Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 160 (2019) 285–292

Energy Procedia

www.elsevier.com/locate/procedia

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

Computational study on thermal energy around diamond shaped cylinder at varying inlet turbulent intensity

Suvanjan Bhattacharyya[a], Kunal Dey[b], Rajdeep Hore[c], Arnab Banerjee[b], 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, Howrh, West Bengal, India.*
[c]*Department of Mechanical Engineering, Dr. Sudhir Chandra Sur Degree Engineering College, Kolkata, India.*
[d]*Department of Applied Mechanics, Motilal Nehru National Institute of Technology Allahabad, Prayagraj. India.*

Abstract

Transport phenomena over a diamond shaped cylinder (DC) at varying inlet turbulent intensity is reported in the work. An infinite diamond cylinder bluff body with hydraulic diameter D is investigated numerically. The turbulent SST model is engaged for simulation. The simulation is accomplished with the aim to achieve an understanding of physical behavior of thermo-hydraulic flow over diamond cylinder at 15D spacing from the inlet. The computational analysis is done to cover cross flow encompassing laminar, transition, quasi turbulent and turbulent flow regime in the range of Reynolds number upto 1,00,000 and inlet intensities from 5% to 20%. The computational fluid dynamics results are confirmed with correlations which show reasonable agreements. Moreover, the diamond cylinder provides better performance than circular cylinder. The effects of Nusselt number are esteemed and the proposed work computes the influence of inlet turbulence intensity on augmenting heat transfer from the diamond cylinder.

Keywords: Bluff body; Heat transfer; Enhancement; Transitional flow;Quasi-turbulent; Turbulent intensity

* Corresponding author. Tel.: +91-532-2271208; fax: +91-532-2545341.
E-mail address: arpaul2k@gmail.com

10.1016/j.egypro.2019.02.155

1. Introduction

Flow phenomena around cylinders are widely studied due to their theoretical and practical use in applied science & having scientific relevance in thermal science and in fluids. On the engineering application, there are numerous devices in mechanical, civil and other engineering where bluff bodies are encountered. Cylinder-like constructions are found in both alone and in a grouping arrangement, in the designing for heat exchangers for cooling system for chimneys, locomotive, off-shore structures, buildings, nuclear power stations. Power cables, lines, screens, and grids both in air and water as the flowing medium. In particular tenures, the flow around circular cylinders shows various significant physical behaviors such as, partitioning of flow, shedding of vortex and also the transition of flow from laminar to turbulence.

Nomenclature

D	diameter of the diamond shaped cylinder, m
h	heat transfer coefficient, $W.m^{-2}.K^{-1}$
k	fluid thermal conductivity, $W.m^{-1}.K^{-1}$
L	channel length, m
p	pressure, Pa
Re	Reynolds number
T	temperature, K
V	bulk velocity, $m.s^{-1}$
Subscripts	
avg	average
0	plain channel
turb	turbulent
b	fluid bulk quantity
w	wall

