Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 00 (2019) 000–000

Energy Procedia

www.elsevier.com/locate/procedia

2[nd] International Conference on Energy and Power, ICEP2018, 13-15 December 2018, Sydney, Australia.

Numerical analysis of heat transfer and flow dynamics in a pipe with square extruded bluff cylinder inserts

Suvanjan Bhattacharyya[a,b], Arnab Banerjee[b], Md. Anisur Rahman[b], Rishika Saha[c], Akshoy Ranjan Paul[d*]

[a]*Department of Mechanical and Aeronautical Engineering, University of Pretoria, Hatfield, Pretoria, South Africa.*
[b]*Department of Mechanical Engineering, MCKV Institute of Engineering, Liluah, Howrah - 711204.West Bengal, India.*
[c]*Mechanical Engineering Department, Dr. Sudhir Chandra Sur Degree Engineering College, Kolkata, India.*
[d]*Department of Applied Mechanics, Motilal Nehru National Institute of Technology, Allahabad - 211004, Uttar Pradesh. India.*

Abstract

This paper presents the computational study of convective heat transfer enhancement and entropy generation performance by using bluff cylinder. A two-dimensional numerical simulation is executed successfully on thermo-hydraulic transport characteristics for a square extruded bluff cylinder placed inside duct. The test fluid, air (Pr 0.707), is used for the experiment. The turbulent SST scheme is engaged for the simulation. The simulation is accomplished with the aim to achieve an understanding of physical performance and activities of the thermo-fluidic flow inside the channel fitted with bluff body at 15D spacing from the inlet under constant temperature conditions at wall. The simulation is done to cover cross flow encompassing both laminar as well as turbulent flow regime in the range of Reynolds number (Re) up to 1×10^5 and inlet intensities from 5% to 40%. The effects of outlet temperature and Nusselt number is esteemed and considered. No ideal Re has been detected which could reduce the total entropy generation. It is highly promising to be executed in the practical relevance like power and energy industry.

Keywords: Heat transfer; Enhancement; Entropy generation; Turbulent flow; Turbulent intensity

* Corresponding author. Tel.: +91-532-2271208; fax: +91-532-2545341.
E-mail address: arpaul@mnnit.ac.in

1. Introduction

Bluff cylinders are widely studied for flow around them due to their theoretical and practical use in applied science & having scientific relevancy in thermal science and in fluids. The flow characteristics in fluids are classed as external and internal flow, depending upon whether fluid flows over the top surface or in the channel.

Nomenclature

c	clearance, m
D	diameter of the channel, m
D_h	hydraulic diameter, m
d	diameter of extruded portion, m
L	channel length, m
Pr	Prandtl number
Re	Reynolds number
S	Entropy, $J.K^{-1}$
T	temperature, K
V	bulk velocity, $m.s^{-1}$
u_i	velocity, $m.s^{-1}$
xi	space coordinates, m
Subscripts	
avg	average
0	plain channel
b	fluid bulk quantity
w	wall
gen	generation

For designing heat exchangers to be used in locomotives, cooling systems (nuclear plants), buildings, chimneys, power lines, screens, cables, etc. cylinder-like structures and swirl generators are used [1] alone as well as in groups for both air and water being used as the flowing medium. In other terms, the characteristics of flow around cylinders show various crucial physical phenomenon, like detachment of flow [2], casting off of the vortex and also the shifting of flow from being laminar to being turbulent. The special attributes of the bodies are detrimental to the way of flow and the energy transformation and flow characteristics. Therefore, it is imperative for the designer to formulate the correct shape and maintain proper dimensioning of the body. Any diversion from correct shape or dimension may loss the efficiency of flow or heat exchange process. In order to realize the flow and the thermal behavior, many researchers put their efforts and few important works are reported here.

