Numerical Simulation of Three Dimensional Turbulent Flow Structure and Heat Transfer in Ribbed-Straight, Divergent and Convergent Ducts

Amin Etminan[1,*], Zambri Harun[2] and Ahmad Sharifian[3]
[1,2]Department of Mechanical and Materials Engineering, Faculty of Engineering and Built Environment, 43600 UKM Bangi, Selangor, Malaysia
[3]School of Mechanical and Electrical Engineering, University of Southern Queensland, Toowoomba QLD 4350, Australia
[*]*corresponding author's e-mail:* etminan.amin@gmail.com

Abstract**—** Turbulent heat transfer and flow structure inside straight, divergent and convergent ducts with square ribs has been investigated for Reynolds number varying from 7000 to 100000 numerically. The simulations have been employed using the one and two turbulent models in particular Re-normalization group (RNG) as the most efficient model. The impact of divergence or convergence on thermal efficiency of ducts during cooling process is explored in two and three dimensional flow regimes. Results reveal that divergent channel transfers higher amount of heat in comparison to other kinds of channels. Furthermore, divergent channel records less pressure loss and higher thermal efficiency where fully developed flow can be seen just through the straight duct. Computational data show a good agreement with experimental results available in the literature.

Keywords—Numerical Simulation; Heat Transfer Enhancement; Square Rib; Vortex Generator; Straight/Convergent/Divergent Ducts.

I. Introduction

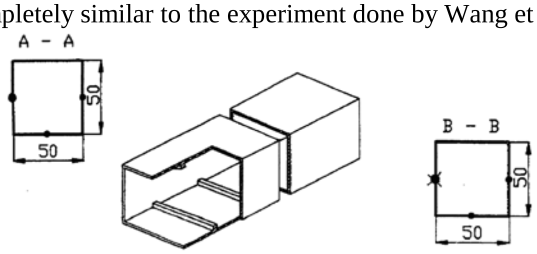
NOWADAYS, a vast variety of vortex generators, fins, swirl chambers, surfaces with arrays of protrusions, dimpled surfaces, and surface roughness have been employed to enhance convective heat transfer coefficient for internal cooling of gas turbine blades. The effects of ribs, angle-of-attack and the channel aspect ratio on the local heat transfer coefficient have been carried out by Hand and Park [1] numerically. They considered a short rectangular channel with a pair of opposite rib-roughened walls for Reynolds numbers varying from 10000 to 60000 and different angle-of-attack of the ribs. Furthermore, Han et al [2, 3] simulated numerically the effect of the rib angle orientation on the local heat transfer coefficient and pressure loss inside a straight ribbed channel for Reynolds numbers varying from 15000 to 90,000. Their results show that the 60° (or 45°) v-shaped rib transfers heat more than the 60° (or 45°) parallel rib and, subsequently, more than the 60° (or 45°) crossed rib and the 90° rib. Additionally, the v-shaped rib generates the highest heat transfer, while the inverted-v-shaped rib produces the greatest pressure loss. Acharya et al [4] investigated periodic fully developed flow and heat transfer for a ribbed duct experimentally and numerically, using the nonlinear and standard k-ε turbulence models. The computed recirculation lengths and highest Nusselt number locations agreed well with the measured values. They found that the nonlinear model predicted more realistic Reynolds stresses in the core flow region immediately above the ribs than the standard k-ε model. But, the local Nusselt numbers were under-predicted by both models. The majority of studies have been focused on the straight ducts meanwhile divergent and convergent channels have become very popular among researchers for the sake of their heat transfer properties only in recent years.

Wang et al [5] have investigated the local heat transfer and pressure drop properties of developing turbulent flows through straight/divergent/convergent channel in three wall-mounted ribs experimentally. They found that in the straight duct, the fluid flow and heat transfer became fully developed after the 2nd-3rd ribs, while in the divergent and convergent ducts there was no such behavior. The comparison showed that among the three ducts, the divergent and convergent ducts caused the highest and lowest heat transfer coefficients respectively, while the straight duct located somewhere in the middle. Ligrani et al. [6] found that vortex generators not only enhanced secondary flows and turbulence levels to increase mixing and form coherent fluid motions in the form of streamwise-direction vortices but also increased three dimensional turbulence parameters by increasing shear and creating gradients of velocity. The overall aim for such internal cooling methods is optimal thermal efficiency with minimal use of coolant fluid and pressure loss through the ducts. The onset and development of the buoyancy driven secondary air flow and enhancement of heat transfer in a horizontal convergent and a divergent channel have been carried out by Liu and Gau [7] experimentally. The onset of secondary flow appearing as transverse instability wave and onset of initial protrusion of the bottom heated layer are identified. However, the deceleration flow in the divergent channel and the acceleration in the convergent make the mean Nusselt numbers approach the results of the parallel-plate channel.