Thompson et al. [1] used a spectral element technique (higher order) to simulate and numerically investigate a wake structure created for fluid flow over a circular cylinder for both 2D and 3D models. The predicted results are verified with accurate experimental results and to reduce the errors within experimental uncertainty limit for the Strouhal number and base pressure coefficient. It is obtained that the energy transfer ratio between the 3D and 2D model is 4. Finite-volume method is utilized by Majumdar et al. [2] for computing incompressible flows in complex three-dimensional situations with curved asymmetrical boundaries. The method works on non-orthogonal structured grids, Cartesian velocity components and cell-cantered variable arrangement. Coutanceau and Bouard [3] used a visualization technique to find out the flow phenomena of viscous fluid passed over a dynamic circular cylinder having uniform speed in perpendicular direction with its generating lines in flow domain. Numerical and experimental results are compared, and explanations are provided about the proposed calculation methods. On other hand, the effect of fluid flow over cylinders combined with convective transfer of heat from bluff to air is experimentally assessed by Oosthuizen et al. [4]. Farouk et al. [5] proposed mixed natural heat convection from a heated cylinder through flowing fluid in surround using numerical modelling. Results are reported exposing with isotherms, streamlines and heat transfer coefficient. This investigation performed with the variation of wall thickness. Chang and Sa [6] reported the effect of buoyancy on wake formation behind a heated cylinder kept in both natural and forced convection regime. The results are drawn on basis of drag coefficient curves, flow patterns and Nusselt number curves. Singh et al. [7] investigated the effect of buoyancy on the results of circular cylinder as heat source/sink in a flow field using a novel finite distribution method with CFD modelling. Saha [8] carried out a numerical assessment of free convection through a cylinder kept at the centre of a fluid domain using MAC method. To obtain better results fourth order spatial as well as temporal discretization was employed. Mixed convectional heat exchanging property for a row of cylinders kept in fluid domain was numerically investigated by Gowda et al. [9]. The heat exchange and flow phenomena of fluid flow over a row of cylinders were observed and reported by Gowda et al. [10]. Euler's explicit algorithm was employed to get results. It was observed that the effect of buoyancy and blockage exists in the Nusselt number. Lacroix and Carrier [11] presented the heat exchanging phenomenon of two horizontal cylinders having vertical separation within adiabatic walls. The results were reported in the basis of heat transfer coefficient varies with separation variation and Nusselt number variation. Wu and Hung [12] analysed the impact of fluid flow around three heated circular cylinders arranged in isosceles right angle triangular from. The vortex shedding effect of the cylinders also observed. Gandikota et al. [13] carried out a study where the effect of thermal buoyancy was observed on upward flow and phenomenal heat transfer of circular cylinder placed in fluid domain. Bhattacharyya et al. [14-16] reported way of energy savings by using vortex generators. Sumner et al. [17] used flow visualization method to investigate the flow phenomena of fluid flow over two and three circular cylinders kept in side by side arrangement. For Re 100 and 200 Farrant et al. [18] applied the cell boundary method to evaluate 2D Navier-Stokes equation for flow around the cylindrical obstacles. The vortex shedding effect was observed and report.

The influences of inlet TI on convective HT and fluid flow over bluff diamond cylinders are not reported widely by the previous researchers. Henceforth studies of inlet disturbances in the form of TI (5% and 20%) are investigated in this present numerical work covering laminar, transition and turbulent flow regimes.

2. Problem formulation and grid generation

The present computational work studies the transport characteristics about a diamond cylinder.The non-dimensional distance between the diamond cylinder and the channel inlet is 15D, where the total length of the flow domain is 50D and inlet diameter is 10D as presented in Fig. 1, where D is the hydraulic diameter of the diamond shaped cylinder. The problem is considered to be two-dimensional. Air is employed as working fluid for which the ‘Pr’ is 0.707.

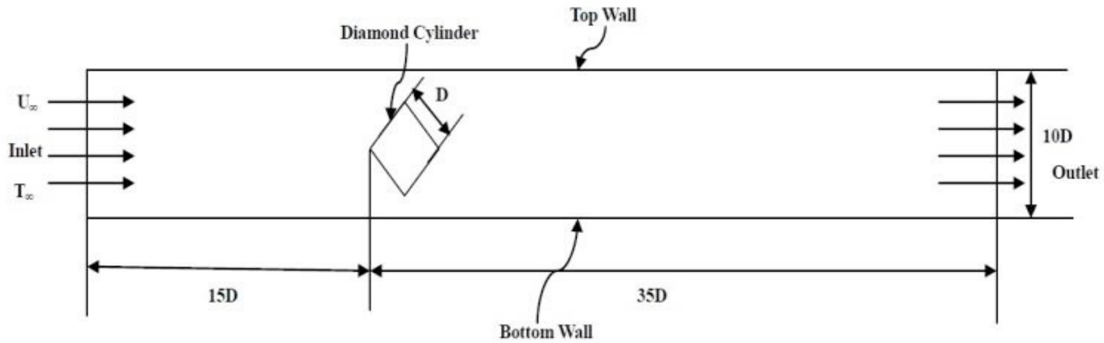

Fig. 1. Computational Domain

The computational studies were executed employing structured 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.

