International Journal of Technical Research and Applications e-ISSN: 2320-8163,
www.ijtra.com Volume 3, Issue 4 (July-August 2015), PP. 130-134

EXPERIMENTAL INVESTIGATION OF FLOW AROUND BLUFF BODIES IN A RECTANGULAR DIFFUSER

Siddhartha Bandyopadhyay[1], Dr. Debashis Roy[2], Dr. Arunava Chanda[3]
[1]Research Scholar, [2, 3]Professor, Mech Engg Dept
Jadavpur University, Kolkata
[1]sid_bandy62@ yahoo.co.in

Abstract: The aim of this paper is to find out flow pattern of air streaming past bluff bodies. From pressure profile it is possible to evaluate velocity profile. From pressure profile it has been measured pressure co efficient. To design tall building pressure coefficient is very much parameter. The instability of flow due to occurrence of point of inflection has verified. From literature review the result is verified. Kai Fan Liaw investigated in his thesis the flow pattern around bluff bodies and simulated the flow pattern of the bluff bodies like rectangular, cylindrical bodies. In this paper it has been represented the flow pattern around of bluff in a diffuser. Flow around rectangular square with x/H=1.5 the velocity ratio with varying y/H is matching with our result although in our experiment rectangular bluffs are 110 mm apart. Objective of this paper is to study the flow pattern between two bluffs which is essential to design of high rise tall tower and building. In this paper it has been presented pressure recovery in the diffuser with the presence of two bluff bodies.

Key words: Point of inflection, Pressure co efficient, Pressure recovery coefficient.

I. INTRODUCTION

Bloor, Roshko(1) and Triton(2) focused on the flow in the near wake region of the cylinder. Nishioka Sato(3) conduced experiment on flow past cylindrical body. The range of Reynolds number is 10 to 80. The result is good argument as it in the low Reynolds number. Kai Fan Liaw (4)simulated of flow around bluff bodies and bridge deck. He observed there is aero instability of long span bridges such as flutter, buffeting galloping, divergence and vortex shedding in his experiment. Experimental study of the flow around a circular cylinder has idenfied regions where significiant patterns of flow occur as the Reynolds number changes, especially when the flow changes from laminar to turbulent state. Generally the following regimes have been identified from Roshko,s(1) experiment

Stable range : $40<Re<150$
Transition range : $150<Re<300$
Irregular range : $300<Re<200000$

Similar observations have been made by Zravkovich.(5)Flow becomes very irregular with instability beyond Reynolds number of 200000. Bloor investigated the flow around a circular cylinder between Reynolds numbers of 200 to 400 when turbulent motion starts to develop in the wake region of the flow. According to Kai Fan Liaw Pressure coefficient occurs a negative value when there is a pressure drop hence velocity is increasing. These statement is very similar with our experiment. According to Kai Fan Liaw work flow around a square cylinder is similar in many ways to the flow around a circular cylinder. Kai Fan Liaw observed the wake region of the flow around a square is wider resulting lower Strouhal number.A similar regular vortex street is observed in the wake region of the flow around a square cylinder at low Reynolds number as for the flow around a circular cylinder.AS Reynolds number increases,flow becomes more turbulent and exhibits three-dimentional nature in the wake region of the flow.Durao et al (6)conducted a detailed experimental investigation of the near wake flow around a square cylinder using laser Doppler velocimetry at a Reynolds number of 14000, when intense velocity fluctuation has been observed in the recirculation region of the flow.Shankar et al(7) employed lES on the simulation of a series of flows around a square cylinder ranging from Reynolds number of a few hundred to 22000.Simulted results Sohankar concluded that Strouhal number was close. Closer agreement of the pressure coefficient with experimental result as well our experiment. Instability in our experiment concluded from point of inflection occurrence. It has been observed that conical diffuser is most simple divices which improve efficiency of the gas turbine by improving pressure rececery co efficient. From Literature review it has been observed that Sovran and Klomp [8] published detailed charts from experimental data of straight-walled conical annular diffusers. In aero space application diffuser is very much essentional device for maximum pressure recovery. It is obtained from literature review that a diffuser with 8^0 divergence angle have maximum pressure recovery in an unswiral straight walled diffuser. Japikse and Baines [9] acknowledged that pressure recovery coefficientsof 0.2–0.5 were tolerated for short diffusers since better performance could not be achieved. Flow around bluff bodies is very important since in the design of long span bridge deck flow pattern is required to study. Kai Fan Liaw investigated in his thesis the flow pattern around bluff bodies and simulated the flow pattern of the bluff bodies like rectangular, cylindrical bodies. In this paper it represents the flow pattern around of bluff in a diffuser. Flow around rectangular square with x/H=1.5 the velocity ratio with varying y/H is matching with our result although in our experiment rectangular bluffs are 110 mm apart. Objective of this paper is to study the flow pattern between two bluffs which is essential to design of high rise tall tower and building. The recirculation zone is studied here which will indicate vortex. Back flow due to adverse pressure gradient creates flow separation. This cause boundary layer separation.