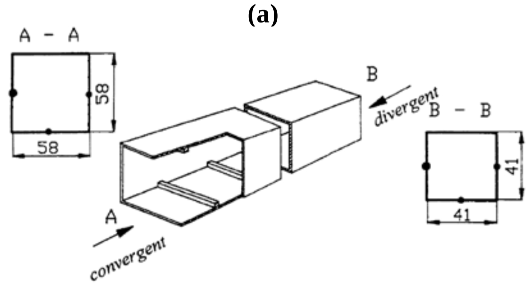
Wang et al [8] studied an experiment involving PIV measurements through a channel with wall-mounted ribs on one side. They found. the maximum Reynolds shear stresses occured at the leading edge of the rib. Besides, the effects of combined ribs and winglet type vortex generators on forced convection heat transfer and friction loss behaviors for turbulent airflow through a constant heat flux channel have been investigated by Promvonge et al [9] experimentally. They delved the larger angle of attack led to higher heat transfer and friction loss meanwhile the in-line rib yielded the highest growth in both the Nusselt number and the friction factor but the rib with staggered array performed higher heat transfer in comparison to the other types of arrangements.

Labbé [10] carried out large eddy simulations in a ribbed channel with a blockage ratio of 30%. According to the channel height and the bulk velocity the Reynolds number was 40000. All simulations reproduced the major flow structures measured experimentally. The found that the secondary cross-sectional flows have a large influence on the heat transfer augmentation on the solid walls of channel.

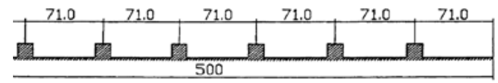
In the present study, the finite volume method has been employed to simulate Navier-Stokes equations which is coupled with the energy equation. Two and three dimensional turbulent convective heat transfer and flow pattern inside straight, divergent and convergent ducts with square ribs has been carried out for Reynolds number varying from 7000 to 100000. The impact of divergence or convergence on thermal performance of ducts during cooling process is delved in details. In all simulations, the cross section area of ribs is 4*4 mm^2 and they are mounted on the walls with 71 mm space between them. All geometrical details are inserted into Figure 1 completely similar to the experiment done by Wang et al [5].



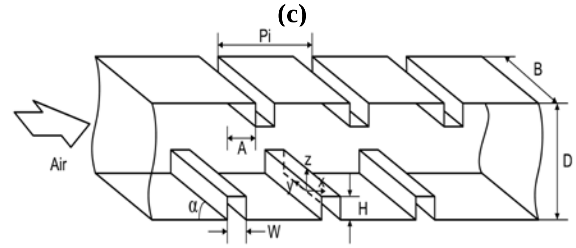
(a)



(b)



(c)



(d)

Fig. 1. Geometrical details in millimeter; (a) Straight duct (b) Convergent/divergent duct (c) The side view of ribs and duct (d) The dimensional parameters of computational domain (Sections A-A and B-B are considered at the left and right of the channel respectively.)

II. Numerical procedure

Generating the computational field has been established by GAMBIT software in two and three dimensional cases. Structured grid with quadratic map as style of meshing has been utilized to produce the computational domain. A sample of generated two dimensional grid is observable in Figure 2 where quadric-structured and non-structured hexagonal grids have been employed for two and three dimensional domains respectively.