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

3. Boundary conditions and governing equations

Fluid approaches to the plate with inlet velocity of $U_\infty$and temperature of the inlet fluid is T_∞ where the cylinder constant surface temperature is maintained about T_s =398k. Air (Pr=0.71) is defined as the working fluid for the given domain, inlet temperature is kept constant at a value of 298K. The upper & lower walls are assumed symmetrical where the first derivative vanishes. The domain size is fixed and the fluid properties are also considered to be constant thus the variation of Reynolds number are done by changing the inlet velocity.

The governing equations were solved employing commercial CFD software ANSYS FLUENT 15.0. The momentum equation was discretized with second order accurate upwinding scheme for achieving desired level of accuracy and the energy equation was solved using third-order MUSCL. The convergence criteria for continuity, momentum and energy equations were set at 10^{-4}, 10^{-5}, and 10^{-7} respectively. The convergence criterion for the four turbulence quantities was set to 10^{-4}. The gradient for intermittency (γ), turbulent kinetic energy (TKE), specific dissipation rate, and momentum thickness was done using second order accurate upwind scheme.

4. Results and discussions

The results obtained after performing the simulations are discussed below. Reynolds number (Re) ranging between 100-1,00,000 is considered for the study. For given turbulence intensity, Nusselt number(Nu) variation is studied with the increase of Re. Correlation for the increase in coefficient of heat transfer with the increase in turbulence intensity is drawn. Variation in turbulence intensity for the present numerical study is considered from 5% to 20%. The assessment of the present computational outcomes with the published results is shown in Table 2. A decent agreement with the literature can be seen in the table.

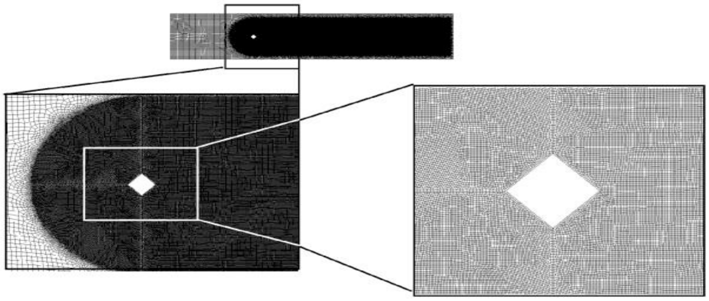

Fig. 2.Detail views of grid

Table 1.Grid independence study (for the case with TI = 5% and Re=20,000).

	Total number of grid nodes	Nu
Grid 1	778,999	96.73249
Grid 2	775,752	96.77249
Grid 3	777,336	96.77649

Zukauskas [19] proposed a correlation (equation 1) for the external flow for both circular bluff and non-circular bluff cylinders. In this present study the confirmation of SST model analysis results is compared with correlation (equation 1) for HT coefficient calculation over the circular cylinders.

$$\overline{Nu}_D = 0.25 + \left(0.4Re^{\frac{1}{2}} + 0.06Re^{2/3}\right) Pr^{0.37} * \left(\frac{\mu}{\mu_w}\right)^{\frac{1}{4}} \quad (1)$$

Table 2. Validation of the present computational study with previous study: Re = 100, and Pr = 0.707

	Present Work	Dhiman et al. [20]	De et al. [21]
Nu_{avg} ➡	**5.593**	**5.584**	**5.670**

The variation of Nu for a value of TI of 5% at inlet as shown in Fig. 3.The CFD simulation is performed for Re of 100 to 100,000 and the results indicates nice agreements with the correlation mentioned above.

In this computational study for the TI level at inlet of 5% - 20%, the rise in Nu has been noted with increase in Re. Well, one can see from Fig. 4 that for lower values of Re, the TI is not affecting considerably the HT rate but once the value of Re is greater than 10,000 there is prominent increase in HT coefficient.

