STM JOURNALS

Journal of Aerospace Engineering & Technology
ISSN: 2231-038X (Online), ISSN: 2348-7887 (Print)
Volume 8, Issue 1
www.stmjournals.com

Numerical Investigation of Macro Flow Dynamics of Downburst Wind Over Buildings and the Interference Effect

***Kalyan Kumar Das*[1,*], *Hirak Ranjan Das*[2]**
[1,2]Department of Mechanical Engineering, Assam Engineering College, Guwahati, Assam, India

Abstract
A "Downburst" is an important weather phenomenon in which an air mass comes down with high velocity towards the ground from the cloud level during a Thunderstorm. Thunderstorms occur frequently in the eastern and Northeastern part of India. Thunderstorms over the Gangetic West Bengal and Assam are popularly known as 'Kalbaishakhi' and 'Bardoisila', respectively. Severe thunderstorms are known to cause detrimental impact on various facets of national activity like civil and defense operations, aviation, space vehicle launching, agriculture in addition to its damage potential to life and properties of human beings and animals. Extreme wind conditions during thunderstorm downburst also cause extensive damages to earth fixed structures and aviation. In this paper an attempt is made to study the macro flow dynamics of downburst wind.

Keywords: *Downburst, streamline pattern, pressure coefficient, building shape, macro flow dynamics*

****Author for Correspondence*** E-mail: kkdas1971@gmail.com

INTRODUCTION

Thunderstorms have always impressed humans and the cumulonimbus cloud is still one of the most visually striking and photogenic of all natural phenomena. However, hazardous weather is associated with thunderstorms. Lightning causes many fires around the world each year and leads to severe injuries or death. Thunderstorms can cause intense rainfall, which can lead to flash flooding and hail larger than a tennis ball. Strong winds associated with thunderstorms can knock down trees and power houses.

Severe thunderstorms are responsible for large amount of wind induced damage around the globe. Unlike large and continental cyclones, severe local storms intensify very rapidly and dissipate after causing damage. The worst severe local storm is the tornado, which is characterized by fast rotating column of rising air which originates on or near the ground where the air swirls and converges at high speed. The downburst is the anti-tornadic storm characterized by slow rotating column of descending air, which bursts out violently after reaching the ground. There are two types of downbursts: wet, which accompanies rain and dry. Fujita (1985) further subdivided downbursts into "microburst", "with damaging wind extending 4 km or less", and "macroburst" with "outburst wind extending more than 4 km in the horizontal direction".

Field experiments are direct method of investigation of the high intensity wind event like downbursts. Programs such as Northen Illinois Meteorological Research on Downbursts (NIMROD), the Joint Airport Weather Studies (JAWS), the Classify and Avoid Wind Shear (CLAWS), the Microburst and Severe Thunderstorms (MIST) are essentially meteorological studies for understanding the formation of downbursts (Alahyari and Longmire, 1995) [1]. However, a full-scale quantitative spatial and temporal characterization of the flow field generated by downbursts near the ground is still lacking. Due to the complexity of the full scale phenomena downburst simulations have been confined to laboratory simulation using impinging jets and simple numerical models of impinging jets. The generic case of an impinging jet is an unsteady flow where a vortex ring forms due to the initial Kelvin-Helmholtz instability. Expansion of the ring can be conceptualized as the time dependent expansion of a circular ripple formed due to a pebble being dropped into still water. This expansion, however, is dependent on the life time and downdraft velocity of the outflow. If the outflow time and downdraft velocity are large, the expansion can continue up to a great distance. But if the outflow time is short and downdraft velocity is small, the life time and radial expansion of the vortex ring is quite small before the vortex ring dissipates all of the downburst's energy.

With computational fluid dynamics (CFD), it is possible to simulate complex wind events and it is relatively easy to alter inlet, outlet and surface conditions. CFD also allows calculations to be made for the influence of various geometric structures on a flow field. The major problem associated with CFD simulation is accurate modeling of the turbulence in the wind as there is little physical data for the turbulence within a true microburst. Physical modeling too has several advantages and disadvantages. One of the major advantages of the physical modeling is the fact that air itself is used for experimentation. This helps to minimize errors, caused due to incorrect modeling of test fluids. Modeling with air as the test fluid has produced relatively good representation of the full scale phenomenon. However, it is very difficult to model the true downburst due to the complexity of the event.