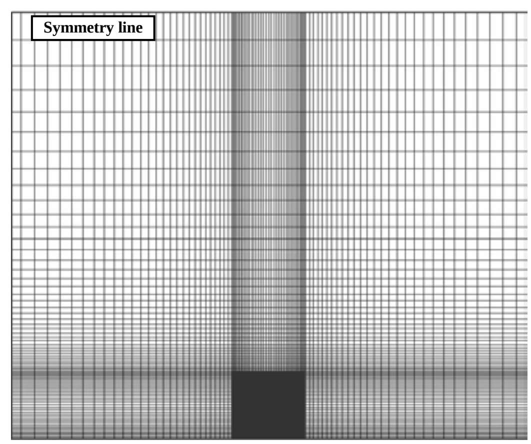


Fig. 2 Sample of 2D generated mesh around a rib

For all of the geometries the operative fluid is air (Pr = 0.7), the Reynolds number with the basis of mean velocity (Um) and mediocre hydraulic diameter of the passage is 7000 to 100000. In all of the processes the flow has been considered turbulent, steady and uncompressible; moreover, the thermo-physical fluid properties such as density, Prandtl number, thermal conductivity coefficient, viscosity, and etc. have been generally averaged in the fluid field along the duct, for having only small changes in the most of mentioned properties in this kind of study. For expediting the convergence of solution and damping the rate of turbulences, in discretization of momentum, energy, and transport equations, the first order upwind model and second order central difference has been employed for separating the convection and diffusion terms in Navier-Stokes equations respectively. For interpolating the pressure of momentum equation in cell's faces, for coupling the pressure and velocity in continuity equation, SIMPLE algorithm has been used.

With regards to the vast domain of Reynolds number, Reynolds Averaged Navier-Stokes (RANS) turbulent models have been employed for all numerical simulations. Therefore, time-averaged Navier-Stokes, energy equation and transport equations for turbulence parameters are considered as the governing equations for this problem.

There are three different approach to simulate flow structure adjacent to the solid walls: 1. Standard Wall Function (SWF) 2. None Equilibrium Wall Function (NEWF) 3. Enhanced Wall Treatment (EWT) where EWT method has been selected for flow analysis. This numerical method not just reduces computational time but also decreases sensitivity to the grid size [1].

III. Numerical Results

For all numerical simulations several turbulent models can be employed involving; Spallart-Allmaras (S-A), shear stress transport (SST) extracted from k-ε model and realizable model (RM) and Re-normalized group (RNG) extracted from k-ω model and Reynolds stress model (RSM). Figure 3 depicts local convective heat transfer coefficient for different turbulent models inside a straight duct. As it can be seen RSM, RNG, RM, SST and S-A turbulent models estimate heat transfer pattern efficiently where the RNG is used as the most practical and time-consumer model.

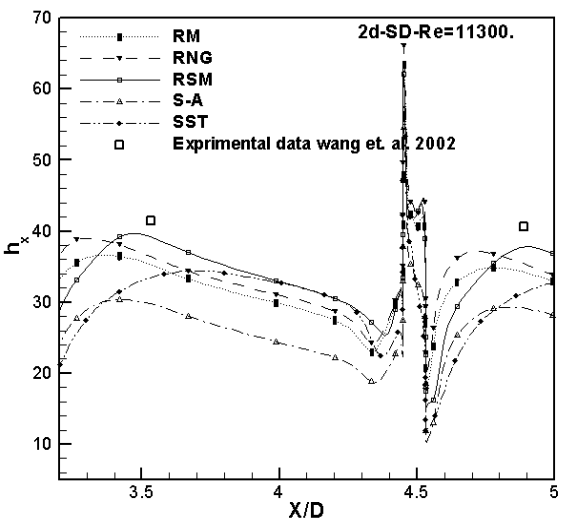


Fig. 3 Local convective heat transfer coefficient for different turbulent models inside straight duct

The no-slip condition, neglecting surface roughness versus rib height, and constant heat flux has been regarded for wall boundaries. Due to the symmetric nature of the flow structure and heat transfer, the boundary condition opposite of the ribbed face has been designed to be symmetrical which has decreased the computational expenses (See Figure 1). Uniform temperature, 300 K and velocity (velocity varies to determine the value of Reynolds number) has been employed at inlet of the duct; moreover, for assuming the intensity of turbulence of fluid flow at inlet, the following equation has been utilized [8];

$$I \equiv \frac{u'}{u_{avg}} = 0.16(\mathrm{Re}_{DH})^{-1/8} \tag{1}$$

At the outlet, fully-developed flow has been applied as the boundary condition. It means that all flow parameters gradient would be equal by zero as $\partial\varphi/\partial x=0$ where φ should be replaced with u, v, w, T and p.