Motlagh and Hashemabadi [3] performed 2D and 3D CFD modeling of transfer of heat from separate round cylindrical corpuscles in four different cases. The result of turbulence intensity (TI) for free stream on the transfer of heat and mass surface of orbital cylinder and oval cylinder was shown by Kondjoyan and Daudin [4] in low Reynolds number. The heat transfer (local and average) was carried out with forced convection by Sanitjai and Goldstein [5] for a round cylinder. The work also confirms that the coefficient of local transfer is more prominent at the point of stagnation of a round cylinder than at the surface. Scholten et al. [6] has studied the velocity vector profile and heat exchange of a cylinder in transverse flow for low freestream turbulence. An empirical relation is developed for the three regions of the cylinder. You and Kwon [7] have investigated of various turbulence models for the simulation of flow about a round cylindrical bluff at a critical Reynolds number.

From the literature, it is found that the special effects of inlet instability on the transport phenomenon over bluff cylinders are not reported widely. Aim of this present numerical investigation is on the transport phenomenon in square extruded bluff cylinder (SEBC). Effects of TI on 'square extruded bluff cylinder' have not been performed by any researcher till date. The present numerical assessment procures a standard investigation methodology of the simulations for heat exchange from square extruded bluff cylinder and validated with the established correlations for permitting further assessment of heat transmission on more complex geometrical profiles.

2. Problem formulation and grid generation

Ambreen et al. [2] presented an idea and performed 2D modeling on flow and heat exchanging phenomenon over a modified square bluff cylinder geometry. The present computational work studies the transport characteristics about a square extruded bluff cylinder (SEBC). The non-dimensional distance between the bluff body and the channel inlet is 15D, where the total length of the duct is 30D as presented in Fig. 1, where 'D' is the diameter of the channel and c is the clearance space. The problem is considered to be two-dimensional. Fig. 1 also shows the bluff cylinder specification. Air is employed as working fluid for which the 'Pr' is 0.707.

The computational studies were executed employing unstructured grids as exemplified in Fig. 2. For confirming adequately low discretization errors, a study was accomplished to validate the grid independence. The outcomes of which are presented in Table 1, for the case with TI 5% at Re = 20,000.

Table 1. The case with TI = 5% and Re=20,000: Grid independence study

	Total number of grid nodes	Nu
Grid 1	658,196	64.125
Grid 2	722,568	64.816
Grid 3	947,364	64.854

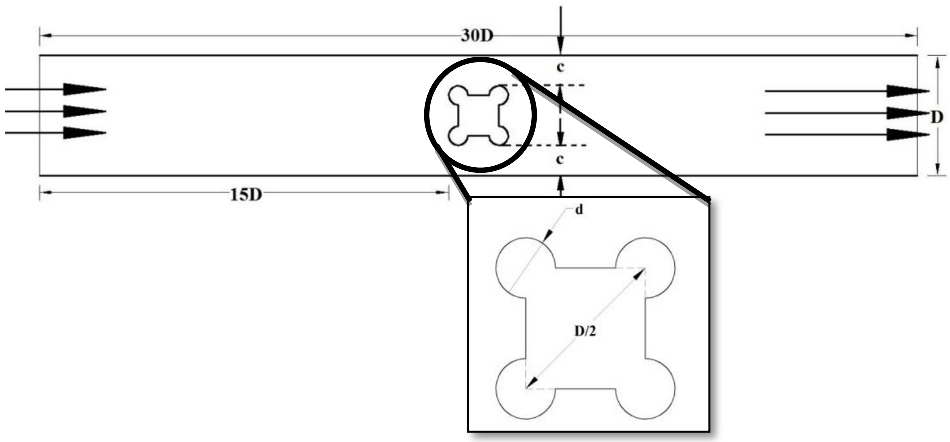

Fig. 1. Computational Domain

One can monitor in Table 1, the variations in the key factors are rather negligible among the grids (3 grid) used. Thus, the Grid 2 is decided as a best choice between the precision and computational economy.

