Experimental Investigation of the Effect of Single and Twin Bluff Bodies on the Turbulent Flow in an Asymmetric Rectangular Diffuser

SOMNATH BHATTACHARJEE[1], ARINDAM MANDAL[2], RABIN DEBNATH[3]
SNEHAMOY MAJUMDER[4]

[1]Department of Computer Science and Engineering, Jadavpur University, 188, Raja S.C.Mullick Road, Kolkata- 700 032, West Bengal, INDIA
[2]Department of Mechanical Engineering, Heritage Institute of Technology, Chowbaga Road, Anandapur, P.O.: East Kolkata Township, Kolkata-700107, INDIA
[3] Directorate of Industrial Training, Government of West Bengal, Karigori Bhavan, Pin-700 160, West Bengal, INDIA
[4]Department of Mechanical Engineering, Jadavpur University, 188, Raja S.C. Mullick Road, Kolkata-700032, INDIA

Abstract: - Experimental investigation of the effect of bluff bodies on the turbulent flow through an asymmetric diffuser has been carried out. The rectangular diffuser is designed and made keeping similarity to that used by Buice and Eaton, [1], having an inclination angle of **10°**. Three geometrical configurations have been selected for the experimentation. **(I)** At first the experiment has been carried out for the validation of the present results with Buice and Eaton, [1], placing no bluff body at all. **(II)** Thereafter measurements have been carried out by placing a single bluff body on the horizontal floor of the diffuser to estimate the effect of the bluff body on the downstream flow. **(III)** Finally, two identical bluff bodies are placed on the horizontal floor of the diffuser and experimental work has been carried out in order to investigate the effect of the existence of two bluff bodies on the downstream flow through the diffuser. The present results agree well with the results of Buice and Eaton, [1], to show that the recirculation zone appears just adjacent to the inclined plane when there is no bluff body in a diffuser. Also, the detailed investigation for velocity of flow field, distribution of skin friction factor along with uncertainty analysis as well as the correlation between friction factor and Reynolds Number have been carried out in the research paper.

Key-Words: - Bluff body, Diffuser, Recirculation, Reynolds number, Skin friction Coefficient, Bulk Average Velocity.

Received: March 28, 2022. Revised: October 25, 2022. Accepted: November 26, 2022. Published: December 31, 2022.

1 Introduction

The axi-symmetric rectangular flow has gotten attention of researchers long ago.The characteristics of air flow in a rectangular two dimensional diffuser with fully developed inlet flow condition have been measured by Obi, Aoki and Masuda, [2]. The authors in [2], have experimentally investigated the flow separation and recirculation region generated in the diffuser of high aspect ratio. Obi, Ohizumi, Aoki and Masuda, [3] have also examined two-dimensional separated flow in an asymmetric diffuser having fully developed inflow conditions with diffuser inclination angle 10^o and an expansion ratio of 4.7. The authors have measured the mean velocities, turbulent quantities and observed the separation occurred at halfway downstream. Gullman-Strand, Törnblom, Lindgren, Amberg and Johansson, [4], have considered the divergent angle of the diffuser as 8.5^o for reducing the size of the separated region and accordingly, a high aspect ratio of the diffuser results achieving a high degree of two-dimensionality of the average flow. They have also made comments that fully attached flow can occur for the angle around 7^o. Buice and Eaton, [1], have experimentally examined asymmetric plane diffuser flow and estimated the recirculation zones. Present authors have been motivated by their very unique and high quality experimental results and give effort to validate their experimental results considering the same aspect ratio and flow conditions through the diffuser. Earlier, experimental research works had been performed on turbulent flows by Okwuobi and Azad, [5] and Azad and Kassab, [6]. According to their investigation, sudden change of adverse pressure gradient at the diffuser throat causes the downstream as mean and turbulent flow fields. Bhattacharjee, Debnath, Roy and Majumder, [7], have experimentally studied the turbulent air flow through a two dimensional asymmetric rectangular diffuser. Majumder, Roy, Bhattacharjee and Debnath, [8], have investigated experimentally the turbulent flow behaviour of air inside a diffuser with an inclination angle of 15°. Coller, [9], has defined the diffuser as an expanding section of a flow-carrying duct used to slow the mean flow. It gives effect in the conversion of kinetic energy to potential energy causing the rise of pressure in the downstream region. Analytical research work on turbulent separated flows in an axi-symmetric diffuser has also been carried out by Sagar, Paul and Jain, [10]. In [10] the authors have narrated that the pressure-induced separation begins in a diffuser with the increase of its half-angle. Mandal, Bhattacharjee, Debnath, Majumder and Roy, [11], have explained that the turbulent flow and boundary layer separation are observed in many industrial applications. Hwang, Chow and Peng, [12], have investigated numerically that the recirculation length decreases with the increase of the length of the bluff body whereas Antoniou and Bergeles, [13], have experimentally studied that by the enhancement of the aspect ratio, the flow reattaches at the downstream with the reduction of turbulence scale and recirculation length. Hwang, Chow and Chiang, [14], have analyzed numerically the turbulent flow around a surface-mounted two-dimensional bluff body of varying length. They have observed that the length of the re-circulating zone remains unchanged with the variation of the length of bluff body in the upstream of flow. In the downstream of flow, the length of the re-circulating region depends on the length of the bluff body. Reattachment on the upper side of the bluff body influences the velocity boundary layer developed at the downstream region. This is valid as the length of bluff body increases. According to [12] the authors have remarked that the length of re-circulating zones depends on the ratio of the boundary layer thickness to the bluff body’s height and on the geometry of the bluff body. Bergeles and Athanassiisdis, [15], have experimentally evaluated the lengths of the re-circulating regions in front of and behind the bluff body. Benodekar, [16], has mentioned that Reynolds number has negligible effect on separation in the case of turbulent flow. Baetke, Werner and Wengle, [17], have remarked that wall boundary conditions at sharp corners strongly control separation of flow. Das, Ghosh and Singh, [18], have asserted in their literature that the flow field around the building or structure situated on the surface boundary layer is fully turbulent and complex with separations at each surface of the building. Bhattacharjee, Debnath, Mandal, Majumder and Roy, [19], have observed through the experimental work using two bluff bodies of different sizes in a rectangular diffuser and recirculation zones are distinctly pointed out. Mehdi and Mushatet, [20], have pronounced that the separation of boundary layer and recirculation zones behind the bluff bodies mainly depend on the spacing between the bluff bodies. Liu, [21], has analyzed the numerically simulated flow around the cylinders of different shapes changing from square to circle. The author has made the opinion that the drag coefficient and reattachment length decreases independent of the Reynolds number. It is also commented that the drag coefficient of a rounded-corner square cylinder can be lower than the circular cylinder of the same size. Poussou and Plesniak, [22], have observed experimentally the wake formed due to a bluff body propagating through a recirculating flow using Particle Image Velocimetry. Lander, Letchford, Amitay and Kopp, [23], have investigated the influence of bluff body shear layer formation and the resulting impact on flow characteristics using two-dimensional square prism with the help of Particle Image Velocimetry. Bandyopadhyay, Sarkar, Roy and Chanda, [24], have experimentally found the flow field in a rectangular diffuser using a bluff body. The authors have remarked that the boundary layer thickness is somewhat lower at higher inlet velocity and maximum momentum occurs near the bluff body. Nasr, Abdel-Fattah and EI-Askary, [25], have investigated the effect of the number of ribbed walls on heat transfer surface and friction property. In [25] the authors have commented that ribs mounted on the heat transfer surface make disturbance on the boundary layer growth and augmentation of heat transfer from the surface to the fluid and better mixing is possible. The useful scientific idea regarding flow separation and recirculation bubble formation near the bluff bodies is still very scarce in the published literature. The present experimental work is associated with the diffusing air fluid flow with generation of recirculation. The main objective of the present study is to explore the turbulent air flow behaviour experimentally at normal atmospheric pressure and temperature in an asymmetric two dimensional diffuser of rectangular cross section fitted with single and twin bluff bodies respectively.