Atmospheric scientist Fujita (1981) [2] has observed and studied the flow due to downburst impacting on the ground and spreading outward in the different directions. He classified downburst as either microburst or macroburst depending on their horizontal extent of damage. For the complexity of the full scale phenomenon, the physical simulation of the downburst is confined to the generic experiments of density currents impinging on a wall. Alahyari and Longmire (1995) [1], Lundgren et al. (1992) [3], Cooper et al. (1993) [4], Didden and Ho (1985) [5], Knowles and Myszko (1998) [6] have studied experimental simulation of the downburst. Letchford and Chay (2002) [7], Chay and Letchford (2002) [8] and Sengupta and Sarkar (2007) [9] investigated downburst velocity profiles using the physical modeling of the event. Numerical simulation of the downburst is performed by many researcher Proctor (1988) [10], Craft et al. (1993) [11] and Selvam and Homes (1992) [12], Homes and Oliver (2000) [13], Kim and Hangan (2006) [14], Mason et al. (2009, 2010) [15,16], Chay et al. (2006), Sengupta and Sarkar (2007) [10] and Chen and Letchford (2006) [17]. Chen and Letchford (2006) [17] have done multiscale correlation analyses of two lateral profiles of full scale downburst.

The primary objective of the present work is to investigate the effect of downburst wind on buildings with various geometrical shapes. A downburst (microburst) near Guwahati, India is detected by the authors on 10th September, 2010 during the present study and is shown in Figure 1.

Fig. 1: *Photographs by the Author of a Microburst on 10th September 2010 near Guwahati, India.*

MATHEMATICAL MODELING USING ANSYS

Impinging jet model is used to simulate the downburst wind numerically.

Computational Domain for Flow Over Single Building

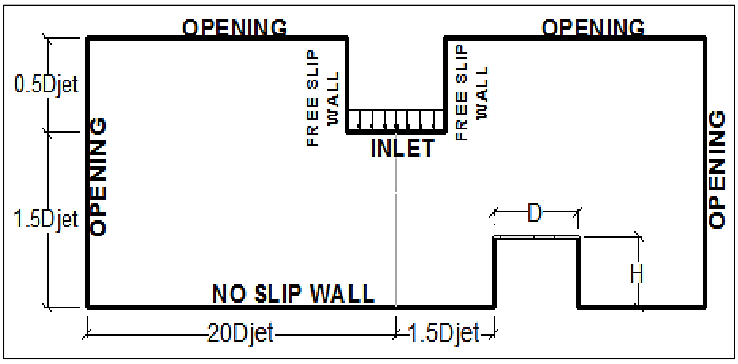
The numerical model is developed using ANSYS CFX 16. Figure 2 shows the computational domain and Figure 3 shows the downburst vertical wind profile. The depth (D) and width (W) are considered as 5% of D_{jet}, where D_{jet} is the diameter of the impinging jet. Height to Diameter ratio for all the buildings is taken as 4 and investigation has been done for different building shapes such as square prism, circular prism and elliptical prism with major axis along and normal to wind flow direction.

Computational Domain for Flow Over Two Buildings

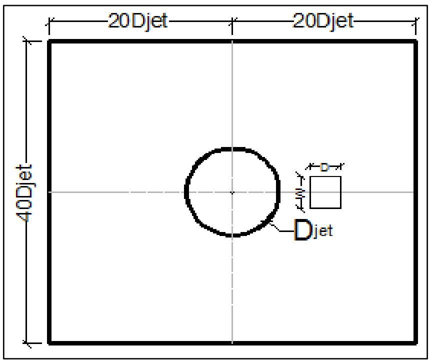
Numerical model to study the interference between buildings has been developed using ANSYS CFX 15 as shown in Figure 4. Where depth (D) and width (H) are considered as 5% of Djet (Diameter of the jet) Whereas H/D ratio is taken as 1 and 2.

Meshing

Though the domain is so simple, its complicated to generate mesh so that the boundary functions will work properly. To generate mesh for each case were generated using the ANSYS ICEM software to get better results. The mesh generated can be seen in the Figures 4 and 5.