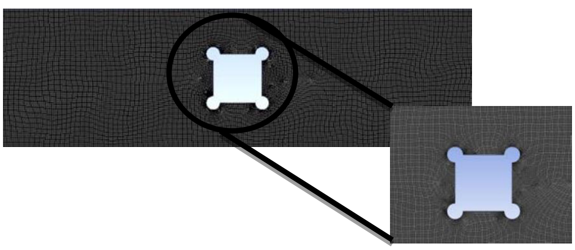

Fig. 2.Detail views of grid

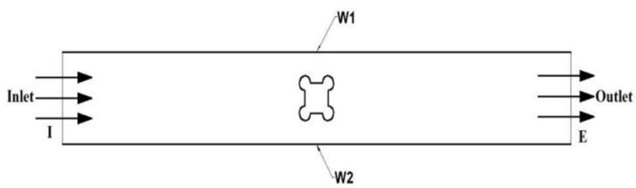

Fig. 3. Computational Domain showing boundary

3. Boundary conditions and governing equations

The working fluid (i.e. air) approaches to SEBC with inlet velocity of U_∞ and temperature of the inlet fluid is T_∞. The working fluid for the present computation is air consisting constant physical characteristics. The working fluid inlet temperature is 300 K. In the Fig. 3 the boundary condition of region I is consider to be velocity inlet. The region W_1 and W_2 is consider to be top and bottom wall which is kept at constant temperature 500 K and region E is consider as pressure outlet. The simulation is done to cover cross flow encompassing both laminar as well as turbulent flow regime in the range of Reynolds number up to 1,00,000 and inlet intensities from 5% - 40%.

The governing equations (momentum, energy and continuity) are executed using SST turbulence model which was given by Abraham et al. [8] and Menter et al. [9].ANSYS 15.0 software is used to compute the turbulent equations by following Bhattacharyya et al. [10] and Kadiyala et al. [11].The governing equations were solved employing finite volume based CFD software ANSYS-Fluent 15.0. The momentum equation was discretized with second order accurate upwinding order for achieving desired level of accuracy and the energy equation was solved using third-order MUSCL. 10^{-4}, 10^{-5}, and 10^{-7} are the different convergence criteria set for continuity, momentum and energy equations respectively, along with 10^{-4} set for other turbulence quantities.

4. Results and Discussion

The results that found after computational investigation by using square extruded bluff cylinder (SEBC) are discussed in this section. The computations are performed for Re ranging from 100 to 100000. The deviation of average Nu with the raise of Re at particular TI is studied and the rise in heat transfer coefficient with the rise in TI is reported. The turbulence intensity varies from 5% to 40%.

As a preliminary step, a plain channel (PC) without SEBC is computed numerically. The outcomes are matched with the proposed correlations by Dittus–Boelter [12] for Nu, in the higher Re region (turbulent regime) (Re $\geq$2300). In the low Re region (laminar regime) (Re $\leq$ 2300) the analytical relations established for the laminar flow are employed for the comparison, i.e. Nu=3.66.

Fig. 4 shows variation of heat transfer for a fixed value of TI of 5%.The computation is executed for Re of 100 to 100000 and the figure shows decent agreements with the correlation stated above.The prediction of heat transfer rate is in well matched with the observational and experimental correlation with a quantifiable misplay of not more than 5%.

The influence of ‘TI’ on heat transfer enhancement had been studied by Bhattacharyya et al. [13], who reported the dependency of heat transfer on ‘TI’ for a lower range of ‘Re’. In the current computational investigation the raise in ‘Nu’ has been recorded with increase in ‘Re’ for the ‘TI’ of 5%, 10%, 15%, 20%, 25%, 30%, 35% and 40%.

The deviation of ‘Nu’ with ‘Re’ for different ‘TI’ is presented in Fig. 5. From the figure one can see that for the lower value of ‘Re’ the ‘TI’ is not much increasing the heat transfer rate but when the value of ‘Re’ is more than 10000 there is higher increasing rate in the coefficient of heat transfer. The value of ‘Nu’ is more or less same up to ‘Re’ = 5000, but beyond this value of Re, heat transfer (HT) rises prominently with increasing ‘TI’. For example, at Re= 20000 Nu increases from 61 to 111 for TI of 5% and 40% respectively. However at Re = 100000, heat transfer rise from 208 to 361 when TI changes from 5% to 40%. Table 2 demonstrates the HT data. It can be seen from the table that Nu (average, Nu_{avg}) increases by about 11% in the channel when a SEBC is present.