The flow pattern in presence of DC could be definitely distinguished by observing at the velocity vector plots at Fig. 5. At this point the velocity field around the DC is presented. One can see from the figure that the fluid flow splits itself in 2 (two) streams as it strikes the DC and meets again after the DC. Nevertheless, that the counter spinning pair (two) of swirls are noticeably detected at the position straightaway following the DC. Same kind of swirls is also noticeable for other such types of swirl generators such as ribs, tapes, etc. [22-29]. The contours plot of turbulent kinetic energy (TKE) is shown in Fig. 6. The DC is establishing to generate an extremely high turbulent flow filed as replicated in the large values of TKE around 20%. Two sectors of extremely large turbulence correspond to the two swirls observed in Fig. 6.

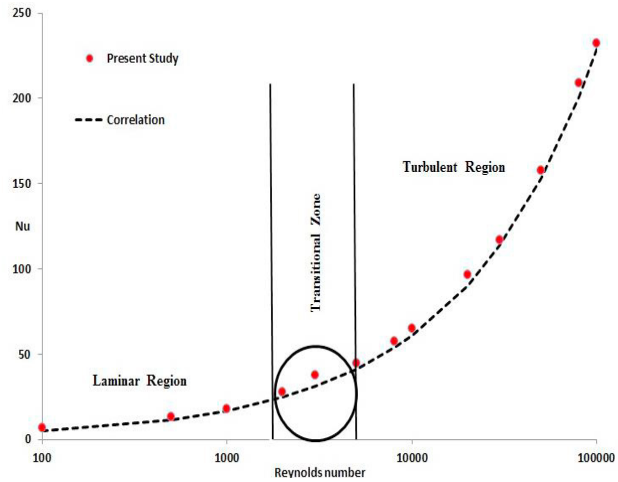

Fig. 3.Validation with correlation

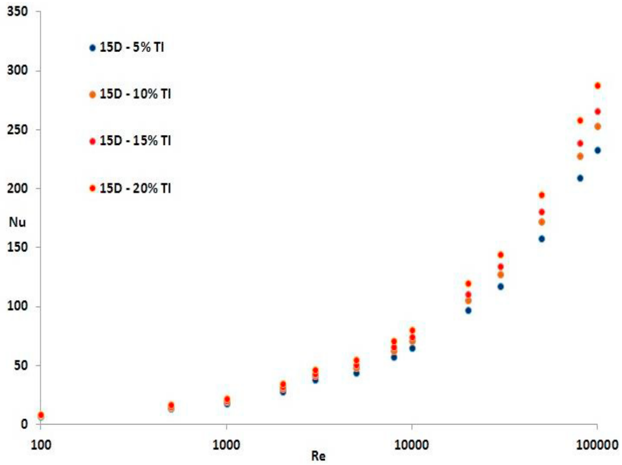

Fig. 4.Influence of TI on Nusselt number (Nu)