(a) Side view



(b) Top view

Fig. 2: *Computational Domain for Downburst Wind.*

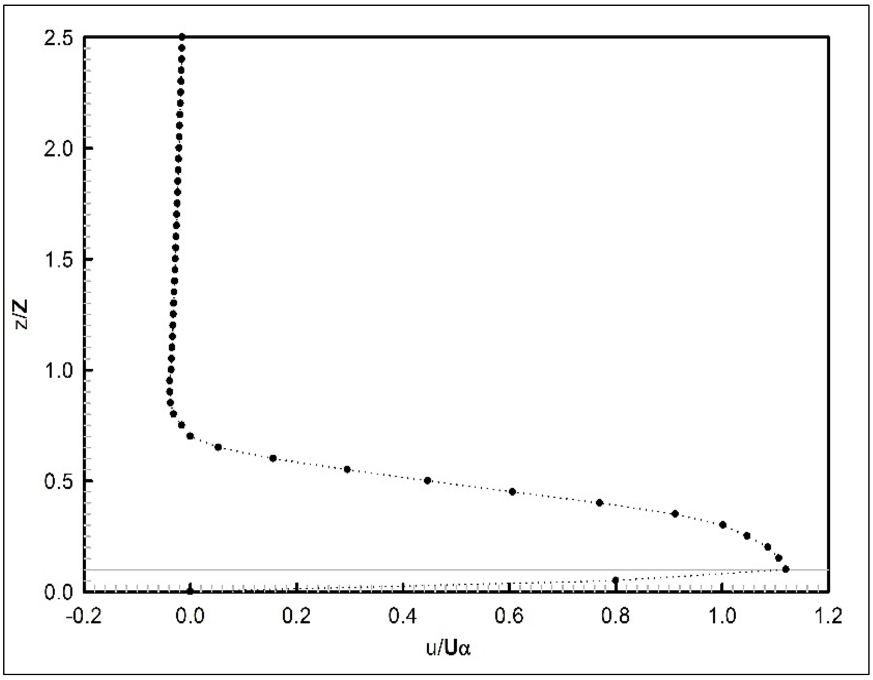
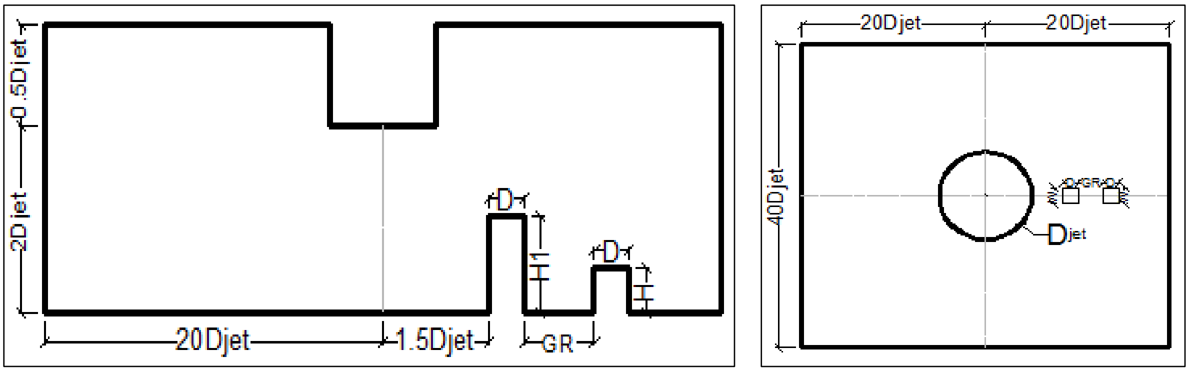