The local heat transfer coefficient has calculated from the heat flux which has been inserted on the walls, the local wall temperature and local bulk mean air temperature:

$$h_x = \frac{Q''}{(T_{wx} - T_{bx})} \tag{2}$$

The local wall temperature used in equation (2) is the results of computational attempts. The local bulk mean temperature of air is calculated by the succeeding equation as follow:

$$T_{bx} = T_{in} + \frac{Q'' A(x)}{\dot{m} C_p} \tag{3}$$

Figure 4 illustrates the total convective heat transfer coefficient along the center line of straight duct for Reynolds number 11300. As it can be seen the low-Reynolds RNG turbulent model is able to anticipate numerical results efficiently in comparison with the experimental data of [5]. The local heat transfer coefficient through the channel reaches a constant level after 3rd rib completely similar to the thermal fully developed conditions. Furthermore, the highest value of heat transfer coefficient is moved to the upstream location in comparison to the experimental details.

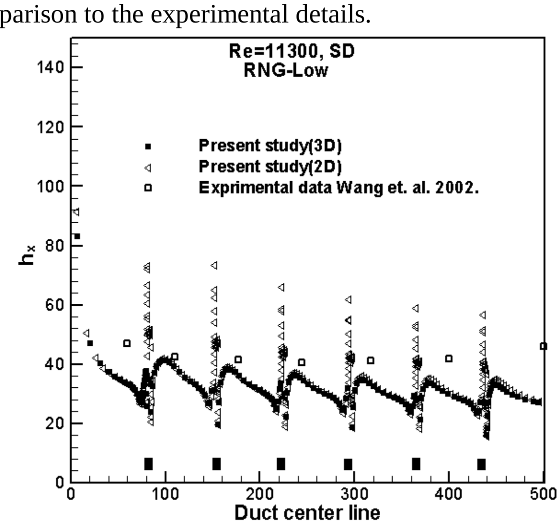


Fig. 4 Two and three dimensional total convective heat transfer coefficient along the center line of straight duct

On the other hand, local Nusselt numbers through the ducts are plotted in Figures 5-7. According to these figures the most significant points can be summarized as follow:

1) Nusselt number rises versus Reynolds number meanwhile the rate of Nusselt number decreases as well. Furthermore, there is no remarkable difference between 2D and 3D simulations in straight duct whilst for other ducts considerable

deviations are found. For instance, 16% and 22% differences are computed between 2D and 3D flow simulations respectively through the divergent duct.

2) 3D simulation anticipates Nusselt number closer to the experimental results for the non-straight ducts meanwhile 2D shows more competency for straight duct in particular when Reynolds number is very high.