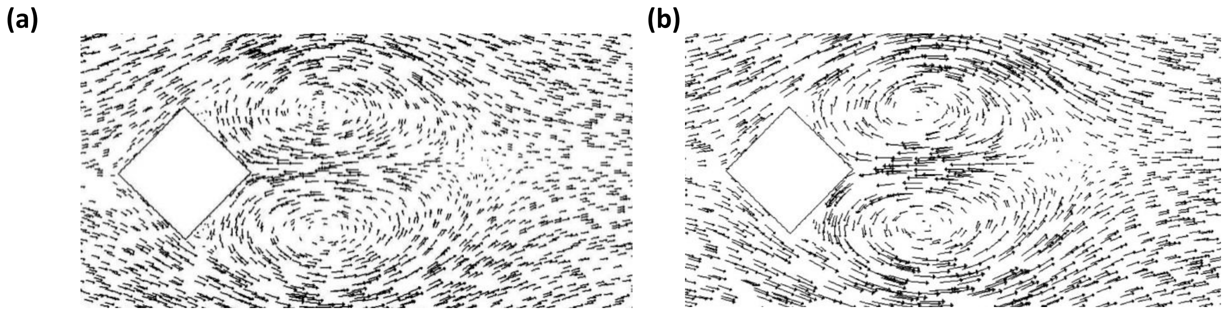

Fig. 5.Velocity vectors around the DC

Non-dimensional velocity contours with different turbulent intensity at Reynolds number 10,000 are expressed in Fig. 7. On refining the TI from 5% to 20%, variation in the velocity profiles is observed in the figure. Also, one can see from the figure that, 5% and 10% TI yield high transverse velocity as compared to 10% and 20% TI. Moreover, 20% inlet TI yields a bit different velocity profile than other tested inlet TI. Temperatures contours are very important to understand the thermal profile. The dimensionless thermal interaction [$((T-T_{IN}) / (T_W-T_{IN}))$] [30] are shown in Fig. 8.

The streamlines plots are shown in Fig. 9. For DC, the plots reflect that the flow coming from inlet is re-joined after it passes the DC, and it happens in every case. At higher inlet turbulent intensity, broader recirculation zone meets disturbance as an effect of back fluid flow thus diminishing the flow reattachment phenomena.

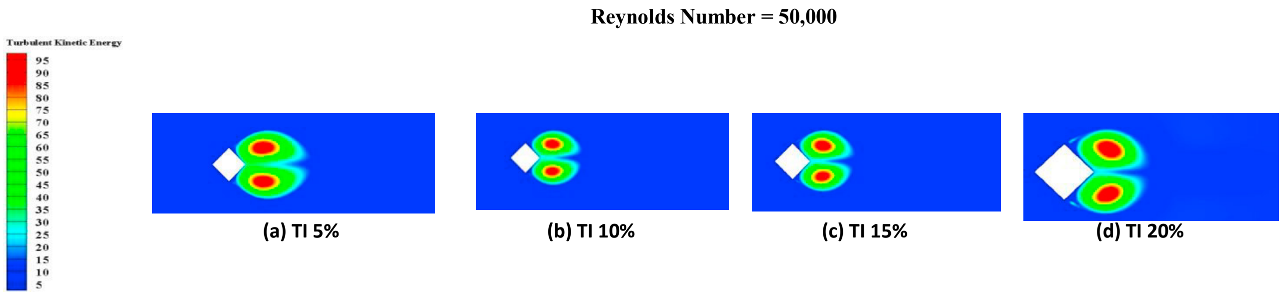

Fig. 6.Contoursplotsof turbulent kinetic energy (TKE) at Re = 50,000

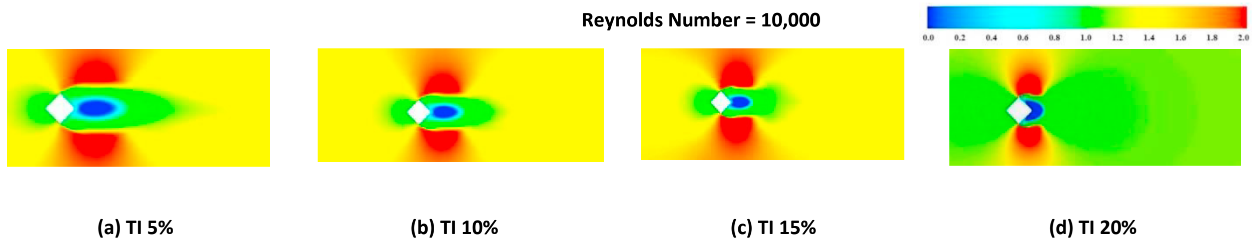

Fig. 7. Non-dimensional velocity contours plots of different inlet turbulent intensity at Re= 10,000