The change of flow regime from laminar region to turbulence region can be seen from Fig. 6. Transition SST model can forecast the entire flow regime involving laminar turbulent and transition regime [14-18]. The results from solely laminar and those from turbulent model meet well with the results from transition SST model. In the laminar region the rate of increase in Nu for SEBC and square bluff cylinder (SBC) is slow where as in the turbulence region this rate of increase is very fast and high. From the figure it is clear that the transition zone is around 3500 - 4000.The variation of ‘Nu’ with ‘Re’ with and without SEBC is shown in Fig. 7. From the figure, one can see that HT with SEBC is compared with the case of a PC at Re = 10000. It can also be seen from the figure that HT increases nearly after the position of the SEBC. At the outlet of the duct, the asymptotic ‘Nu’ value is greater when the SEBC is present.

To catch the HT distributions at different Re in presence of SEBC, the variation of local Nu along the channel length is significant. Fig. 8 displays the deviation of local Nu along the duct length for the value of Re100 to 100000 with TI of 5% at the inlet of the channel. One can distinguish from the figure that the patterns are comparable for all the Re values. However the degree of HT increases at increased of Re. At Reynolds number 100000, 80000 and 50000, first and second peaks are higher in magnitude compared to the other tested Reynolds number in presence of SEBC.

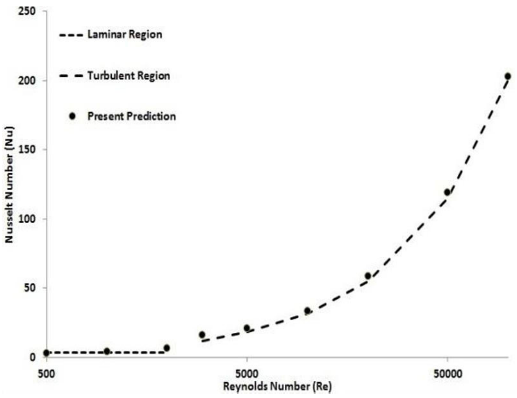

Fig. 4. Plain Channel: Nusselt Number

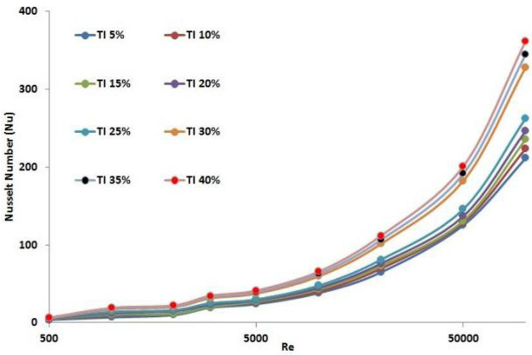

Fig. 5. Effect of turbulence intensity on Nusselt number

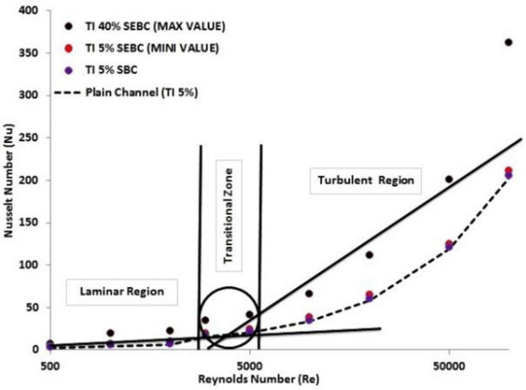

Fig. 6. Prediction of Transition zone

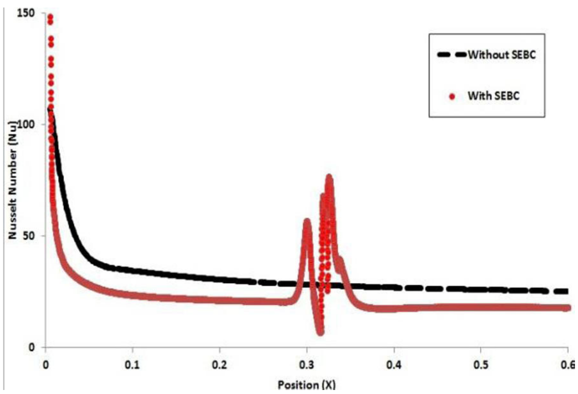

Fig. 7. Distribution of Nu at Re = 10 000.