The flow is here turbulent. Reynolds number is of order 8227 to 20153. Objective of study of this experiment is to establish the stability of flow in the wake region and to reduce the vortex shedding. The point of inflection for unstable flow is very very important for this study. In this experiment it is observed the pressure coefficient for the bluff bodies, for structural design pressure coefficient is very important factor.

II. GOVERNING EQUATION

- Navier-Stokes Equations: (conservation of momentum),

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial u_i}{\partial x_j}\right)$$

- The continuity equation: (conservation of mass),

$$\frac{\partial u_i}{\partial x_j} = 0$$

Pressure coefficient:**C_p=** $\frac{P - P_\infty}{\frac{1}{2}V^2\rho}$ Where P=Pressure at any point

P_∞=Pressure at free stream velocity

V=Free stream velocity

ρ =Density of air

Pressure recovery coefficient=C_{rp}= $\frac{P_{stag} - P_{static}}{\frac{1}{2}\rho V^2}$

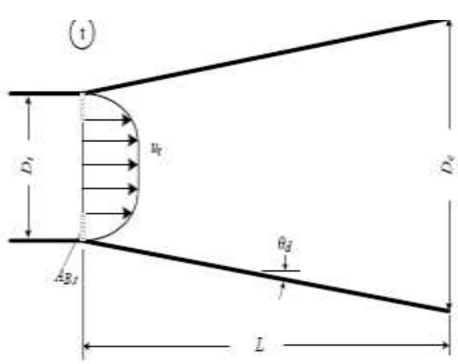
P_{stag} = Stagnation pressure

P_{static} = Static pressure

V=Free stream velocity

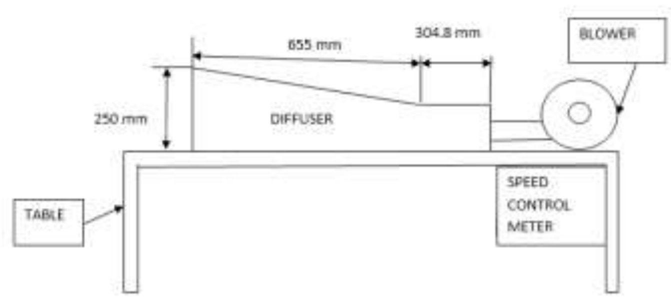
Bernoulli's equation= $\frac{p_1}{\rho g} + \frac{V_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{V^2{}_2}{\rho g} + z_2$

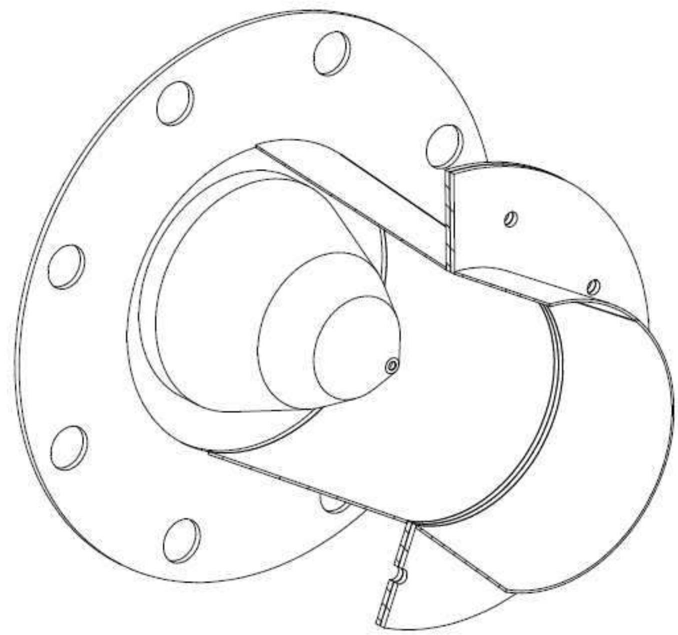
Experimental set up:



Annular Diffuser

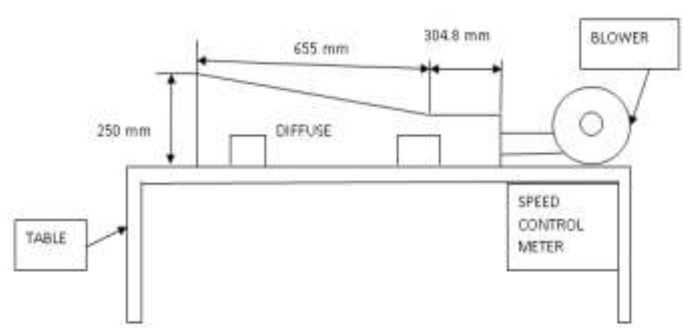
Experimental SET UP of Diffuser





Three dimensional view of Annular diffuser

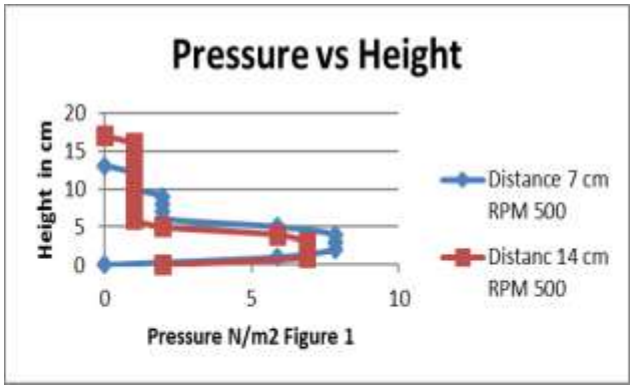
Experimental SET UP of Diffuser with two bluff.

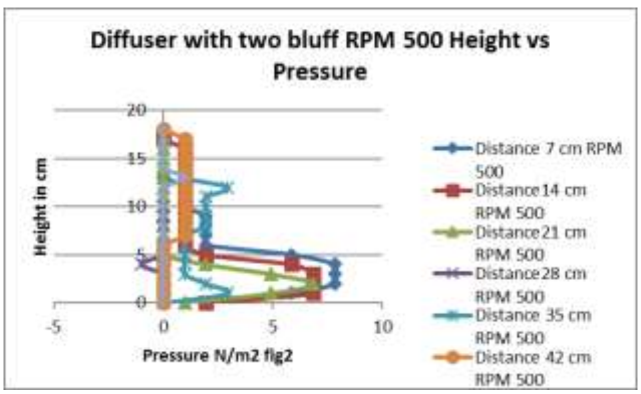


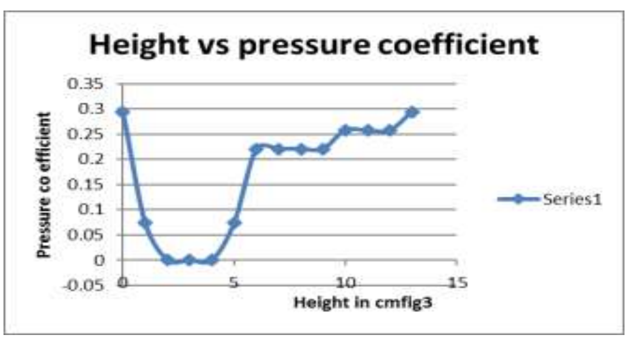
A diffuser 755 mm long is fitted with a blower. Two bluff bodies of size 100 X 100 mm bottom and height 75 mm is placed 110 mm apart. The distance from blower end to the bluff is 310 mm. The blower speeds are 500, 1000 and 1500 RPM. A digital manometer measures the pressure profiles at 8 stations which are at a distance 70mm, 140 mm, 210 mm, 350 mm, 420 mm, 490 mm and and 560mm. The flow is turbulent. Reynolds number range is 8227 to $4X10^5$.