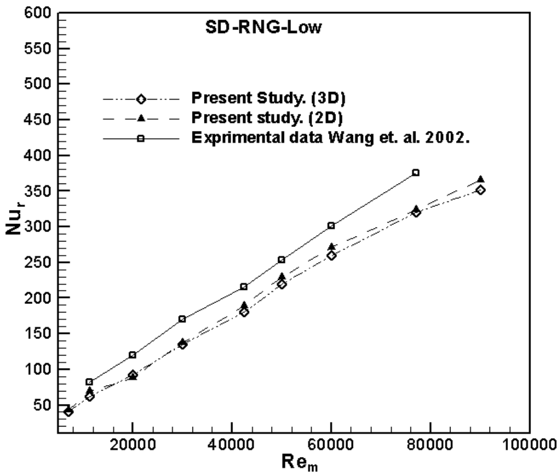


Fig. 5 Total Nusselt number versus Reynolds number inside the straight duct

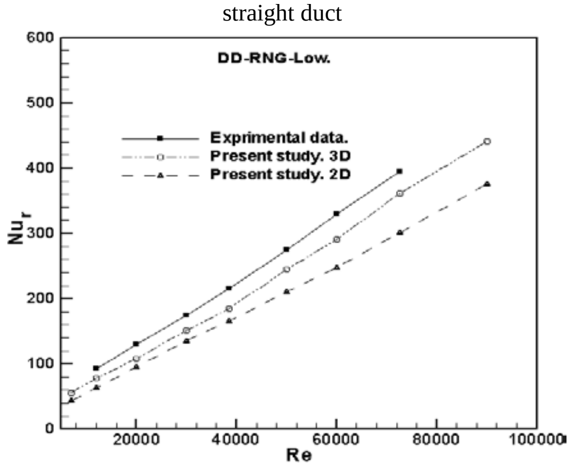


Fig. 6 Total Nusselt number versus Reynolds number inside the divergent duct

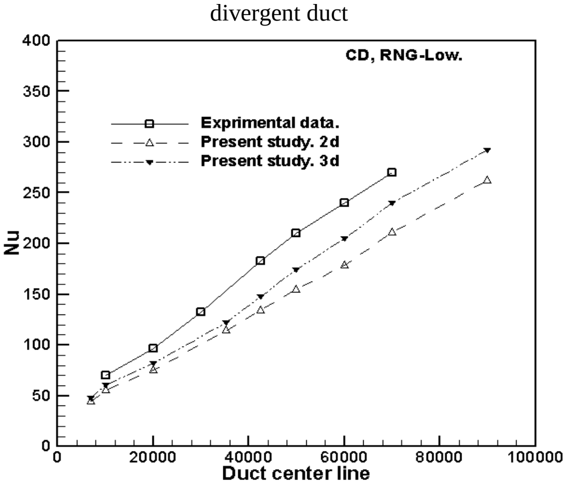


Fig. 7 Total Nusselt number versus Reynolds number inside the convergent duct

Turbulent viscosity ratio and turbulent intensity contours of three dimensional flow for Reynolds number 20000 on symmetry plans xz and xy are illustrated in Figures 8 and 9 respectively. The shown turbulent parameters can be considered as the two index of turbulence in flow. As it can be seen the highest magnitude of turbulence is related to divergent duct where the local Reynolds number is higher in comparison with other ducts.

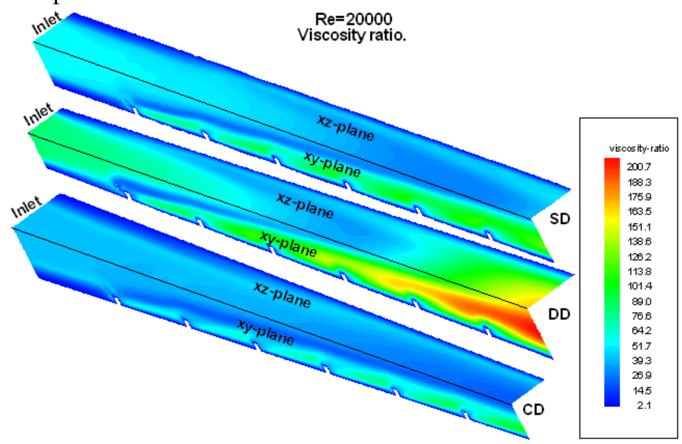


Fig. 8 Turbulent viscosity ratio contours of three dimensional simulation of flow for Reynolds number 20000 on symmetry plans xz and xy

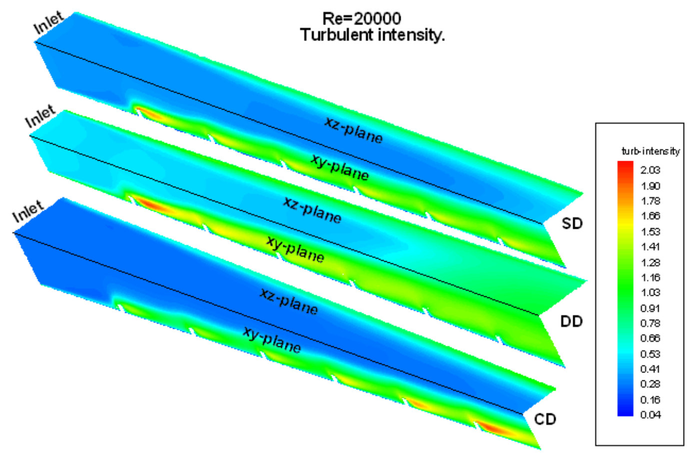


Fig. 9 Turbulent intensity contours of three dimensional simulation of flow for Reynolds number 20000 on symmetry plans xz and xy

Instantaneous streamlines around the 3rd rib in three dimensional simulation of flow for Reynolds number 20000 on symmetry plan xy for straight, divergent and convergent ducts and the location of reattachment point of them has been drawn. As it can seen the longest and shortest recirculation length are related to the divergent and convergent ducts respectively. Adverse pressure gradient in divergent duct causes the momentum of fluid particles moves downstream and therefore streamlines attach to solid wall far away from the rib accordingly in comparison to other ducts.