The consequence of using SEBC on HT rate was significant for all the Re. In general, the SEBC produced swirl flow or secondary flow as shown in Fig. 9(a) contributing a longer flowing trail of fluid flow through the channel; intensive mixing of fluid and pressure gradient might be generated towards the radial direction. One can see from the figure that the flow separates in two streams as it strikes SEBC and combines after the SEBC. However, two vortices (counter rotating air) are seen at the position immediately next to SEBC, it is expected that the vorticity level should be higher in presence of a SEBC as compared to the case without SEBC. The contour plot of TKE is presented in Fig. 9(b). It can be distinguished from the contour that SEBC creates a high turbulent field as the value of TKE is high which almost 20%.

Intermittency (γ) at the outlet of the channel is presented in Fig. 10. As the Re increases the value of the intermittency is touch the value 1. The intermittent performance varies considerably relating to swirl strength at further downstream locations. The phase of transition will depend on γ which varies from 0 to 1. Near 0 value of γ concludes the regime as laminar, whereas the value near 1 concludes the regime as turbulent. From the figure it can be distinguished that γ values moves more towards the radial direction with increased swirl number as an outcome of the high centrifugal force. Fig. 11 shows the important understanding of total entropy generation (S_{gen})for different

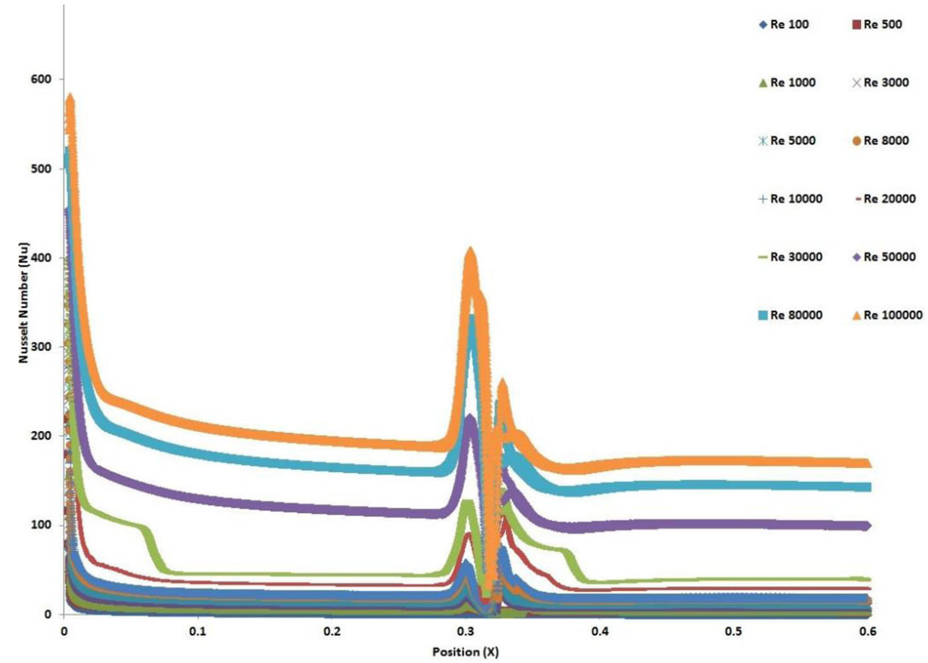

Fig. 8. Distribution of Nu along the channel wall.

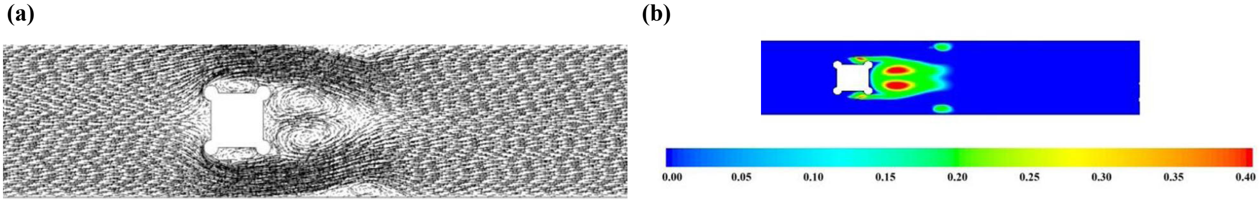

Fig. 9.(a) Velocity vectors around the SEBC, and (b) Contours of turbulent kinetic energy (TKE) at Re = 10 000.