2 Experimental Method

The experiment has been conducted in an asymmetric horizontal rectangular diffuser maintaining similarity with the geometry of the experimental set up used by Buice and Eaton, [1]. The lower portion of the diffuser is horizontal and its upper one is tilted. The inlet section of the diffuser is 0.015 × 0.200 m which is combined with the outlet section of the air blower. The aspect ratio of the diffuser and the horizontal length of the inlet channel are (13.33:1) and 0.83 m respectively. For maintaining similarity with the experimental set up used by Buice and Eaton, [1], the normal height of the downstream tail end of the diffuser is $4.7H$ and the axial length of the diffuser is $21H$ with an inclination angle of 10°. The outlet section of the diffuser of rectangular shape is 0.07 × 0.2 m extending the length up to $77H$. Both the upper and lower walls of the diffuser are made of 0.006 m thick Plexi-glass sheet. The side walls of the diffuser are constructed of plain transparent glass sheet of 0.004 m thickness. The turbulent air from the blower finally enters the inlet section of the diffuser after passing through the vibration absorber, wire-mesh and honey comb chamber accommodated in between the blower and the diffuser. This type of construction ensures the uniform turbulent fluid flow through the inlet section of the diffuser. The blower which is coupled with a D.C. motor supplies air to the diffuser. Pressure Transducer is used in the experimental work for measuring stagnation and static pressure difference at a certain location inside the diffuser. The specification of the Pressure Transducer used is as follows: Model-CP300-HOP, Sl.No.-12040821, Type-304, and Range: -10000/10000 Pa, Air velocity range - 2 to 100 m/s with accuracy: ± 0.5% of reading ± 1 Pa. The measured pressure difference is utilised to calculate the axial mean velocity of the turbulent fluid flow. The calibrated Pitot tube has been used to determine pressure heads. The probe is introduced from numbers of drilled holes situated over the top inclined surface along the mid-stream plane $\mathbf{X-X}$ at different station locations along the diffuser length. Pressure differences (Δp Pa) between the stagnation and static pressures are calculated at various vertical heights measured from the bottom horizontal wall by using Pitot tubes mounted on a traversing equipment. Thermometer and Barometer are employed for taking the reading of local atmospheric temperature and pressure inside the laboratory. The diffuser is installed perfectly on a wooden frame structure. The working fluid is air assuming its density ρ = 1.132 kg/m^3 at normal room temperature (300 K) and pressure (101.6 KPa) inside the laboratory.

At first, the current experimental work for validation of the results using the set up without a bluff body has been carried over only at Reynolds number equal to 1.367×10^4. The two-dimensional parameters measured for the mean turbulent fluid velocity profile have been plotted comparing the benchmark experimental results of Buice and Eaton, [1].

Secondly, a single polished wooden bluff body made of dimension 0.20 × 0.02 × 0.02 m is placed over the horizontal bottom wall at a distance of 0.18m from the inlet section of the diffuser. For third experimentation work utilising the same set up, two wooden bluff bodies, each of equal dimensions of 0.20 × 0.02 × 0.02 m, are placed on the bottom wall at the distances of 0.18 m and 0.30 m respectively from the inlet section of the diffuser and a clear gap existing between them is 0.12 m. For second and third experimental works the inlet flow condition is turbulent with Re = 1.367×10^4, 1.749×10^4 and 1.833×10^4 respectively. Reynolds numbers are calculated based on the inlet height of the diffuser and axial mean flow velocity. Here also two-dimensional measurements have been taken in the mid-stream plane $X-X$. The velocity distribution curves are obtained at different stations along the horizontal length of the diffuser.

The equations for the mean velocity of fluid u, u_{avg} m/s, Reynolds numbers and coefficient of skin friction of the working fluid flowing passing through the diffuser C_f are hereby given below**:**