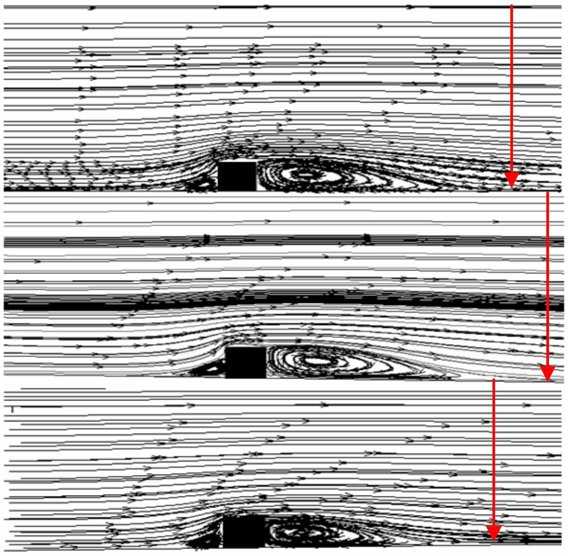


Fig. 10 Instantaneous streamlines around the third rib in three dimensional simulation of flow for Reynolds number 20000 on symmetry plan xy for straight, divergent and convergent ducts and the location of reattachment point of them from top to bottom respectively.

Thermal efficiency enhancement of straight, divergent and convergent ribbed-ducts versus Reynolds number in comparison with smooth-straight duct has been illustrated in Figure 11. As it be can seen divergent duct shows more improvement than others and reaches a constant value of 0.78 for Reynolds number between 70000 and 100000. On the contrary, convergent duct has lowest thermal efficiency.

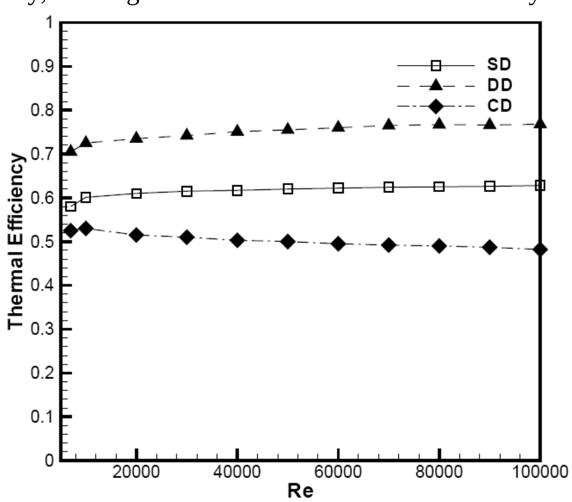


Fig. 11 Thermal efficiency enhancement of straight, divergent and convergent ribbed-ducts versus Reynolds number in comparison with smooth-straight duct

IV. Conclusion

Present study investigates the three dimensional turbulent flow and heat transfer in ribbed-straight, divergent and convergent ducts for a vast range of Reynolds number varying from 7000 to 100000 numerically. One and two turbulent models have been employed during numerical simulation meanwhile each one has several merits. As it can be observed, in order to reach more accuracy and reduce calculation time, Re-normalization group (RNG) turbulent models have been employed for all simulations. Furthermore, flow analysis reveals that the flow fluctuations and secondary flow inside divergent duct is more than other geometries where the recirculation length behind the ribs is longer than others as well. It should be noted that results also show less pressure loss, more heat transfer as well as higher thermal efficiency in comparison to other ducts. For instance, about 18% rise is found for thermal efficiency of divergent duct in comparison with straight duct. Last but not least, some newer as well as efficant turbulence models including LES and DNS can be used for two-phase operational nanofluid such as Al2O3. It seems that nanofluid enhances heat transfer coefficient inside the channels as well.

Acknowledgment

This research is supported by Universiti Kebangsaan Malaysia (UKM) under grant Fundamental Research Grant Scheme code FRGS/1/2013/TK01/UKM/03/1.

Nomenclature

English Letter

A	The staggered distance of two opposite ribs
$A(x)$	The cross section area at position of x
B	The dimension of duct along y-axis
C_p	Heat capacity at constant pressure
D	The dimension (height) of duct along z-axis
D_m	Hydraulic diameter of channel
h_x	Local convective heat transfer coefficient
H	The height of rib
I	The intensity of turbulence
$\dot{m}$	Mass flow rate
n	Normal direction
Nu	Nusselt number
P	Static pressure
Pi	Rib pitch
Pr	Prandtl number
Q''	Heat flux
Re_{DH}	Reynolds number based on hydraulic diameter
T	Temperature
T_{bx}	Bulk mean temperature at position of x
T_{in}	The inlet flow temperature
T_{wx}	The local wall temperature at position of x
u	Velocity component along x-direction
u'	The fluctuation of x-velocity component
u_{ave}	Mean x-velocity component on inlet plane
U_{in}	Inlet velocity
U_m	Mean x-velocity component
U_{out}	Outlet velocity
v	Velocity component along y-direction
w	Velocity component along w-direction
W	The width of rib
x	Coordinate axis along the stream wise
y	Coordinate axis along the height of channel
z	Coordinate axis along the width of channel