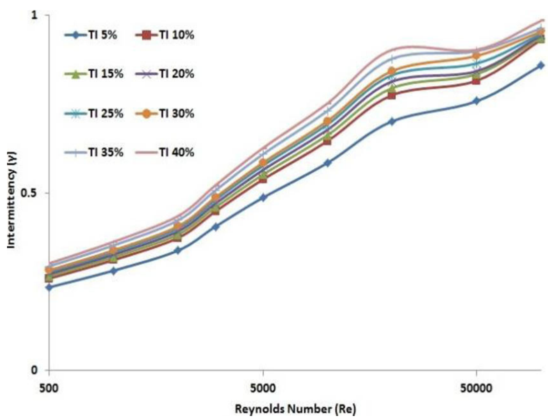

Fig. 10. Variation of intermittency with Reynolds number for different turbulent intensity

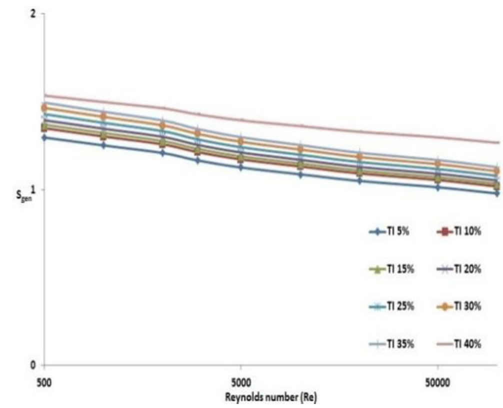

Fig. 11. Variation of total entropy generation with Reynolds number for different turbulent intensity

