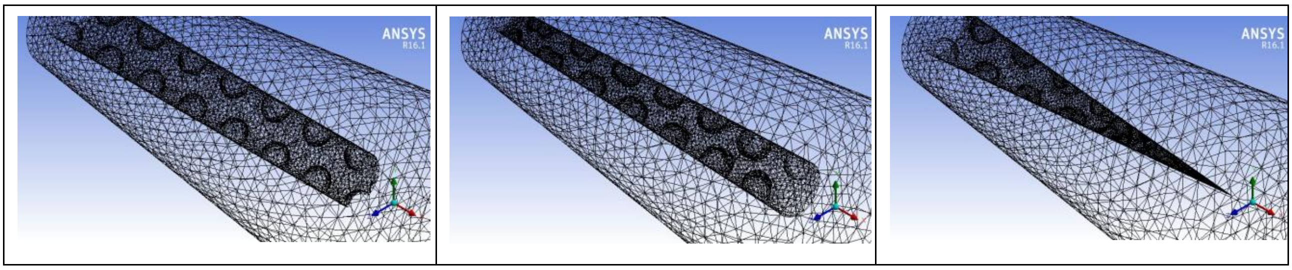

Fig. 3: Schematic view of the grid generation for three numerical models

To evaluate the grid sizes on the numerical results accuracy, three grid sizes were tested. It has been observed that 127636 elements grid is suitable for the simulations.

3.2. Governing Equations of the Computational Model

The governing equations of incompressible flow in the annular diffuser including conservation equations of mass, momentum, and energy are given as below:

- Mass Conservation Equation

$$\frac{\partial}{\partial x_i}(r\rho u_i) = 0 \tag{2}$$

- Momentum Conservation Equation:

$$\frac{\partial}{\partial x}(r\rho u_i u_j) = -r\frac{\partial \mathrm{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[r\left(\mu\frac{\partial u_i}{\partial x_j} - \overline{\rho \acute{u}_i \acute{u}_j}\right)\right] \tag{3}$$

- Energy Conservation Equation:

$$\frac{\partial(\rho\bar{T})}{\partial x_i} + \frac{\partial(\rho\overline{u_i}\bar{T})}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\frac{\lambda}{C_p}\frac{\partial(\bar{T})}{\partial x_i}\right) \tag{4}$$

where *u* is the inlet velocity magnitude, *r* is the inlet radius, ρ air intensity, p inlet pressure, the quantity $-\overline{\rho u_i u_j}$ represents the turbulent Reynolds stresses, λ is the thermal conductivity, and C_p is the specific heat at constant pressure.

3.3. Turbulence Model

The calculations of turbulence kinetic energy *k* and its rate of dissipation ε were obtained from the following transport equations [18],

$$K = 3/2\,(uI)^2 \tag{5}$$

$$I = 0.16\,(Re)^{-1/8} \tag{6}$$

$$\varepsilon = (C\mu^{3/4}.\,K^{3/2})\,l^{-1} \tag{7}$$

$$l = 0.07L \tag{8}$$

where *u* is the inlet velocity magnitude, *I* is the initial turbulence intensity, Re Reynolds number, $C\mu$ is a *k*-ε model parameter whose value is typically given as 0.09, *l* is the turbulence length scale and *L* is a characteristic length. For this study, *L* is considered as the hydraulic diameter.

4. Results and Discussion

The velocity and pressure distribution in an annular diffuser fitted with dimpled hub were simulated for Re = 2.7 × 10^4 with three different dimpled hub models.

4.1. Effect of the Dimpled Hub on the Velocity Profile

In order to display the air recirculation induced by dimples, Figs. 4-7 show the velocity distribution for the three dimpled hubs SDH, DDH, CDH, and a plane diffuser, respectively. The velocity variation can be seen from these Figs. As shown in these Figs., that at a given Re, the dimples provide high-velocity distribution and this due to the increase of the turbulence and the tangential velocity component that will be induced by the dimples.