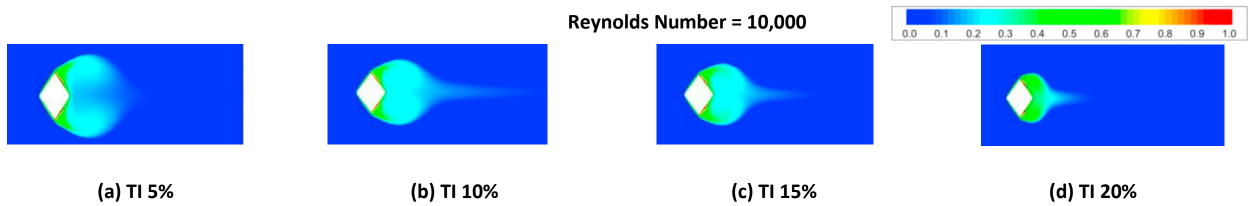

Fig.8.Non-dimensional temperature contours plots of different inlet turbulent intensity at Re= 10,000

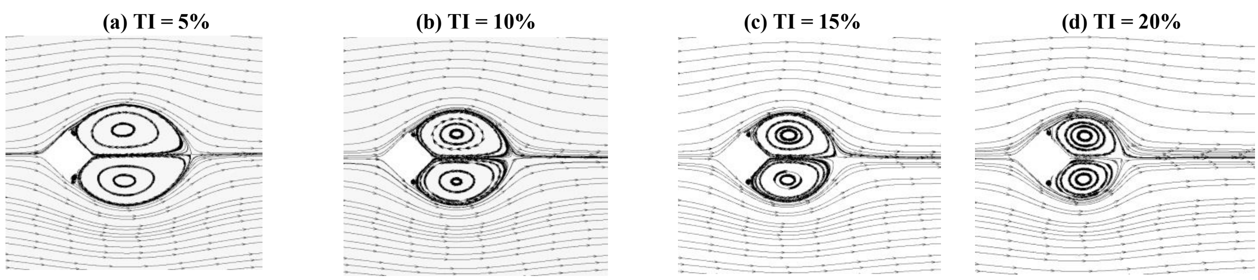

Fig. 9.Streamlineplots of different inlet turbulent intensity at Re= 100

The point of separation is identified by investigating the variation in local Nusselt number (Nu). Fig. 10 shows the fluctuation of Nu over DC curve length for the value of Reynolds number 1,000 to 1, 00,000 with turbulence intensity of 10% and 15%. For each and every value of Reynolds number, Nu has the higher value at point C, it increases slightly nearer to 800 to 900 but beyond that, this value very sharply drops. It is also seen that with increasing Reynolds number the separation point is also shifting.

The variation of C_p with curve length is shown in Fig. 11.A parabolic curve for flow over DC surface BC which describes a steady rise in the value of C_p from its range -2.0 to 1.0 in the x-y plane.

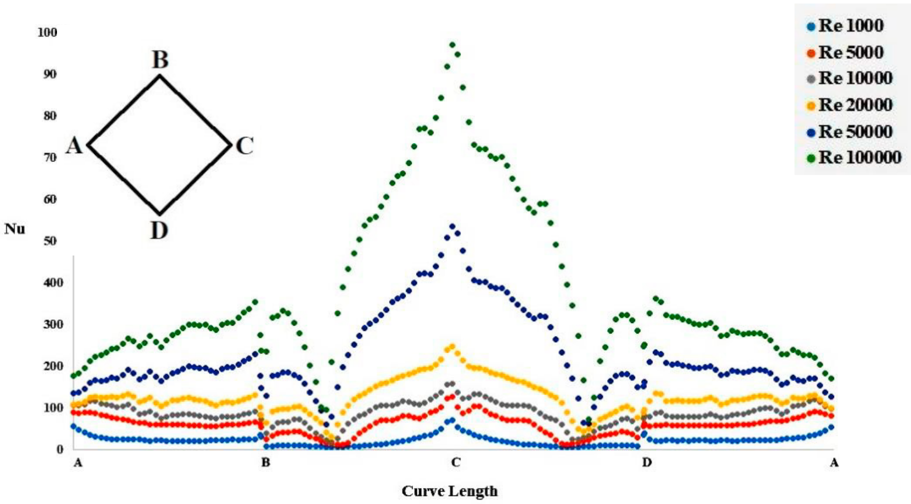

Fig. 10.Distribution of Nusselt number around diamond cylinder at 10% turbulent intensity.