$$u = \sqrt{2\Delta p \times 9.81 / \rho} \quad m/s \tag{1}$$

$$\mathrm{Re} = \rho U_b H / \mu \tag{2}$$

$$C_f = \mu\left(du/dy\right) / \tfrac{1}{2}\rho u_{avg}^2 \tag{3}$$

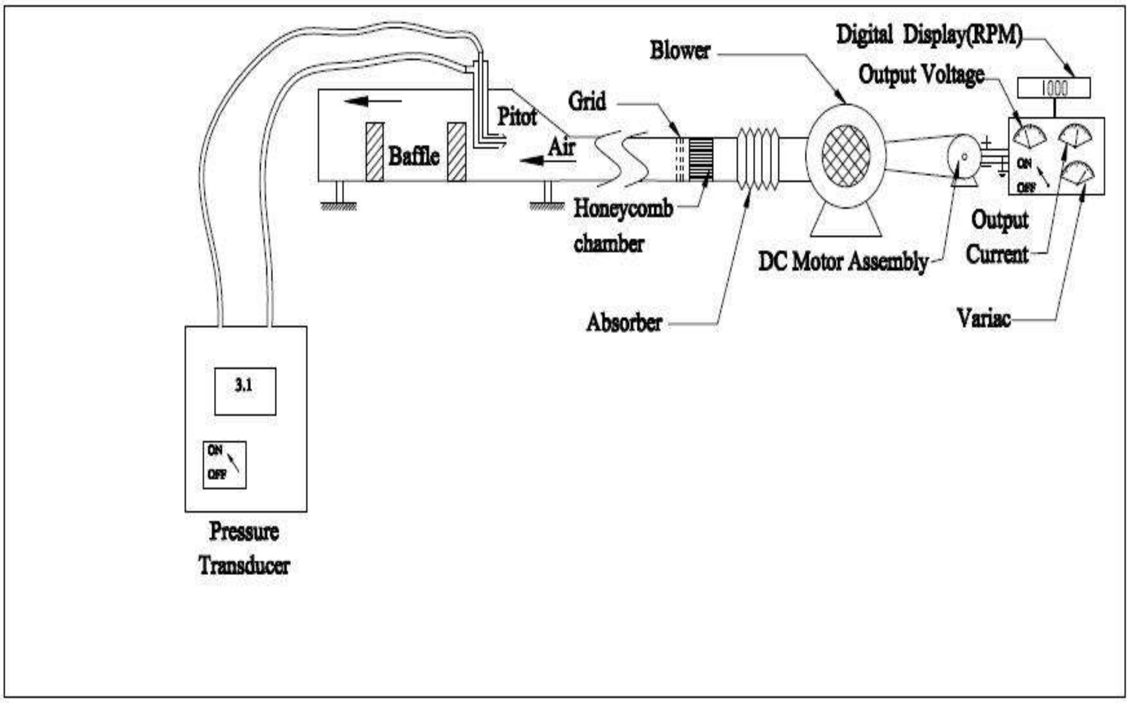
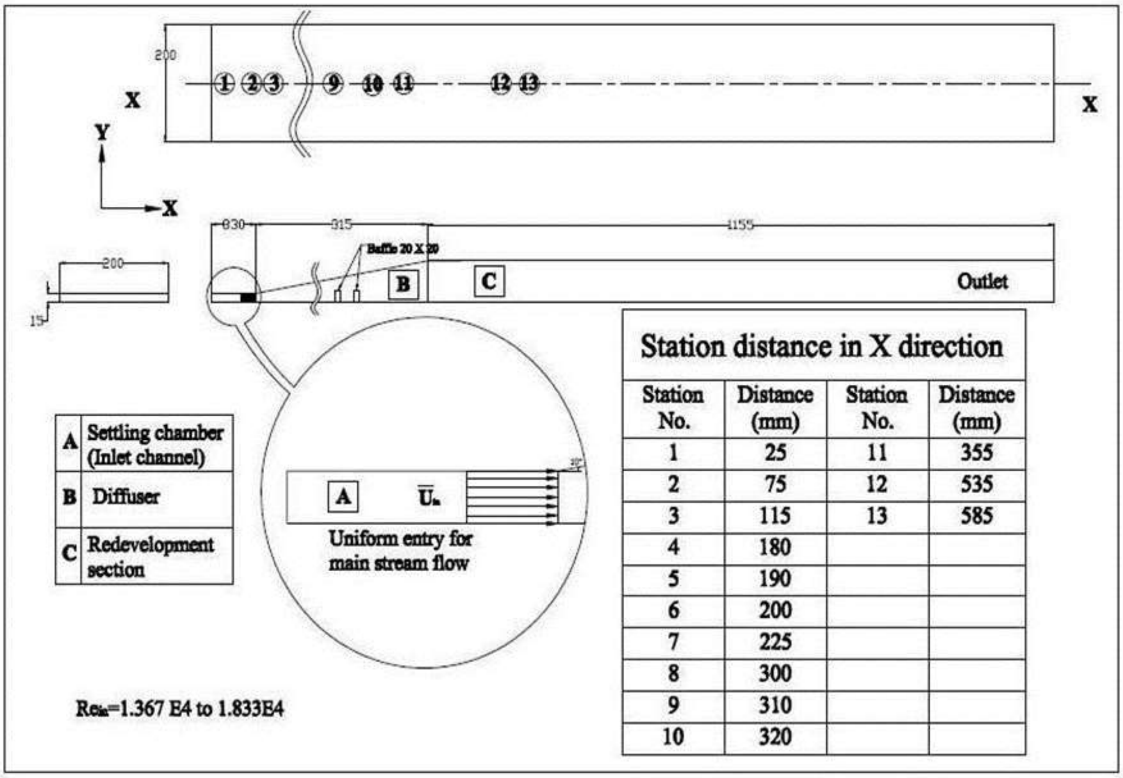


Fig. 1 Schematic diagram of the experimental set up (using twin bluff bodies/baffles)



Station No.	Distance (mm)	Station No.	Distance (mm)
1	25	11	355
2	75	12	535
3	115	13	585
4	180		
5	190		
6	200		
7	225		
8	300		
9	310		
10	320		

Fig. 2 Schematic diagram of the diffuser using twin bluff bodies/baffles (not to scale), all dimensions are in mm

Fig.3 Photograph of the Experimental Set Up

In figures 1 and 2 the schematic diagrams of an experimental setup fitted with two bluff bodies and holes on the inclined surface at different station distances have been shown. For the case of the experiment with a single bluff body, the diagrams are necessary to be modified by replacing the double bluff body with the single one.

3 Results and Discussion

3.1 Diffuser without Bluff body and Validation of Results

The experimental results shown in Fig.4 (a and b) of turbulent air flow through the asymmetric rectangular diffuser used in the experimental work have been validated with the published experimental work of Buice and Eaton, [1]. Recirculation and reattachment of the flow have been found out inside the diffuser adjacent to the inclined surface. The velocity distributions are presented in the two-dimensional coordinate system in which $\boldsymbol{X}$ -axis is parallel to the inlet upstream flow and the origin of $\boldsymbol{X}$ - axis is lying at the onset of diffuser after the constant rectangular section while the $\mathbf{Y}$ axis is normal to $\boldsymbol{X}$ - axis. The centre-line velocity at the inlet is $1.14U_b$ where $\boldsymbol{U_b}$ is obtained as 12.0 m/s and the corresponding Reynolds number is calculated using the equation (2) or Re $=1.367\times10^4$.