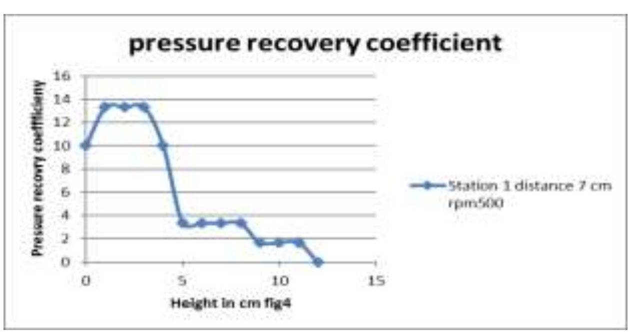
III. RESULTS AND DISCUSSION

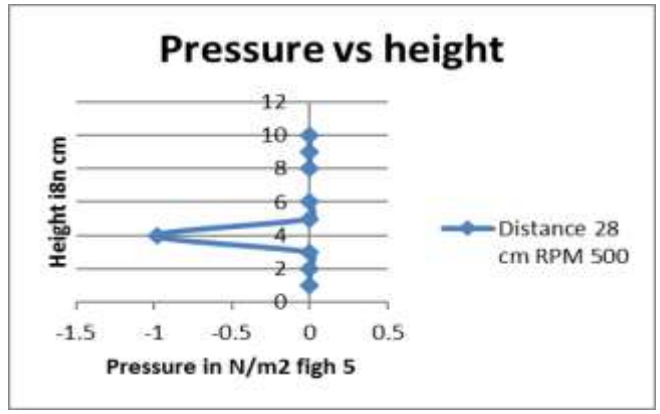
The pressure profile for Reynolds number 8227 to $2X10^4$ has captured. From the graph (fig-1) it is observed that pressure attains a maximum value then reduced a minimum again then it is zero at the top of the diffuser. At a distance 280 mm there occurs negative value of pressure where there is a back flow of fluid. instability occurs due to point of inflection. This shows from fig 2. Distance of the station is 28 cm from the diffuser end.