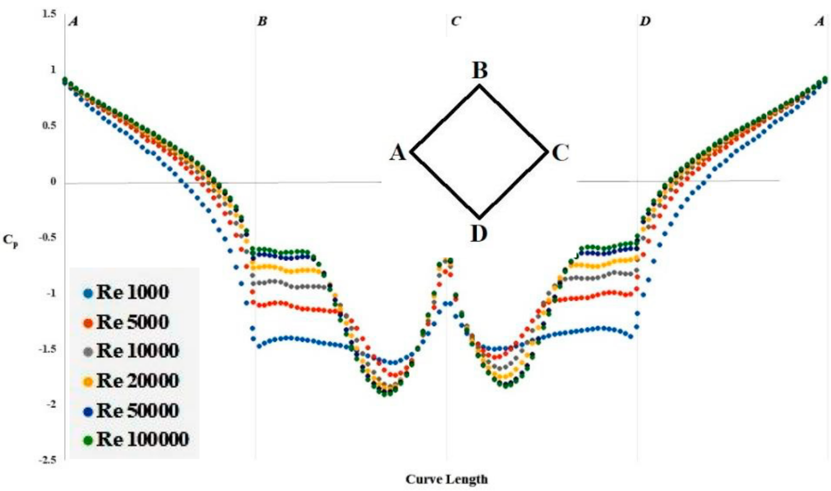

Fig. 11.Variation of C_p over the diamond cylinder at TI = 10%

After reaching the maximum value, the value of C_p again drops slowly. Also, one can see from the figure that the value of C_p is almost varies at same rate at the surface AB and DA.

5. Conclusions

Influence of TI at inlet on the mixed convective fluid flow and heat transfer around square and circular bluff cylinder are studied in a wide range of Re. The turbulent i.e.; transition SST model has been used for computational analysis purpose. The influence of various inlet turbulent intensity and Reynolds number on DC at a distance 15D from the inlet are comprehensively studied at Pr 0.707. The results reflect that the influence of TI on local Nu as well as the Nu_{avg}. The total HT rate of the DC increases with Re. But, also the results shows that the pressure coefficient is not considerably affected by inlet TI. The order of enhancement is about 10%.