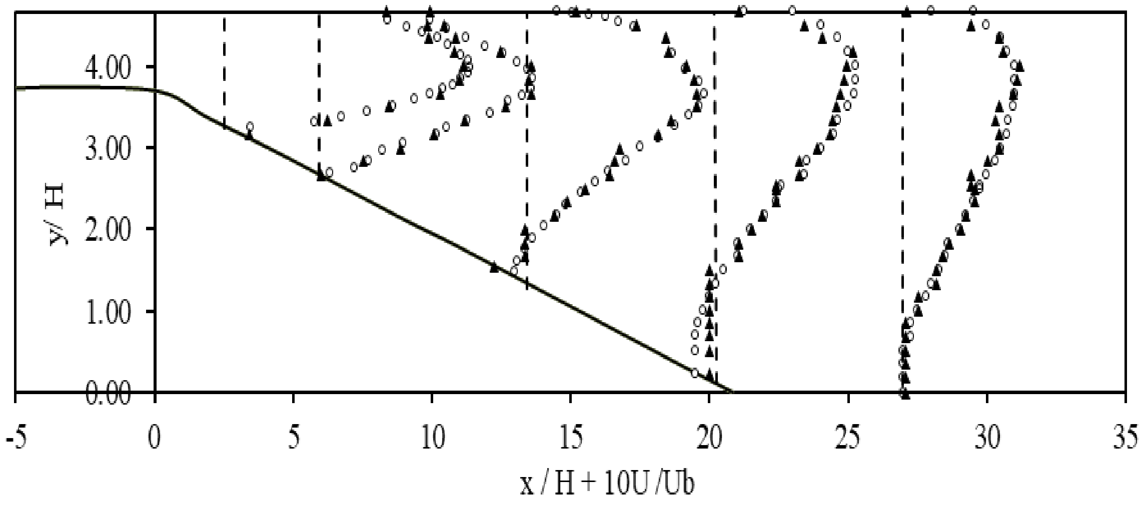


Fig.(a)

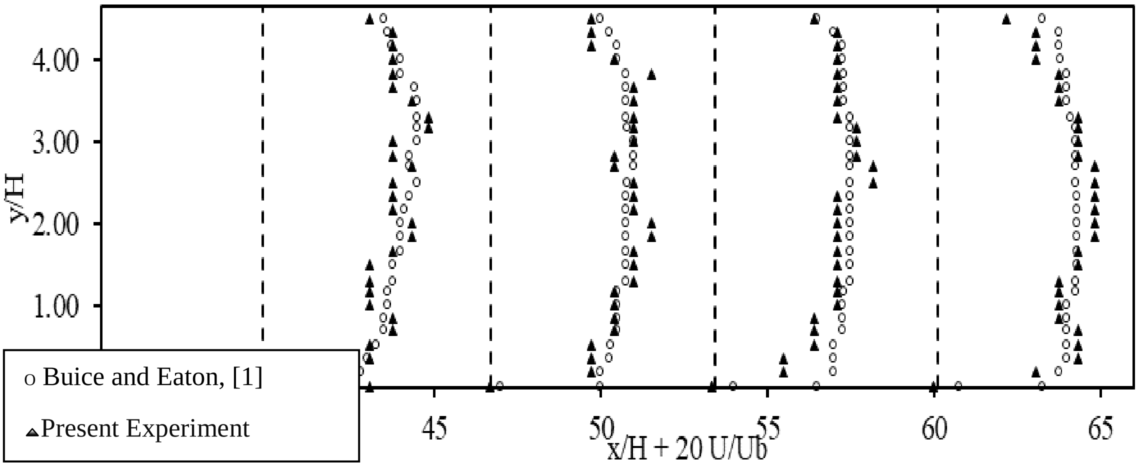


Fig.(b)

Fig. 4 Validation of the current experimental results with the results of Buice and Eaton, [1]

At the station $X = 0.035\,\text{m}$ i.e. $\left(X/H + 10U/U_b = 3.405 \right)$ the velocity profile shows no sign of the presence of flow separation. In fact, this is the just outer section of the inlet rectangular section receiving the incoming flow in the settling chamber which is 0.083m long. The height of the settling chamber is 0.015m; therefore, the ratio of the length of the settling chamber and height for this section is almost 53.33. It is observed that the velocity distribution along vertical Axis-Y obtained from the present experiment agrees very well with that of Buice and Eaton, [1], experimental work. The same good agreement has also been observed at a station distance $X = 0.09\,\text{m}$ for $\left(X/H + 10U/U_b = 6.0 \right)$, but no phenomena of any recirculation i.e. separation or the reattachment of the turbulent flow has occurred. This is due to the fact that the flow is still not facing sufficient adverse pressure to have a backflow near the wall. But the presence of a separating flow is clearly observed at the axial distances $X = 0.20\,m\,and\,0.30\,m$ respectively corresponding to $X/H + 10U/U_b = 13.4\ and\ 20.2$ with a clear similarity and of the same nature of the back flows shown by Buice and Eaton, [1]. At a station of distance $X = 0.39\,m$ for $X/H + 10U/U_b = 27.08$ the recirculation bubbles disappear showing the reattachment of the separating flow. The other axial station distances chosen for the current experimentation are $X = 0.60\,m, 0.70\,m, 0.80\,m\,and\,0.90\,m$ respectively with the corresponding non-dimensional values of $X/H + 20U/U_b = 40, 46.67, 53.33\,and\,60$ respectively. In all of the above cases the current results agree well with the experimental results of Buice and Eaton, [1].

3.2 Diffuser with Single and Twin Bluff bodies:

The present investigation provides a complete set of information in the upstream position, diffuser region and downstream of the diffuser model of 10° inclinations. The single bluff body of size 0.20 × 0.02 × 0.02 m is placed over the bottom horizontal wall at a distance of 0.18 m from the inlet section of the diffuser. The inlet condition of the air flow is turbulent with $\text{Re} = 1.367 \times 10^4$, 1.749×10^4 and 1.833×10^4 respectively. Measurements have been taken in the mid-stream plane $X - X$.