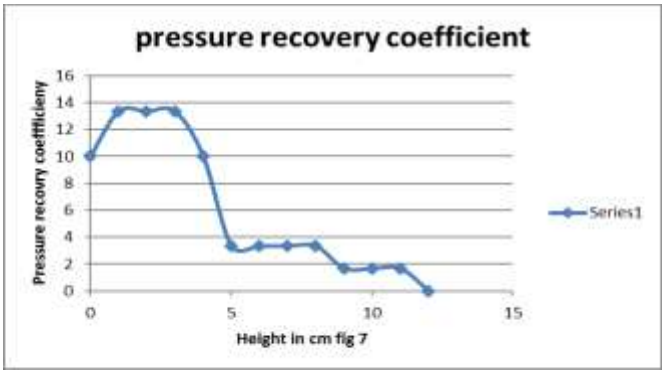


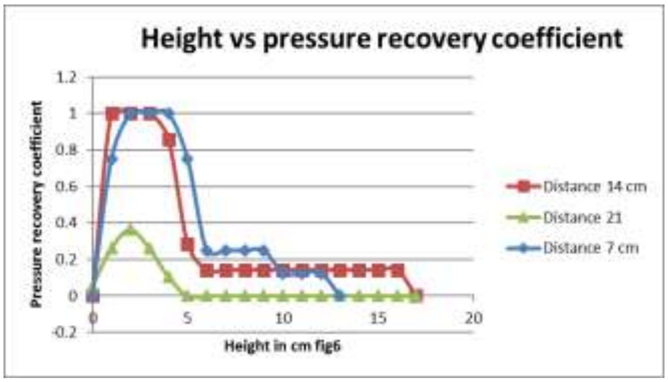


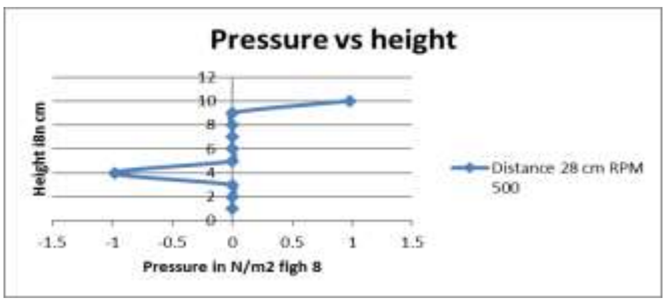


From fig 2 it is observed that here are two bluff bodies hence there is a variation of pressure. Due presence of two bluff there is pressure drop due to gap of two bluff bodies. At out let pressure again attains maximum. As it is a diffuser so pressure is increasing. This shows that there is pressure recovery.(see fig 1) It is more clear from the graph that at station 2 at a distance 140 mm from diffuser end velocity is reduced and pressure is increasing hence pressure recovery occurs.(Figure 1).From figure3 it is clear that pressure is increasing ie there is a pressure recovery in the diffuser. Since velocity is increasing from fig3.From fig 4 for station 1 pressure co efficient is shown. It has been shown pressure coefficient attains two minimum then attains maximum;. It means wind load on tall building decreasing first then increasing as the height is increasing. From the graph (fig3) it is clear that pressure recovery coefficient first increasing and attains maximum then decreases to a minimum as height increases. As pressure recovery is $C_{rp} = \dfrac{p_{st} - p_{static}}{\frac{1}{2}\rho V^2}$ therefore if pressure recovery coefficient decreases than it indicates diffuser has better pressure recovery. Fig 4 shows the pressure recovery for station 1.in the diffuser.

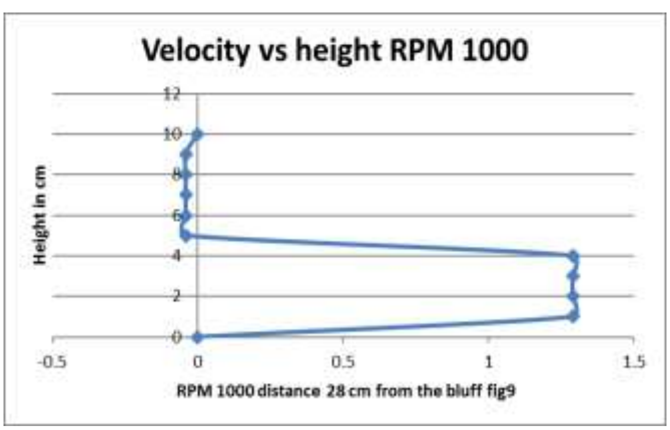
As the pressure recovery coefficient attains a maximum value static pressure recovery minimum since it is in front of bluff bodies. But after that pressure recovery coefficient increasing. Which is better for diffuser. (Fig4). From fig 8 it clear that above bluff there is negative pressure which indicates that there is a recirculation zone. Here there will be flow separation. Here there will occur a point of inflection. This cause instability of flow. From the graph it is clear that there is point inflection for higher speed for same position of the bluff, from fig 9 recirculation occurs below the bluff but in case higher speed recirculation occurs at the edge of the bluff. See fig 9.From the fig 9 it is there is a recirculation zone for there is a boundary layer separation occurs at station 4.