Fig. 3: *Velocity Profile.*



(a) Section at ZX Plan (b) Section at XY Plan

Fig. 4: *Mesh Generation Using ANSYS ICEM Software (at ZX and XY Plan).*

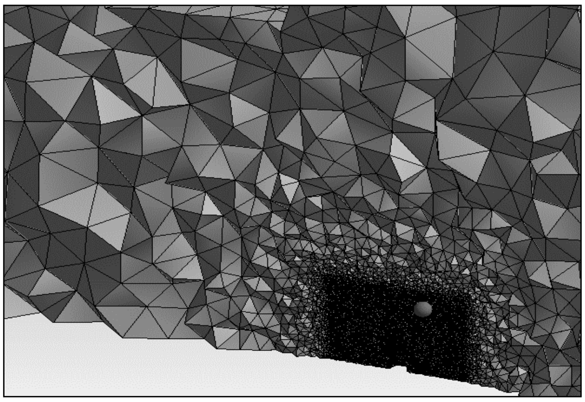
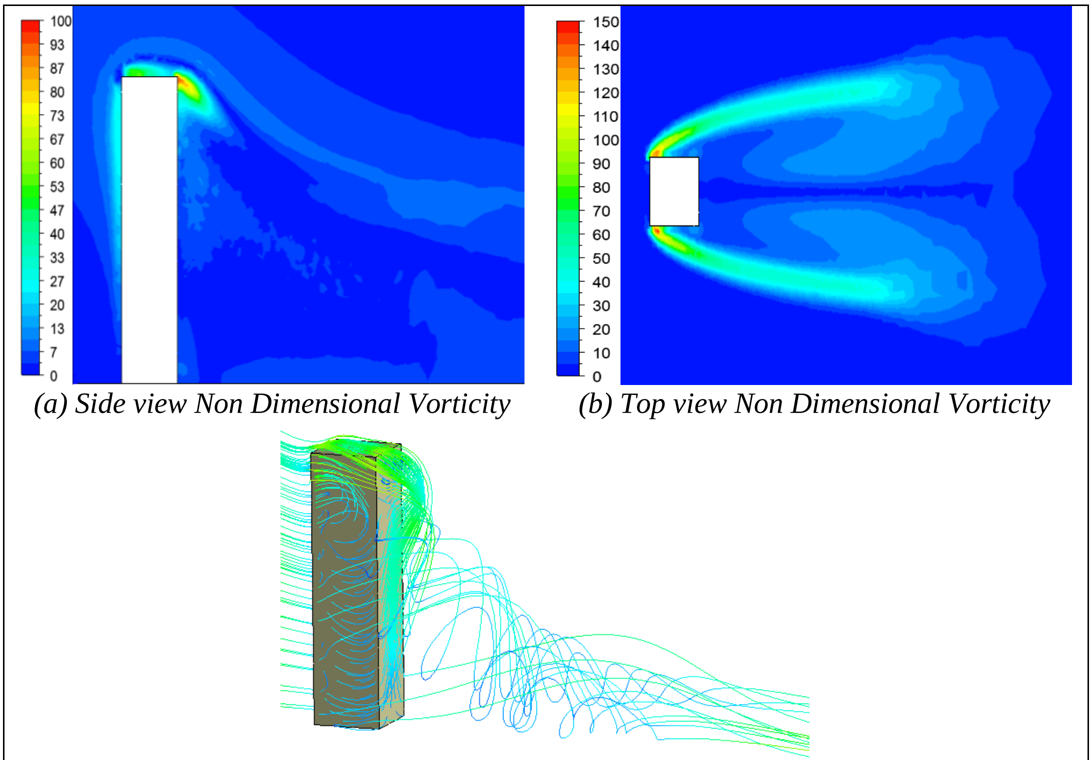