References

[1] Thompson M, Hourigan K, Sheridan J. Three-dimensional instabilities in the wake of a circular cylinder. Thermal Fluid Sci1996:12;190–196.
[2] Majumdar S, Rodi W, Zhu J. Three-dimensional finite volume method for incompressible flows with complex boundaries. J. Fluid Eng. ASME 1992;496–503.
[3] Coutanceau M, Bouard R. Experimental determination of the main features of the viscous flow in the wake of a circular cylinder in uniform translation: steady flow. J. Fluid Mech1997;70:231.
[4] Oosthuizen PH, Madan S. The Effect of Flow Direction on Combined Convective Heat Transfer from Cylinders to Air Trans. ASME C. J. Heat Transfer 1971;93:240–242.
[5] Farouk B, Guceri SI.Natural and Mixed Convection Heat Transfer Arounda Horizontal Cylinder within Confining Walls.Numer. Heat Transfer 1982;5(A):329–341.
[6] Chang KS, SaJY.The Effect of Buoyancy on Vortex Shedding in the Near Wakeof a Circular Cylinder. J. Fluid Mech 1990;220:253–266.
[7] Singh S, Biswas G, Mukhopadhyay A. Effect of Thermal Buoyancy on the FlowThrough a Vertical Channel with a Built-In Circular Cylinder. Numer. Heat Transfer 1998;34(A):769–789.
[8] Saha AK. Unsteady Free Convection in a Vertical Channel with a Built-In Heated Square Cylinder. Numer. Heat Transfer 2000;38(A):795–818.
[9] Gowda YTK, NarayanaPAA,Seetharamu KN. Numerical Investigation of Mixed Convection Heat Transfer Past an In-Line Bundle of Cylinder. Heat Mass Transfer 1996;31:347–352.
[10] Gowda YTK, Narayana PAA, Seetharamu KN. Mixed Convection Heat Transfer Past In-Line Cylinders in a Vertical Duct. Numer. Heat Transfer1997;31(A):551–562.
[11] Lacroix M, Carrier R. Mixed Convection Heat Transfer from Vertically Separated Horizontal Cylinders within Confining Walls. Numer. Heat Transfer 1995;27A(4):487–498.
[12] Jue TC, Wu HW, HungSY.Heat Transfer Predictions around ThreeHeated Cylinders between two Parallel Plates.Numer. Heat Transfer 2001;40A(7):715–733.
[13] Gandikota VSG, Amiroudine S, Chatterjee D, Biswas G. Effect of Aiding-Opposing Buoyancy on Two-Dimensional Laminar Flow and Heat Transferacross a Circular Cylinder.Numer. Heat Transfer 2010;58(A):385–402.
[14] 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 2016;157:19-28.
[15]Bhattacharyya S, Chattopadhyay H, Saha SK. Numerical Study on heat transfer enhancement of laminar flow through a circular tube with artificial rib roughness. Journal of Refrigeration, Air conditioning, Heating and ventilation2014:1(3):14-19.
[16]Bhattacharyya S, Chattopadhyay H, Swami A, Uddin MK. Convective heat transfer enhancement and entropy generation of laminar flow of water through a wavy channel. International Journal of Heat and Technology2016:34(4):727-733.
[17] Sumner D, Wong SST, Price SJ, Paidoussis MD. Fluid behaviour of side-by-side circular cylinders in steady cross-flow. J Fluids Struct 1999;13:309–38.
[18] Farrant T, Tan M, Price WG. A cell boundary element method applied tolaminar vortex shedding from circular cylinders. Comput Fluids2001;30:211–36.
[19] Zukauskas A, HarnettJP, Irvine Jr. T.F. (Eds.). Heat transfer from tubes in cross flow. Advances in Heat Transfer1978; 8.
[20] Dhiman A, Shyam R. Unsteady Heat Transfer from an Equilateral Triangular Cylinder in the Unconfined Flow Regime. ISRN Mech. Eng.2011: 1–13.
[21] De AK, Dalal A. Numerical Simulation of Unconfined Flow Past a Triangular Cylinder. Int. J. Numer. Methods Fluids2006;52:801–821.
[22] Bhattacharyya S, Saha S, Saha SK. Laminar flow heat transfer enhancement in a circular tube having integral transverse rib roughness and fitted with centre-cleared twisted-tape. Thermal and Fluid Science2013;44:727-735.
[23] Bhattacharyya S, Saha SK. Thermohydraulics of laminar flow through a circular tube having integral helical rib roughness and fitted with centre-cleared twisted-tape. Thermal and Fluid Science 2012;42:154 -162.
[24] Saha SK, Bhattacharyya S, Pal PK. Thermohydraulics of laminar flow of viscous oil through a circular tube having integral axial rib roughness and fitted with center-cleared twisted-tape. Thermal and Fluid Science 2012;41:121 – 129.
[25] Saha SK, Bhattacharyya S, Dayanidhi GL. Enhancement of heat transfer of laminar flow of viscous oil through a circular tube having integral axial rib roughness and fitted with helical screw-tape inserts. Heat Transfer Research2012:43(3):207–227.
[26] 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 Journal2017;17(3):193-197.
[27] Bhattacharyya S, Chattopadhyay H, Benim AC. Numerical Investigation on Heat Transfer in a Circular Tube with Inclined Ribs. Progress in Computational Fluid Dynamics, An International Journal 2017;17(6):390-396.
[28] 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. DOI.10.1080/01457632.2018.1474593.
[29] Bhattacharyya S. Experimental study on effect of heat transfer enhancement of heat exchanger tube inserted with short length spring tapes. Iran J SciTechnol Trans Mech Eng. Online. DOI.org/10.1007/s40997-018-0251-0.
[30] 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 2018;7 (1):118-126.