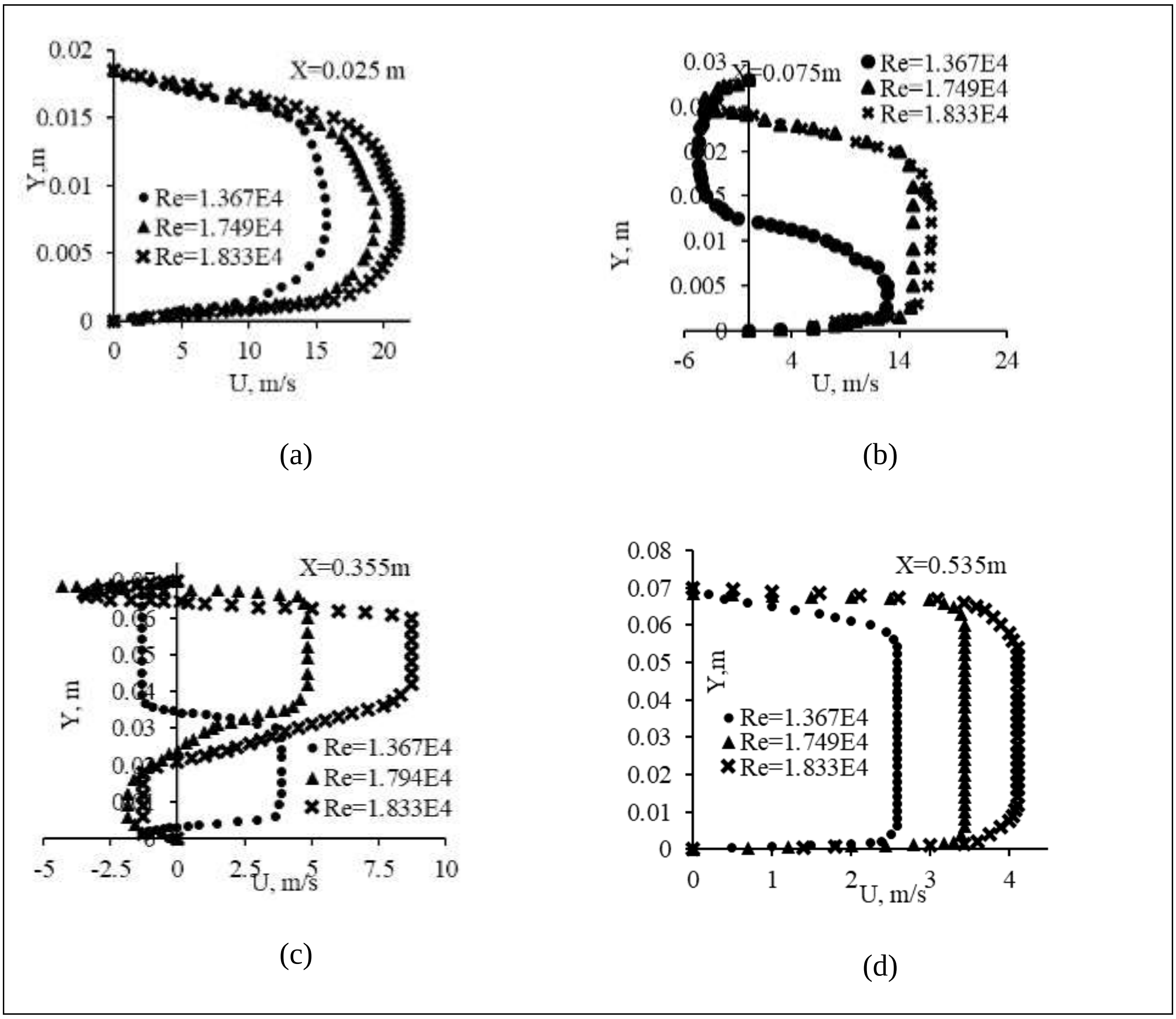


Fig. 5 Transverse distribution of Stream wise velocity (for single bluff body)

Figures 5(a) to 5(d) illustrate the distributions of mean velocities of fluid at different axial station distances along the $\mathbf{X}$-axis from the inlet cross section of the diffuser. From the figure 5(a), it has been observed that no generation of recirculation occurs before the edge of the bluff body, the measurements being taken from the lower horizontal wall of the diffuser towards the upper inclined wall. As the measurements are conducted in the downstream direction it has been found that there exists recirculation of the turbulent flow adjacent to the inclined surface. The recirculation occurs at a distance of 0.075 m away from the inlet of the diffuser and it ends at a distance of 0.535 m away from the inlet of the diffuser. The effective recirculation bubble length therefore equals to 0.460 m. Flow separation is generated due to the adverse pressure gradient of the flow caused by the gradual expansion of the volume of flow with larger inclination angle of the diffuser.

When two bluff bodies of equal size are placed, measurements have been carried out over the top surfaces of the bluff bodies at the station distances of 0.18 m to 0.20 m and 0.30 m to 0.32 m respectively. Recirculation occurs in the region between the distances of 0.075 m and 0.535 m away from the inlet section of the diffuser. Figure 5(d) as well as 6(f) clearly depicts the occurrence of re-attachment of the flow in the downstream region of the diffuser. From figures 5(b) to 5(c) and figures 6(b) to 6(e), it has been observed that recirculation size is somewhat smaller above the top surface of the baffles than that in other locations. Recirculation bubble generated before the second baffle towards the inlet section is smaller in size than that generated just after the baffle in the downstream region inside the diffuser. From the figures 5(a) to 5(d) and 6(a) to 6(f) it is also evident that the axial mean velocity increases with increase of Reynolds number.