Fig. 5: *Mesh Generation Using ANSYS ICEM Software.*

RESULTS AND DISCUSSIONS



(a) Side view Non Dimensional Vorticity *(b) Top view Non Dimensional Vorticity*

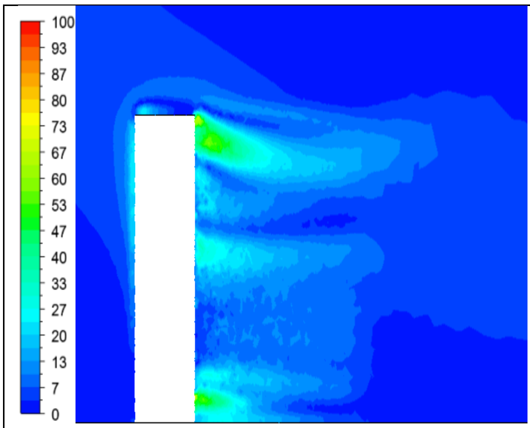
(c) 3D Stream Lines

Fig. 6: *Square Prism.*

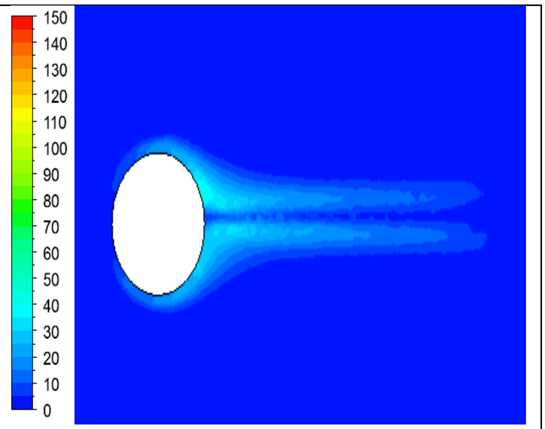
Figures 6(a) and (b) show the vortex formation around the building surface. Vortices of high vorticity with vortex stretching downstream are observed as the flow turns around two sharp corners on the windward side. A symmetrical vortex pattern is observed in the top view (Figure 6(b)). Vortices of comparatively lower intensity are observed to form at the corners on the leeward side Figure 6(a). The 3-D streamline picture (Figure 6(c)) shows complicated flow pattern involving flow curvature, curling and recirculation downstream the building.

The vortex formation around a circular building can be seen from Figures 7(a) and (b). The difference in vorticity pattern can be clearly observed. The absence of sharp corner facing the wind flow makes all the difference. The vortices are not as strong as those for the prismatic building with sharp corner. The vortices are not stretched far enough downstream. The streamline pattern Figure 7(c) shows that the streamlines follow the body shape upstream with complicated flow recirculation and vorticity pattern in the wake. Figures 8(a) and 8(b) show the vortex pattern for an elliptic building with major axis. In this case, being a streamlined body the flow streamlines follow most part of the body without any flow detachment. The wake is characterized by flow recirculation, streamlines cris-crossing each other and formation of vortex sheet. Figure 8(c) shows vortex pattern attached to the top of the building.

Figures 9(a) and 9(b) show vorticity pattern over an elliptic building with major axis normal to the flow. Figure 9(a) shows a curved vortex pattern. In this case wake is wide and Figure 9(b) shows emergence of conical vortex lines from the top and bottom of the building. Complicated streamline pattern in the wake involving flow curvature, recirculation and streamline interaction is clearly evident.



(a) Side View Non Dimensional Vorticity.



(b) Top View Non Dimensional Vorticity.

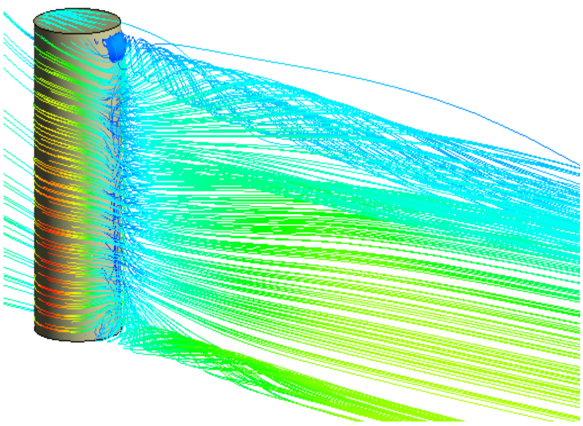
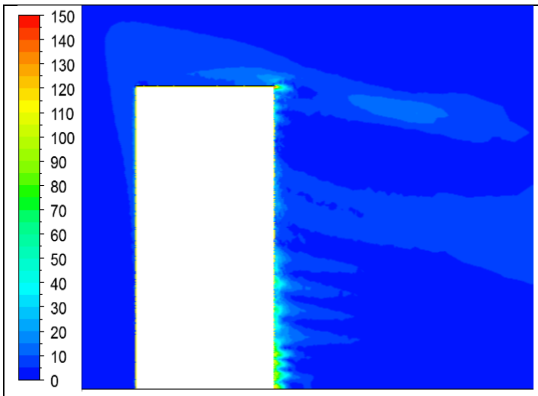
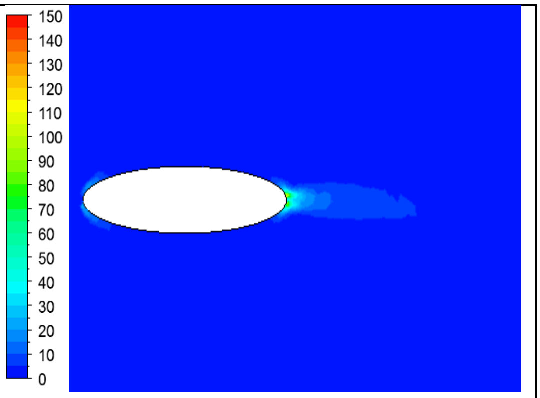