It can be visualized that the flow around the dimples has a high velocity and forms a high-speed recirculation zone. As the air flow through the dimples wall, it caused impingement effect that can improve the fluid flow mixing and increased turbulent intensity. Moreover, these effects will lead to improve the velocity distribution.

The numerical results showed that utilization of dimples leads to an increase in the velocity distribution up to 10%, 20% and 4% for the SDH, DDH, and CDH, respectively greater than the plane annular diffuser for Reynolds number (Re) 2.7 × 10^4. This enhancement due to the air when it flows into the dimples surface, a strong recirculation will occur on the dimpled wall and because of that, the flow mixing with the streamlines of the air flow through the diffuser enhances the disturbance of velocity.

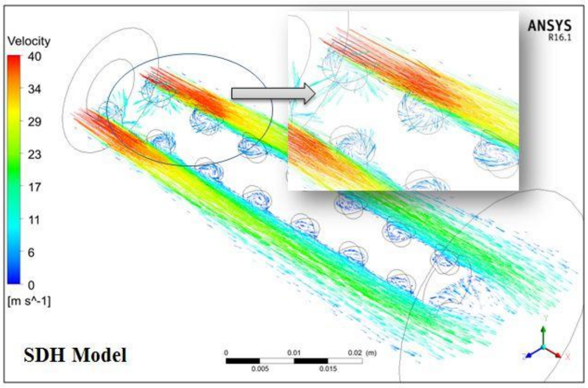

Fig. 4: Velocity distribution vectors for the straight dimple hub (SDH)

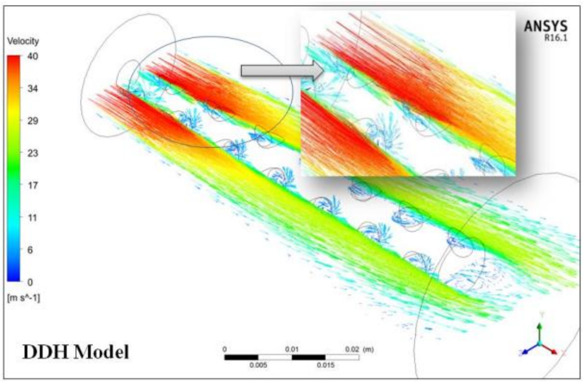

Fig. 5: Velocity distribution vectors for the divergent dimple hub (DDH)

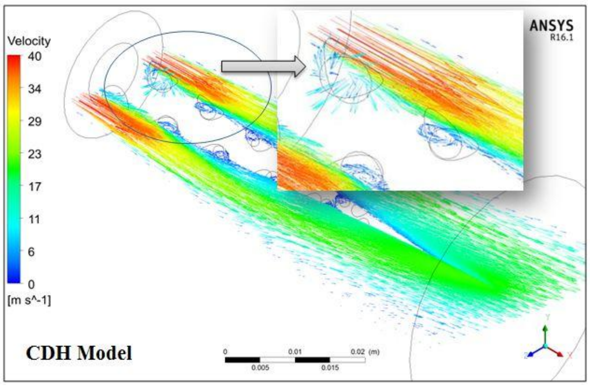

Fig. 6: Velocity distribution vectors for the cone dimple hub (CDH)