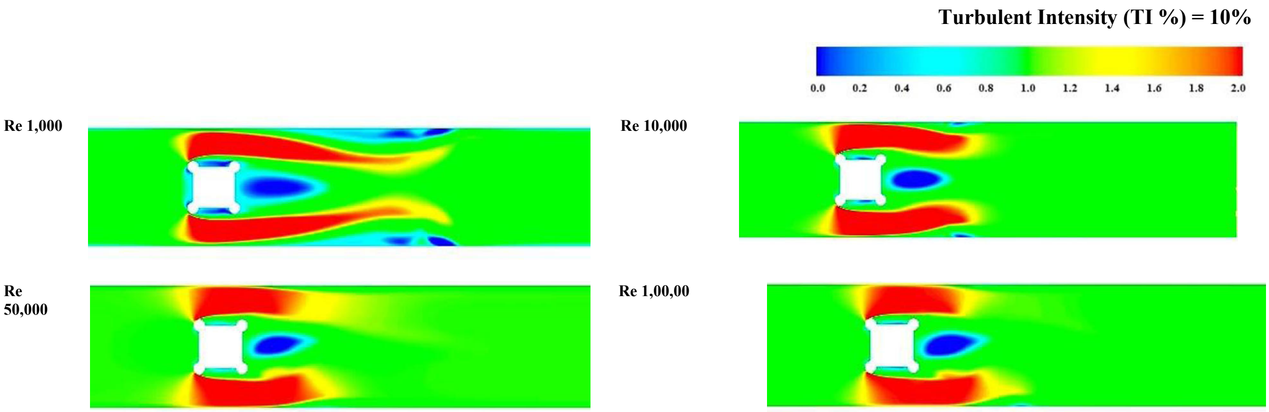

Fig. 12. Dimensionless contours of axial velocity at different Reynolds number

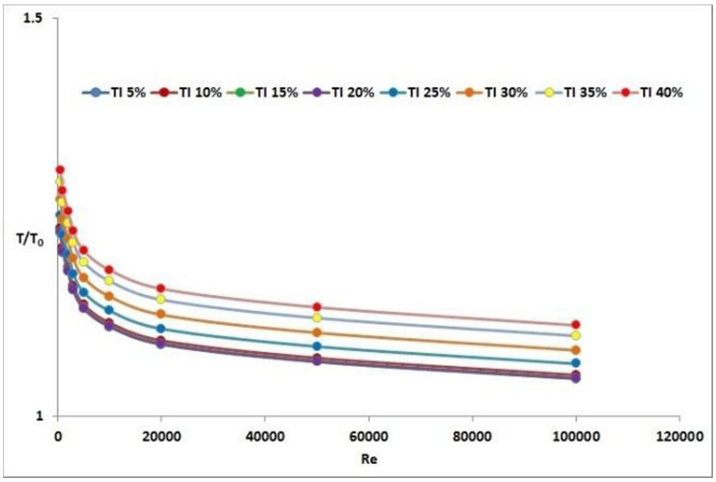

Fig. 13.Variation of dimensionless T/T0 (at exit of the channel) with Reynolds number at different turbulent intensity

Temperatures were gauged at the exit of the channel. This result after non-dimentionlization (T/T0) has been shown in Fig. 13. In all cases (T/T0) is above unity. The dimensionless thermal interaction [((T-TIN) / (TW-TIN))] [18] are shown in Fig. 14.

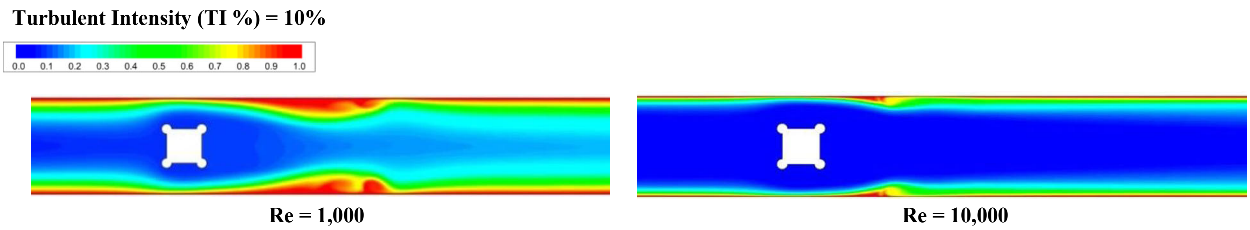

Fig. 14. Non-dimensional temperature contours at Re=1000 and Re= 10,000

TI at inlet following Ratts et al. [19]. It can be witnessed that S_{gen} decreases as Re increases with the drop of TI percentage. This is due to the augmentation of average Nu. Moreover, the quick fall in S_{gen} means that influence of friction S_{gen} is very small and the end product of thermal S_{gen} becomes more significant. Fig. 12 shows the important understanding of dimensionless velocity [((u = U/ u*))] [18] at five stages of Re with different TI at inlet. It can be witnessed that TI has solid impact over the flow field as higher TI leads to reduced flow strength. By seeing the contour plots, it can be understood that the flow fields are more or less comparable at the stages of Re.

5. Concluding remarks

Heat transfer enrichment in a duct due to the occurrence of a square extruded square cylinder (SEBC) is investigated numerically. It is seen that 'Re' has an important effect on area weighted average of 'Nu'. Heat transfer increases as 'Re' increases. 'TI' has an influence on local Nu, with the rise in 'TI' local 'Nu' increases. The order of augmentation is about 10-12%. Transition SST model can predict the entire flow regime involving laminar turbulent and transition regime. 3D flow domain in case of a SEBC was not attained in the current study and is intended to be investigated in future.