(c) 3D Stream Lines

Fig. 7: *Circular Prism.*



(a) Side view Non Dimensional Vorticity



(b) Top View Non Dimensional Vorticity

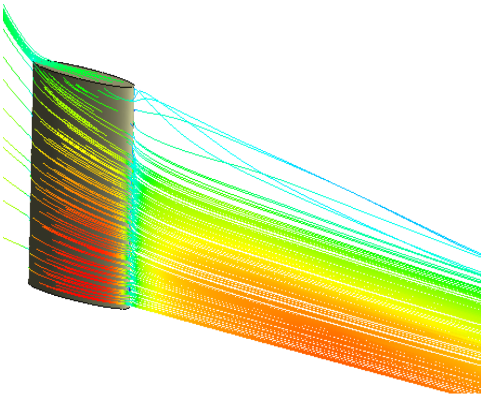
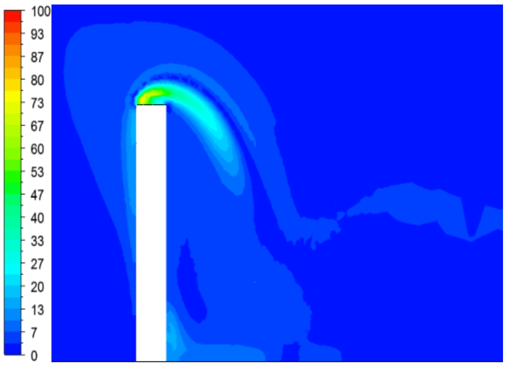
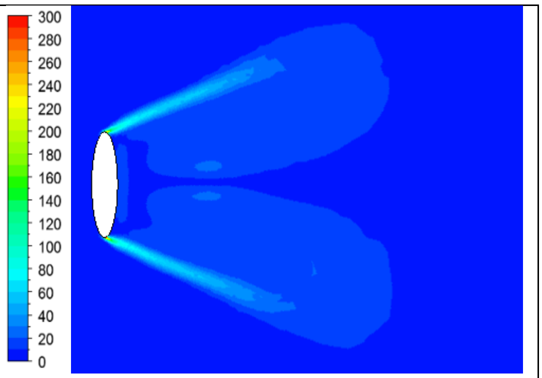