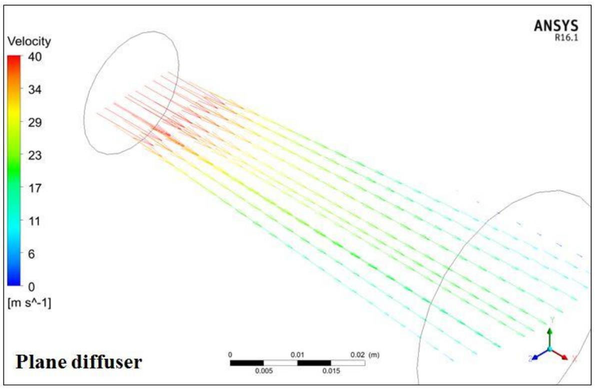

Fig. 7: Velocity distribution vectors for the plane diffuser

Fig. 8 plots a comparison of the axial velocity distribution through the annular diffuser for the three different dimpled hub models SDH, DDH, CDH and the plane diffuser. The Fig. predicts the influence of the dimpled hub on the velocity distribution in a plane positioned at (0.001, 0.0, -0.006) from the diffuser inlet and at (0.08, 0.0, -0.01) from the diffuser outlet. As shown in Fig. 8, the velocity profile for the DDH model is greater than the other models and the plane diffuser. The value of the velocity decreases downstream as this is the aim of the diffuser work.

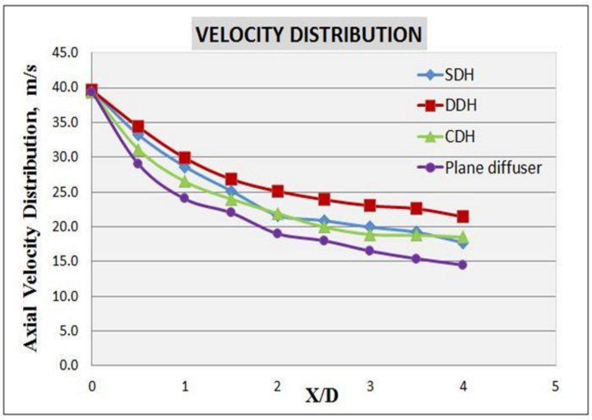

Fig. 8: Comparison of the axial velocity profile for annular diffuser fitted with SDH, DDH, CDH and Plane diffuser

It can be concluded by comparing the velocity variation for the three different dimpled hub models with the plane diffuser that the velocity distribution was increased by using dimples and this enhancement became greater in the case of DDH model.

4.2. Effect of the Dimpled Hub on the Static Pressure Profile

The pressure drop in the annular diffuser along the flow direction induced by dimples is shown in Figs 9-11.

The results are performed for three different dimpled hubs SDH, DDH, CDH, and a plane diffuser.

Fig. 9 shows the effect of SDH on the pressure profile. It is observed that the static pressure in the annulus increases with distance from the inlet. This is because of the theory of the flow through the divergent areas. Fig.10 displays the influence of DDH on the pressure profile. It is detected that the phenomena are the same as SDH. From Fig. 11, it observes that the effect of CDH on the pressure profile is the best compering with the previous models SDH and DDH. Overall, due to the velocity and pressure variation because of the dimples addition, air flow mixing is improved and enhanced.

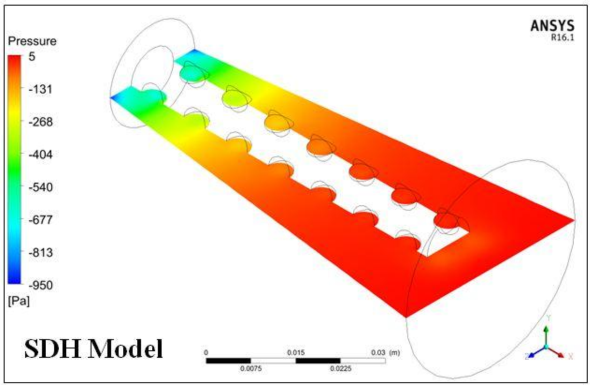

Fig. 9: Static pressure contour for the straight dimple hub (SDH)