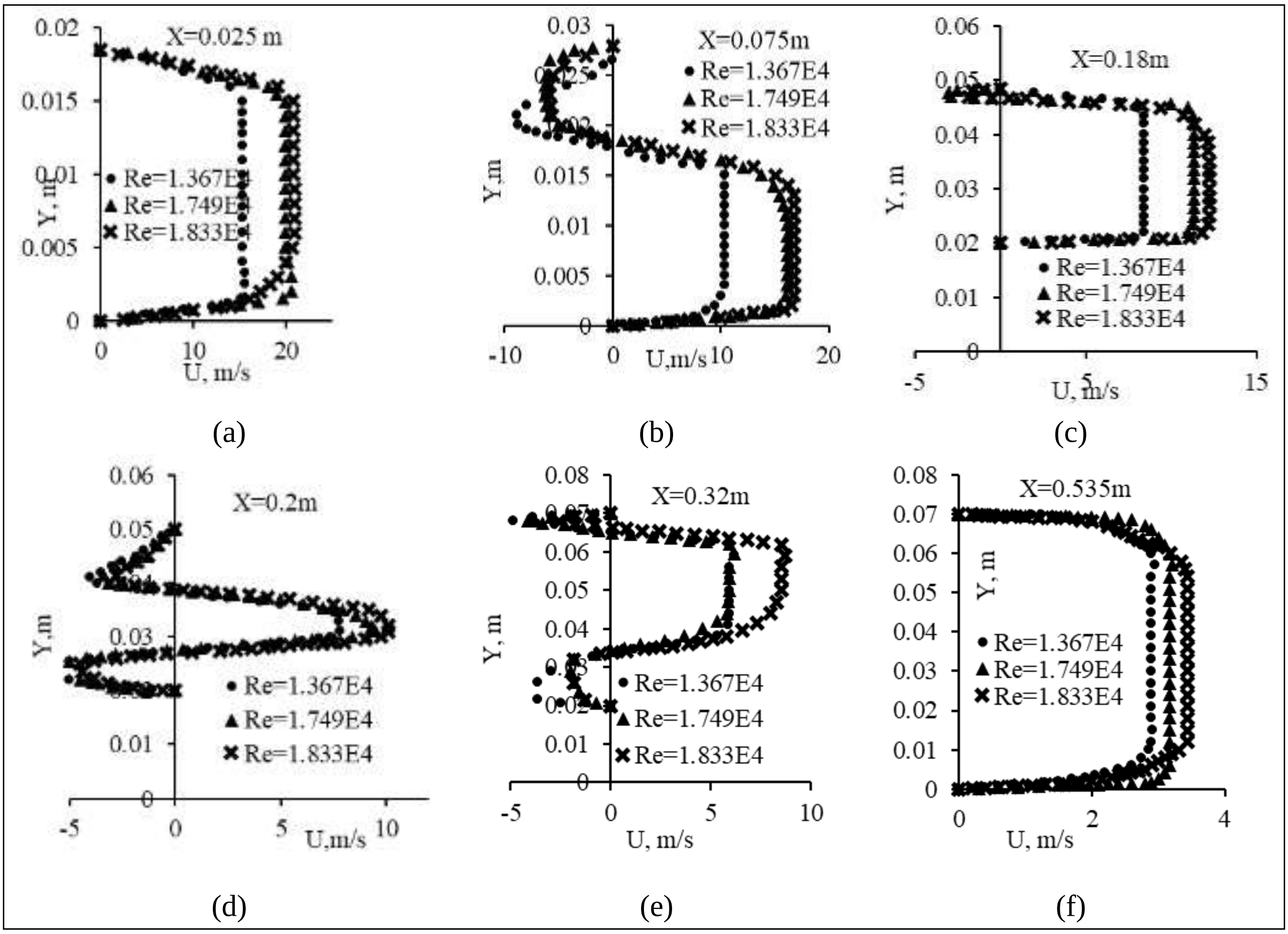
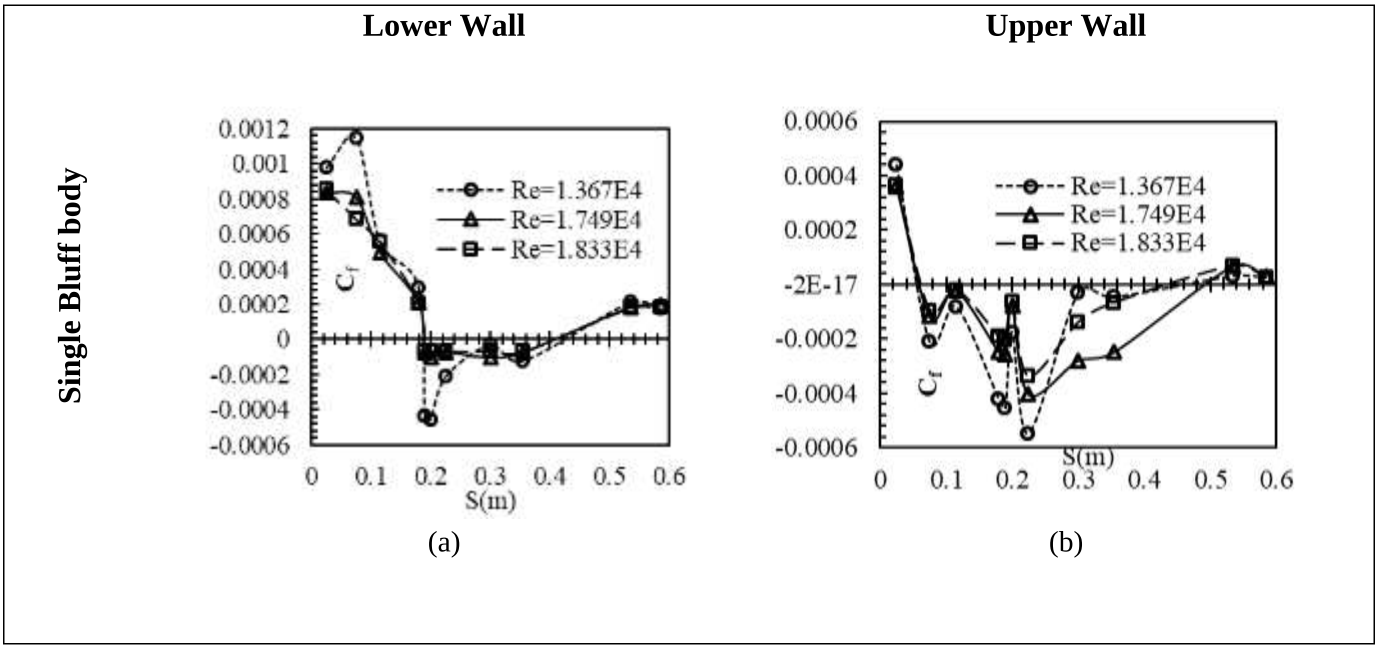
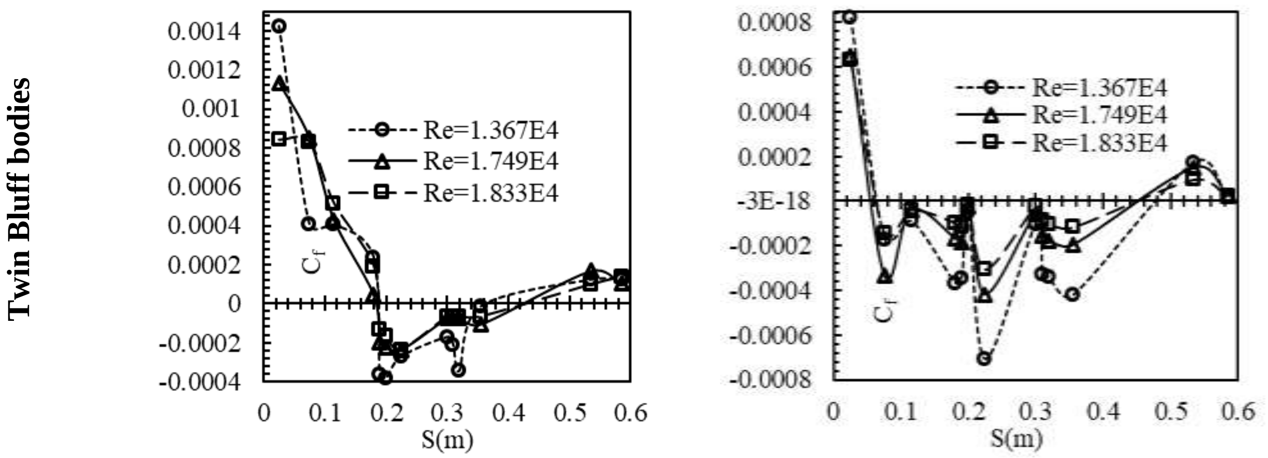


Fig. 6 Transverse distribution of Stream wise velocity (for double bluff bodies)