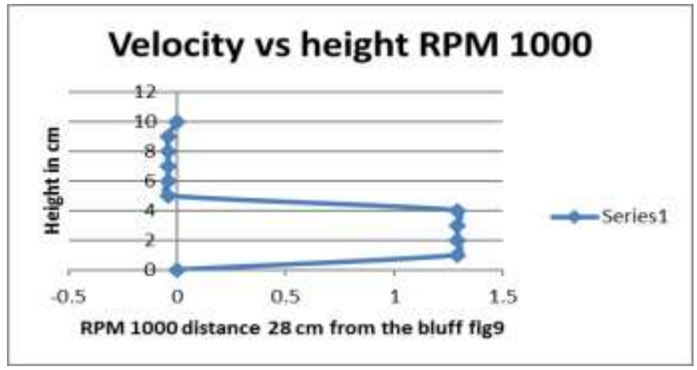


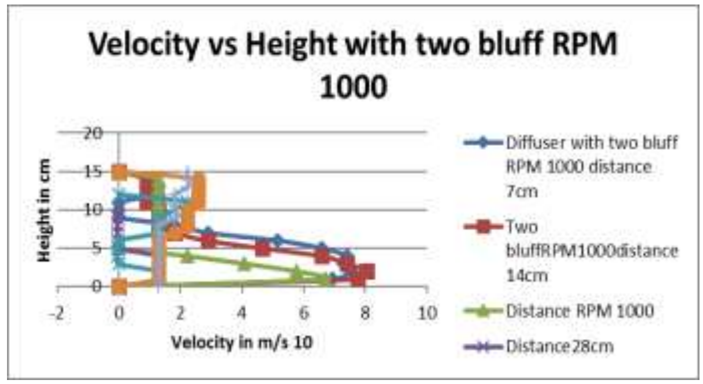




From the experiment it is observed that U/U_∞=0.25 at the mean height at an axial distance ratio 0.93 which is very similar to that Kai Fan Liow thesis. In his experiment when x/H=1 then U/U_∞=0.28. The graph is given here for Kai Fan Liow thesis. Only discrepancy is pressure attains at constant value above the bluff bodies then attains maximum.



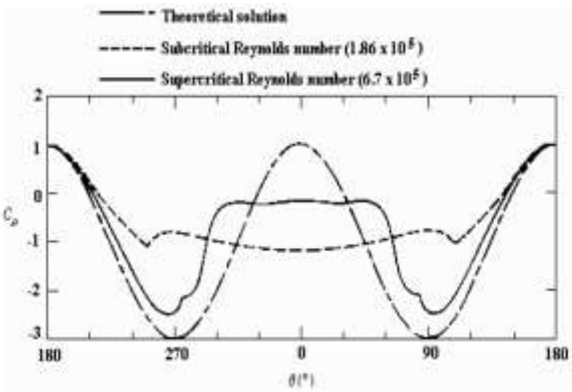




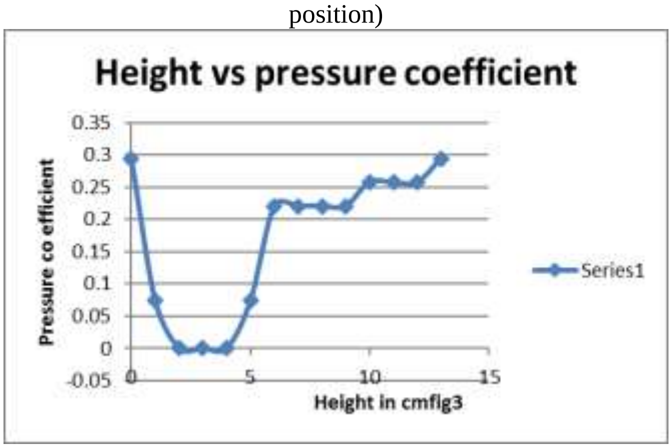
For Reynolds number 8227.47 to 51380 with speed 1000 RPM, from the pressure profile it is observed that pressure attains a maximum then minimum then zero at the wall. For $R_e=2.58 \times 10^5$ to $R_e=1.16 \times 10^6$ and for speed 1500 RPM it attains a flow separation and back flow. It attains a point inflection at a height h=80 mm. For both these speed point of inflection occurs at a distance x=280 mm from the 1st station which is above the bluff. Pressure co-efficient attains a maximum at zero Reynolds number. As the Reynolds number decreases pressure recovery co efficient increases. For low Reynolds number there is a pressure recovery.

Flow pattern between two bluff in a diffuser which is very important from the academic standpoint has observed. It is observed that in the range of Reynolds8227.437 to 47973.79 with blower speed 1000 RPM pressure behind 1st bluff i.e. x=200 mm increasing to a maximum at a height 20 mm from the bottom. It is then zero pressure at a height 80 mm to 110 mm from the bottom plane. After this height pressure is increasing and then occurs a maximum and then zero. This shows that there is a pressure recovery. From Kai Fan Liow's thesis the velocity profile is very similar with this experimental result.

Validation: From Kai Fan Liow's thesis pressure coefficient is similar to our experiment. From David J. Cerantola thesis pressure recovery co efficient is very similar to my paper.More over it has been observed that Pressure recovery coefficient is more negative than From David J. Cerantola's thesis.