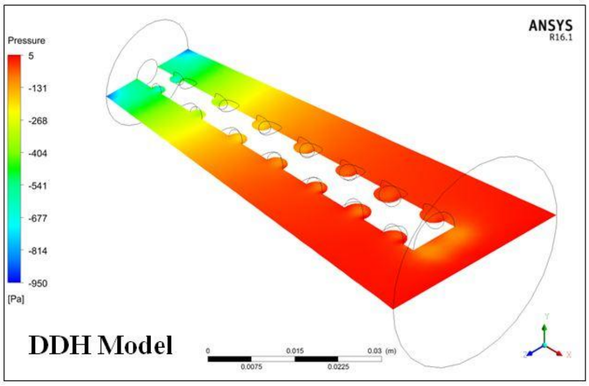

Fig. 10: Static pressure contour for the divergent dimple hub (DDH)

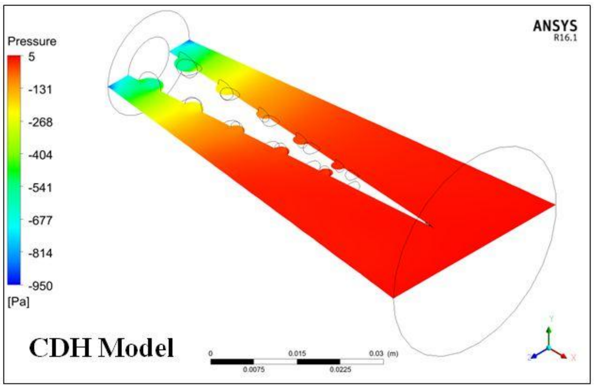

Fig. 11: Static pressure contour for the cone dimple hub (CDH)

Fig. 12 presents the pressure distribution for three different dimpled hub models SDH, DDH, and CDH. It. shows the dimpled hub influence on the pressure distribution in a plane positioned at (0.001, 0.0, -0.006) from the diffuser inlet and at (0.08, 0.0, -0.01) from the diffuser outlet. It can be observed that the static pressure drop through the flow direction of the annular diffuser is bigger when compared with the plane diffuser.

Moreover, the results indicate that the static pressure drop for the CDH model is the best and for the SDH and DDH models are nearly the same.

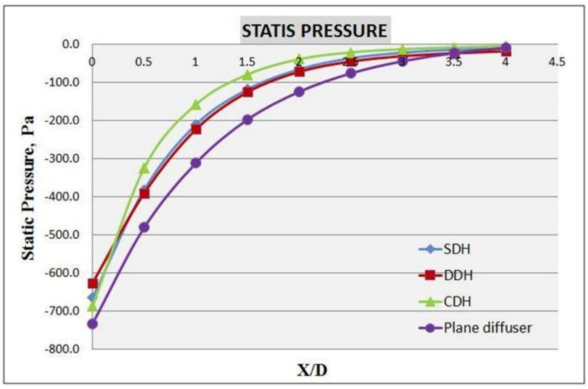

Fig. 12: Comparison of the static pressure profile for annular diffuser fitted with SDH, DDH, CDH and Plane diffuser

5. Conclusion

Numerical simulations analyses were performed for three different dimpled hub models; a straight dimpled hub (SDH), a divergent dimpled hub (DDH), and a cone dimpled hub (CDH). These analyses were carried out to study the velocity distribution and static pressure characteristics. From the numerical findings, the conclusion can be as follows,

1. The results indicated that the insertion of the dimples has a positive effect on both the velocity and pressure distribution.
2. Numerical results showed that adding these types of dimpled hub models enhanced the velocity distribution up to 10%, 20% and 4% for the SDH, DDH, and CDH, respectively compared with the plane diffuser for Reynolds number (Re) 2.7 x 10^4.
3. The velocity distribution enhancement of the DDH models is the best.
4. The static pressure in the annular diffuser with CDH model is the largest followed by SDH and DDH models.
5. The addition of the dimples increased the static pressure compared with the static pressure in the plane diffuser
6. It is clearly seen that the disturbance and mixing induced by the insertion of dimples improve the velocity and pressure distribution.

Acknowledgement

The authors would like to thank the Engineering Technical College-Baghdad, Middle Technical University (MTU), Baghdad, Iraq and the Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM) Malaysia for providing assistance and support for this research.