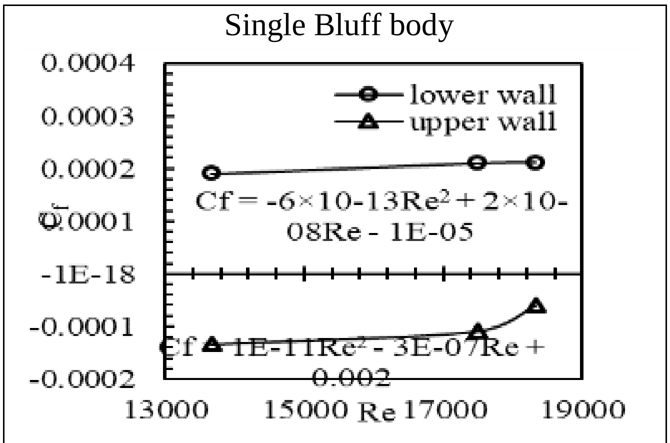




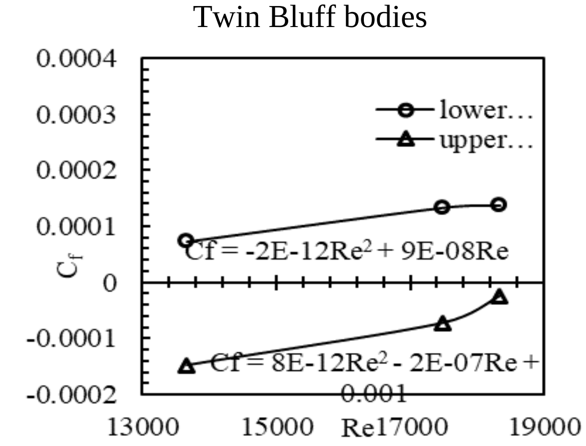
(c) **Lower Wall** (d) **Upper Wall**

Fig. 7 Variation of Coefficient of Friction

Figures 7(a) to 7(d) illustrate the values of coefficient of skin friction on the lower and upper walls at different Reynolds numbers for the cases of single and double bluff bodies respectively. Friction force increases as the solid surface of the bluff bodies are introduced disrupting the fluid flow and it is clearly evident from the figures 7(a) to 7(d). From the figures 7(a) to 7(d), it is observed that coefficient of friction decreases with increase of axial distance measured from the inlet section of the diffuser and is found negative at the recirculation zones. Coefficient of skin friction reaches nearly a constant value at the outlet of the diffuser. It has been also seen from the figures 7(a) to 7(d), coefficient of skin friction increases with the increase of Reynolds number of the turbulent flow in a diffuser. The curves so obtained are drooping in nature where the value of coefficient of skin friction decreases with the increase of station distances.



(a)



(b)

Fig.8 Variation of Coefficient of friction with Re

Figures 8(a) and 8(b) show the variation of coefficient of skin friction with the increase of Reynolds numbers for the cases of single and twin bluff bodies respectively. The current experimental value of the coefficient of friction correlates as the function of Reynolds number and the relevant correlations achieved for upper wall as well as lower wall of the diffuser are given in Table1. These correlations are valid for Re=1.367 × 10^4, 1.749 ×10^4 and 1.833 × 10^4.

Table 1

	Lower Wall	**Upper Wall**
Single Bluff Body	$C_f = -6\times10^{-13}\mathrm{Re}^2 + 2\times10^{-8}\mathrm{Re} - 1\times10^{-5}$	$C_f = 1\times10^{-11}\mathrm{Re}^2 - 3\times10^{-7}\mathrm{Re} + 0.002$
Twin Bluff Bodies	$C_f = -2\times10^{-12}\mathrm{Re}^2 + 9\times10^{-8}\mathrm{Re}$	$C_f = 8\times10^{-12}\mathrm{Re}^2 - 2\times10^{-7}\mathrm{Re} + 0.001$

3.3 Uncertainty of experimental results

Uncertainty has been calculated for the fluid flow parameters measured as (i) for Reynolds number = **3.3% to 3.4%** and (ii) for Coefficient of friction = **1.198% to1.67%** respectively.

4 Conclusion

The present experimental results are validated with the benchmark work of Buice and Eaton, [1], considering the same geometry of diffuser and boundary conditions of the turbulent flow in an asymmetric diffuser. The current study shows a good agreement with Buice and Eaton, [1]. This validation approves the authenticity and accuracy of the experimental set up designed, developed and fabricated to measure the turbulent flow parameters through a rectangular diffuser. The matching of the velocity distribution also confirms the authenticity of the current experimental methodology.

The present experiment has been aimed to distinguish the effect of single and double bluff bodies of identical geometry in shape and size fitted on the bottom wall of an asymmetrical two dimensional horizontal diffuser with an inclination angle of 10° on the turbulent fluid flow. It has been observed that recirculation zones are generated due to the presence of the bluff bodies opposite to the flow direction. Recirculation occurs on the top of the bluff body for the case of single bluff body. Thereafter it exists after the bluff body towards the outlet of the diffuser with larger size. For the case of twin bluff bodies, recirculation also exists between the spacing of the two bluff bodies. Coefficient of skin friction increases with the increase of Reynolds number. With the increase of axial distance measured from the inlet section of the diffuser, coefficient of skin friction decreases and is found to be negative at the recirculation zones. Coefficient of skin friction reaches a constant value at the downstream side of the diffuser. The value of coefficient of skin friction is lesser in the upper wall and higher in the lower wall in case of the flow through the diffuser. Some relevant correlations have also been established between the coefficient of skin friction and Reynolds number which is shown in Table no.1. These correlations are valid for corresponding Reynolds numbers of 1.367×10^4, 1.749×10^4 and 1.833×10^4 respectively.

The significant use of the bluff bodies of different sizes at different spacing and orientations inside the diffuser duly validated with the results of Buice and Eaton, [1], makes it possible to optimize the length of the diffuser and it economises the industrial use of the diffuser.

Acknowledgment:

The authors express their sincere thanks and gratitude to all who have extended their helping hands by giving valuable suggestions and technical support for this experimental work done at Hydraulics Laboratory, Mechanical Engineering Department, Jadavpur University, Kolkata (India).

Nomenclature

C_f	Coefficient of Skin friction
H	Inlet height of the diffuser, m
du/dy	Velocity gradient of the upper and lower wall of the diffuser
Re	Reynolds number
X, x, S	The distance of the station measured from the inlet end of the diffuser, m
U_b	Bulk average velocity, m/s
U	Velocity at certain location and height, m/s
u_{avg}	Average Velocity of Fluid, m/s
Δp	Difference of Stagnation and Static pressure, Pa
ρ	Density of air, kg/m^3
μ	Co-efficient Dynamic viscosity of fluid at room temperature, Pa-s

References:

[1] Buice, C. U. and Eaton, J. K., Experimental Investigation of Flow through an Asymmetric Plane Diffuser, *Journal of Fluids Engineering*, Vol. 122, 2000, pp. 433-435.
[2] Obi, S., Aoki, K. and Masuda, S., Experimental and Computational Study of Turbulent Separating Flow in an Asymmetric Planar Diffuser, *In: Proceedings of the 9th Symposium on Turbulent Shear Flows,* Kyoto, Japan, Vol. 305, 1993, pp. 1-4.
[3] Obi, S., Ohizumi, K., Aoki, K. and Masuda, S., Turbulent Separation Control in a Plane Asymmetric Diffuser by Periodic Perturbation, *Elsevier Science*, 1993, pp. 633-642.