Greek Letter

α	The angle of attack of ribs
ε	Turbulent dissipation
ρ	Fluid density
ν_t	Turbulent viscosity
ν_m	Molecular viscosity

φ Symbol of flow parameters
ω Dissipation per turbulent energy

REFERENCES

[1] J. C. Han and J. S. Park, "Developing Heat Transfer in Rectangular Channels with Rib Turbulators," International Journal of Heat and Mass Transfer, Vol. 31, No. 1, pp. 183-195, 1988.
[2] J. C. Han, Y. M. Zhang and C. P. Lee, "Augmented Heat Transfer in Square Channels with Parallel, Crossed, and V–Shaped Angled Ribs," Journal of Heat and Mass Transfer, Vol. 113, No. 3, pp. 590-596, 1991.
[3] J. C. Han, and Y. M. Zhang, "High Performance Heat Transfer Ducts with Parallel Broken and V-Shaped Broken Ribs," International Journal of Heat and Mass Transfer, Vol. 35, No. 2, pp. 513–523, 1992.
[4] S. Acharya, S. Dutta, T.A. Myrum and R.S. Baker,"Periodically developed flow and heat transfer in a ribbed duct," International Journal of Heat and Mass Transfer, Vol. 36, No. 8, pp. 2069-2082, 1993.
[5] L. B. Wang, W. Q. Tao, Q. W. Wang and T. T. Wong, "Experimental Study of Developing Turbulent Flow and Heat Transfer in Ribbed Convergent/ Divergent Square Ducts," International Journal of Heat and Fluid Flow , Vol. 22, No. 6, pp. 603–613, 2001.
[6] P. M. Ligrani, M. M. Oliveira, and T. Blaskovich, "Comparison of heat transfer augmentation techniques," AIAA Journal, vol. 41, No. 3, pp. 337–362, 2003.
[7] C. W. Liu and C. Gau, "Onset of Secondary Flow and Enhancement of Heat Transfer in Horizontal Convergent and Divergent Channels Heated from Below," International Journal of Heat and Mass Transfer, Vol. 47, No. 25, pp. 5427-5438, 2004.
[8] L. Wang, M. Salewski and B. Sundén, "Turbulent flow in a ribbed channel: Flow structures in the vicinity of a rib," Experimental Thermal and Fluid Science, Vol. 34, No. 2, pp. 165–176, 2010.
[9] P. Promvonge, T. Chompookham, S. Kwankaomeng and C. Thianpong, "Enhanced heat transfer in a triangular ribbed channel with longitudinal vortex generators," Energy Conversion and Management, Vol. 51, No. 6, pp. 1242–1249, 2010.
[10] O. Labbé, "Large-eddy-simulation of flow and heat transfer in a ribbed duct," Computers & Fluids, Vol. 76, pp. 23–32, 2013.

My full name is **Amin Etminan** and I was born on 1st September 1982 in Shiraz, Iran. I received my B.Eng and MSc. degrees from Persian Gulf University and Yazd University on Fluid Mechanics/Energy Conversion in 2004 and 2007 respectively. I started my academic career as an official lecturer in the Department of Mechanical Engineering, Islamic Azad University in 2007 coincident with the membership of Iranian Engineers organization as a licensee member. My major interests are Turbulent Flow, Nanofluid, Blood Flow Simulation, Heat Transfer Enhancement and Internal Cooling of Gas Turbine Blades. I have taken part in many regional, national and international conferences held in Spain, Singapore, Thailand, India and China where my presentation in India has been introduced as a nice presentation.

Besides, I introduced as top researcher of my university in 2011, 2012 and 2013. Furthermore, I have a scientific collaboration with Dr Zambri Harun and Dr Ahmad Sharifian from National University of Malaysia (UKM) and the University of Southern Queensland (USQ), Australia respectively and we have published three peer reviewed journal papers and one conference paper until now.