References

[1] H. A. Abdalla, M. M. El-Mayit, A. A. Abd El-Hamid, and A. M. El-Shazly, “Study of swirling turbulent flow and heat transfer characteristics in conical diffusers,” Proceedings of ICFDP 8: Eighth International Congress of Fluid Dynamics & Propulsion. Egypt ICFDP8-EG-114N, 2006.

[2] S. Bhattacharyya, T. Roy, N. Ghosh, Bandyopadhyay, and P. P. Sarkar, “Convective heat transfer enhancement in low Reynold number of a circular pipe with full length,” International Journal of Mechanical and Production Engineering, vol. 2, no. 9, pp. 6–10, 2014.

[3] P. S. Rao, and K. K. Kumar, “Numerical and experimental investigation of heat transfer augmentation in double pipe heat exchanger with helical and twisted tape inserts,” International Journal of Emerging Technology and Advanced Engineering., vol. 4, no. 9, pp. 180–192, 2014.

[4] R. Prakash, “Experimental investigations of flow through annular diffuser with and without struts,” European Journal of Scientific Research, vol. 52, no. 3, pp. 366–384, 2011.

[5] S. Selvam, P. Thiyagarajan, and S. Suresh, “Experimental studies on effect of bonding the twisted tape with pins to the inner surface of the circular tube,” Thermal Science, vol. 18, no. 4, pp. 1273–1283, 2014.

[6] A. R. Anvari, K. Javaherdeh, M. Emami-Meibodi, and A. M. Rashidi, “Numerical and experimental investigation of heat transfer behavior in a round tube with the special conical ring inserts,” Energy Conversion and Management, vol. 88, pp. 214–217, 2014.

[7] Zheng, P. Liu, F. Shan, Z. Liu, and W. Liu, “Numerical investigations of the thermal-hydraulic performance in a rib-grooved heat exchanger tube based on entropy generation analysis,” Applied Thermal Engineering, vol. 99, pp. 1071–1085, 2016.

[8] F. Sun, Y. Yao, X. Li, G. Li, and Z. Sun, “A numerical model for predicting distributions of pressure and temperature of superheated steam in multi-point injection horizontal wells,” International Journal of Heat and Mass Transfer, vol. 121, pp. 282-289, 2018.

[9] Jerson R.P. Vaz, and David H. Wood, “Effect of the diffuser efficiency on wind turbine performance,” Renewable Energy, vol. 126, pp. 969-977, 2018.

[10] A. Bosioc, C. Tanasa, S. Muntean, and R. F. Susan-Resiga, “Pressure recovery improvement in a conical diffuser with swirling flow using water jet injection,” The Publication House Proceeding of The Romanian Academy, Series A, vol. 11, pp. 245–252, 2010.

[11] Anupriya, and S. Jayanti, “Experimental and modeling studies of gas-liquid vertical annular flow through a diverging section,” International Journal of Multiphase Flow, vol. 67, pp. 180–190, 2014.

[12] L. Zheng, Y. Xie, and D. Zhang, “Numerical investigation on heat transfer performance and flow characteristics in circular tubes with dimpled twisted tapes using Al2O3-water nanofluid,” International Journal of Heat and Mass Transfer, vol. 111, pp. 962-981, 2017.

[13] N. Chimres, C. Wang, and S. Wongwises, “Optimal design of the semi-dimple vortex generator in the fin and tube heat exchanger,” International Journal of Heat and Mass Transfer, vol. 120, pp. 1173-1186, 2018.

[14] S. Wanga, W. Dua, 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.

[15] 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.

[16] 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.

[17] D. Erdemir, S. Gunes, V. Ozceyhan, and N. Altuntop, “Numerical investigation of heat transfer enhancement and pressure drop in heat exchanger tube fitted with dual twisted tape elements”, Recent Researches in Mechanical Engineering, no. ISBN: 978–1–61804–153–1, pp. 167–172, 2013.

[18] 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.