[4] Gullman-Strand, J., Törnblom, O., Lindgren, B., Amberg, G. and Johansson, A. V., Numerical and experimental study of separated flow in a plane asymmetric diffuser, *International Journal of Heat and Fluid Flow*, Vol.25, 2004, pp. 451-460.

[5] Okwuobi, P. A. C. and Azad, R. S., Turbulence in a conical diffuser with fully developed flow at entry, *Journal of Fluid Mechanics*, Vol. 57, 1973, pp. 603- 622.

[6] Azad, R.S. and Kassab, S. Z., Turbulent flow in a conical diffuser: overview and implications, *Physics of Fluids*, Vol. A1: 1989, pp. 564-573.

[7] Bhattacharjee, S., Debnath, R., Roy, D. and Majumder, S., An Experimental Study on Turbulent Air Flow through a Two Dimensional Rectangular Diffuser, *Journal of The Institution of Engineers (India)*, Vol.92, 2011, pp. 3-7.

[8] Majumder, S., Roy, D., Bhattacharjee, S. and Debnath, R., Experimental Investigation of the Turbulent Fluid Flow Through a Rectangular Diffuser Using Two Equal Baffles, *Journal of The Institution of Engineers (India)*, doi 10.1007/s40032-014-0100-x, series C, Vol. 95, issue 1, 2014, pp. 19-23.

[9] Coller, B.D., Vortex model for control of diffuser pressure recovery, *Proc. of Decision and Control*, Australia, 39th.IEEE Conference, Dec.,2000, pp. 1730-1735.

[10] Sagar, D., Paul, A. R. and Jain. A., Computational fluid dynamics investigation of turbulent separated flows in axisymmetric diffusers, *Inter. J. Engg. Sc. and Tech*., Vol. 3(2), 2011, pp. 104-109.

[11] Mandal, A., Bhattacharjee, S., Debnath, R., Majumder, S. and Roy, D., Experimental Analysis of the Turbulent Fluid Flow through a Rectangular Diffuser using Baffles, *Proceedings of National Conference on Recent Advances in Fluid & Solid Mechanics(RAF&SM)*, 2010, pp. 238-242.

[12] Hwang, R. R., Chow, Y. C. and Peng, Y. F., Numerical study of turbulent flow over two-dimensional surface-mounted ribs in a channel, *International Journal for Numerical Methods in Fluids*, Vol.37, 1999, pp. 767- 785.

[13] Anotoniou, J. and Bergeles, G., Development of the Reattached Flow Behind Surface-Mounted Two-Dimensional prisms, *Journal of Fluids Engineering*, Vol.110, 1988, pp. 127-133.

[14] Hwang, R. R., Chow, Y. C. and Chiang, T. P., Numerical Predictions of Turbulent Flow Over A Surface-Mounted Rib, *Journal of Engineering Mechanics* , Vol.125, 1999, pp. 497-503.

[15] Bergeles, G. and Athanassiisdis, N., The flow past a surface-mounted obstacle, ASME, *Journal of Fluids Engineering* , Vol.105, 1983, pp. 461-463.

[16] Benodekar, R. W., Goddard, A. J. H., Gosman, A. D. and Issa, R. I., Numerical prediction of turbulent flow over surface-mounted ribs, *AIAA Journal* , Vol.23, 1986, pp. 359-366.

[17] Baetke, F., Werner, H. and Wengle, H., Numerical Simulation Of Turbulent Flow Over Surface-Mounted Obstacles With Sharp Edges And Corners, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 35, 1990, pp. 129-147.

[18] Das, A. K., Ghosh, A. K. and Singh, N., A Study on Flow Field Around Prismatic Buildings using Les Turbulence Model, *Proceedings of National Conference on Recent Advances in Fluid & Solid Mechanics (RAF&SM)*, February, 27-28, 2010, pp. 166-176.

[19] Bhattacharjee, S., Debnath, R., Mandal, A., Majumder, S. and Roy, D., Experimental Investigation Of Recirculatory Turbulent Flow Past Twin Obstructions Of Different Height Placed At Different Stations Of A Rectangular Diffuser, *Proc. 37th.National and 4th. International Conference on Fluid Mechanics and Fluid Power*, IIT Madras, India, Dec 16-18, 2010.

[20] Mehdi, Q. S. and Mushatet. K. S., Simulation of Turbulent Flow and Heat Transfer Through a Duct with Baffle Plates, *Journal of Engineering and Development*, Vol.12, 2008, pp. 142-157.

[21] Liu, R., FLOW AROUND BLUFF BODIES WITH CORNER MODIFICATIONS ON CROSS-SECTIONS, *M.Sc. Thesis in Mech. Engg., Nanjing University of Aeronautics and Astronautics*, China, 2019.

[22] Poussou, S. B. and Plesniak. M. W., Flow field in the wake of a bluff body driven through a steady recirculating flow, *Experiments in Fluids*, Vol. 56 (40), 2015, pp. 1-16.

[23] Lander, D.C., Letchford, C. W., Amitay M. and Kopp, G. A., Influence of the bluff body shear layers on the wake of a square prism in a turbulent flow, *Phys. Rev. Fluids*, Vol. 1(044406), 2016.
[24] Bandyopadhyay, S., Sarkar, M. K. S., Roy, D. and Chanda, A., Experimental observation of boundary layer of the turbulent flow over a bluff body inside rectangular diffuser, *Procedia Engineering*, Vol.56, 2013, pp. 275- 280.
[25] Nasr, M., Abdel-Fattah, A. and EI-Askary, W.A., STUDY OF TURBULENT FLOW IN RECTANGULAR CHANNEL WITH INCLINED BAFFLES, *Engineering Research Journal, ERJ,* Faculty of Engineering, Minoufiya University, Egypt, Vol.32, No.2, April 2009, pp. 131-143.

Authors' Contributions:

Dr. Somnath Bhattaharjee, Dr. Arindam Mandal and Dr. Rabin Debnath have conducted research and performed the experimental process including validation. Dr. Somnath Bhattacharjee (the corresponding author) has developed the methodology and prepared the initial draft as well as edited subsequently. Prof. (Dr.) Snehamoy Majumder has coordinated the execution of the research work with his immense depth of knowledge and mentorship. His discretionary power and editing capability make the research work more enriched with scientific innovation.

Sources of funding for research presented in a scientific article or scientific article itself

No specific financial support for research is received to carry out this experimental study.