Kai Fan Liow's thesis(pressure coefficient vs cylinder position)



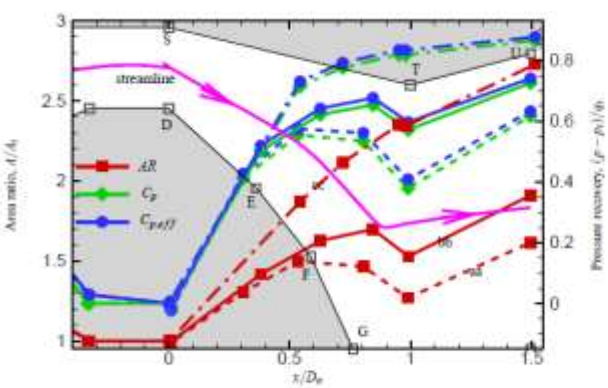
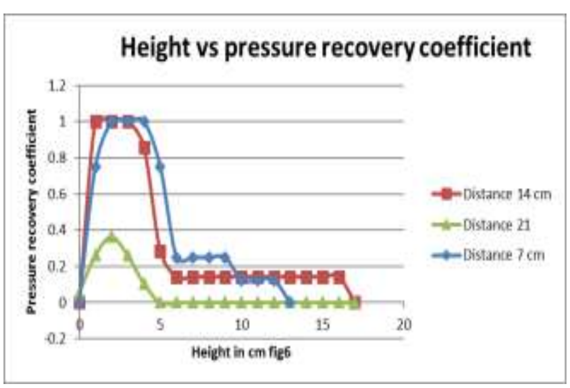


Figure 5.3: Area distributions and pressure coefficients for Configs. aa (dashed), bb (solid), and cc (dash-dot) based on the flow path of an assumed streamline



IV. CONCLUSION

From the above experiment it is observed that in the combination of two bluff in a diffuser between Reynolds Number range 8227 to 2 X 10^4 flow attains a maximum pressure like fully developed flow. But on the bluff there is reversal of flow here there is point of inflections occurs. Which causes flow separation due to adverse pressure gradient $\delta p/ \delta x > 0$ which decelerating the flow at a distance x=280 mm on the 1st bluff. As $\delta p/ \delta x > 0$ hence pressure recovery occurs. Again between two bluffs at a distance x=380 mm from the 1st station between two bluff there is separation of flow. But after the bluff there is no flow separation at x=490 mm and x=560 mm from the 1st bluff reattachment of flow.From graph (fig 3) it is very clear that pressure co efficient attains minimum then increasing ie wind load is increasing.

REFERENCES

[1]. Bloor M S, The transition to turbulence in the wake of a circular cylinder, Journal of Fluid Mechanics, 1964, Vol.19, p. 290-304.

[2]. Tritton D, Experiements on the flow past a circular cylinder at low Reynolds

[3]. Nishioka M, Sato Hiroshi, Measurement of velocity distributions in the wake of a circular cylinder at low Reynolds numbers, Journal of Fluid Mechanics, 1974, Vol.65, p.91-112. numbers, Journal of Fluid Mechanics, 1959, Vol.6, p. 547-567.

[4]. Kai Fan Liaw Simulation of Flow around Bluff Bodies and Bridge Deck Sections using CFD

[5]. Zdravkovich M, Flow Around Circular Cylinders, Fundamental Vol. 1, Oxford

[6]. Durao D G, Heitor M V, Pereira J F, Measurement of turbulent and periodic flowsaround a square cylinder, Experiments in Fluids, 1988, Vol.6, p. 298-304University Press., Oxford, England, 1997.

[7]. Sohankar A, Norberg C, Davidson L, Simulation of three-dimensional flow around a square cylinder at moderate Reynolds numbers, Phys. of Fluids, 1999, Vol.11, (2) p. 288-306

[8]. Sovran, G. and Klomp, E. D., *Fluid Dynamics of Internal Flow*, chap. Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular, Conical, or Annular Cross-Section, Elsevier Publishing, 1967, pp. 270–319. 1, 6, 7, 8, 18, 24, 35, 43, 147, 203, 204, 277

[9]. Japikse, D., "Correlation of Annular Diffuser Performance with Geometry, Swirl, and Blockage," *Proceedings of the 11th NASA/OSU Thermal and Fluids Analysis Workshop*, 2002, pp. 107–118, NASA CP–211486. 15