References

[1] Tong JCK, Sparrow EM, Minkowycz WJ, Abraham JP. A new archive of heat transfer coefficients from square and chamfered cylinders at angles of attack in crossflow. Int. J. Therm. Sci. 105 (2016) 218–223.
[2] Ambreen T, Kim MH. Flow and heat transfer characteristics over a square cylinder with corner modifications. International Journal of Heat and Mass Transfer. 117 (2018) 50–57.
[3] Ahmadi MAH, Hashemabadi SH. CFD based evaluation of heat transfer coefficient from cylindrical particles. International Communications in Heat and Mass Transfer, 12(35), (2008) 674–680.
[4] Kondjoyan A, Daudin JD., Effects of free stream turbulence intensity on heat and mass transfers at the surface of a circular cylinder and an elliptical cylinder, axis ratio 4.Int. J. Heat Mass Transfer, 10(38), (1995) 1735 1749.
[5] Sanitjai S, Goldstein RJ. Forced convection heat transfer from a circular cylinder in cross flow to air and liquids. International Journal of Heat and Mass Transfer, 12(47), (2004) 4795–4805.
[6] Scholeten JW, Murray DB. Unsteady heat transfer and velocity of a cylinder in cross flow- I. Low free stream turbulence. Int. J. Heat Mass Transfer, 10(41), (1998) 1139-1148.
[7] You JY, Kwon OJ. Numerical assessment of turbulent models at a critical regime on unstructured meshes. Journal of Mechanical Science and Technology, 26 (5), (2012) 1363~1369.
[8] Abraham JP, Sparrow EM, Tong JCK. Heat Transfer in All Pipe Flow Regimes: Laminar, Transitional/Intermittent, and Turbulent. International Journal of Heat and Mass Transfer, 52(3-4), (2009), 557–563.
[9] Menter F, Esch T, Kubacki S. Transition Modelling Based On Local Variables. Proc. Fifth International Symposium on Engineering Turbulence Modelling and Measurements, Mallorca, Spain, Elsevier, Oxford, (2002) 555-564.
[10] Bhattacharyya S, Chattopadhyay H, Guin A, Benim AC. Investigation of inclined turbulators for heat transfer enhancement in a solar air heater. Heat Transfer Engineering (2018), Online Published.DOI: 10.1080/01457632.2018.1474593.
[11] Kadiyala PK, Chattopadhyay H. Numerical analysis of heat transfer from a moving surface due to impingement of slot jets.Heat Transfer Engineering, doi: 10.1080/01457632.2017.1288045, (2017), 1-9.
[12] Ozisik MN. Heat Transfer: A Basic Appoach. Mc-Graw Hill, PA, USA, (1985).
[13]Bhattacharyya S, Das S, Sarkar A, Guin A, Mullick A. Numerical simulation of flow and heat transfer around hexagonal cylinder. International Journal of Heat and Technology, 35(2), (2017) 360-363.
[14] Bhattacharyya S, Chattopadhyay H, Benim AC. 3D CFD Simulation On Heat Transfer Enhancement In Turbulent Channel Flow With Twisted Tape Inserts. Progress in Computational Fluid Dynamics. An International Journal, 17(3), (2017) 193-197.
[15] Bhattacharyya S, Chattopadhyay H, Bandhopadhyay S. Numerical Study on Heat Transfer Enhancement through a Circular Duct Fitted With Centre-Trimmed Twisted Tape. International Journal of heat and Technology, 34 (3), (2016) 401-406.
[16] Bhattacharyya S, Chattopadhyay H, Benim AC. Heat Transfer Enhancement Of Laminar Flow Of Ethylene Glycol Through A Square Channel Fitted With Angular Cut Wavy Strip. Procedia Engineering, 157, (2016) 19 – 28.
[17] Bhattacharyya S, Khan AI, Maity DK, Pradhan S. Hydrodynamics and Heat Transfer of Turbulent Flow Around a Rhombus Cylinder. Chemical Engineering Transactions, 62, (2017) 373-378.
[18] Bhattacharyya S, Chattopadhyay H, Haldar A. Design of Twisted Tape Turbulator at Different Entrance Angle for Heat Transfer Enhancement in a Solar Heater. Beni-Suef University Journal of Basic and Applied Sciences, 7 (1), (2018)118-126.
[19] Ratts EB, Raut AG. Entropy Generation Minimization of Fully Developed Internal Flow with Constant Heat Flux. Journal of Heat Transfer, 126, (2004) 656-659.