(c) 3D Stream Lines

Fig. 8: *Elliptic Prism- major Axis Parallel to the Flow.*



(a) Side View Non Dimensional Vorticity



(b) Top View Non Dimensional Vorticity

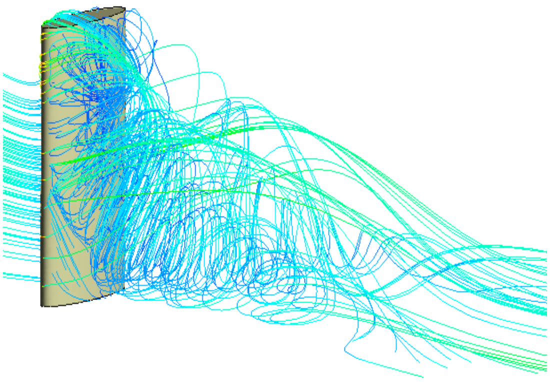
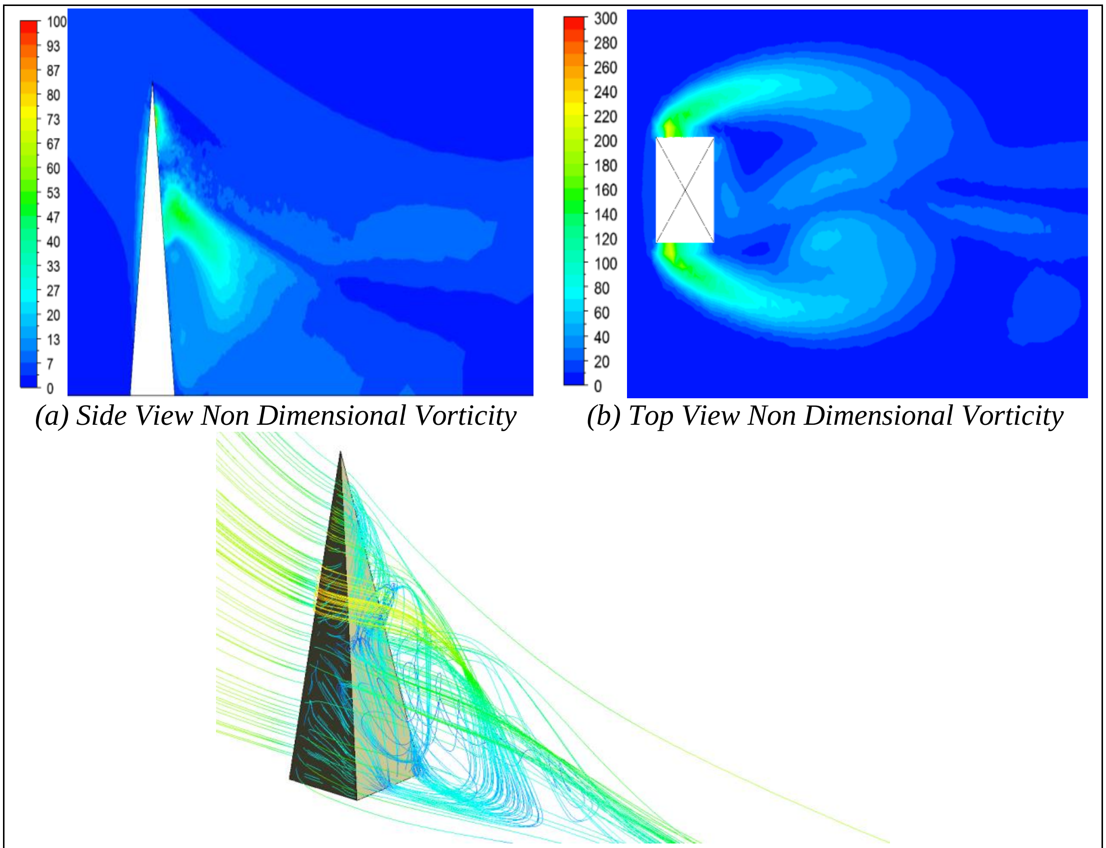

(c) 3D Stream Lines

Fig. 9: *Circular Prism Major Axis Normal to the Flow.*



(a) Side View Non Dimensional Vorticity *(b) Top View Non Dimensional Vorticity*

(c) 3D Stream Lines

Fig. 10: *Square Pyramid One Face Normal to the Flow.*

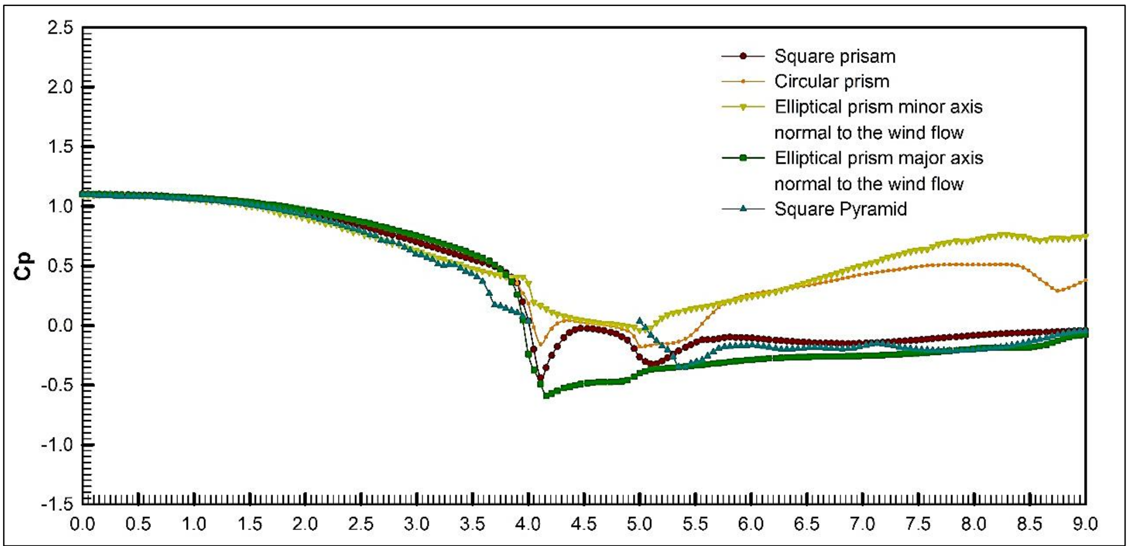


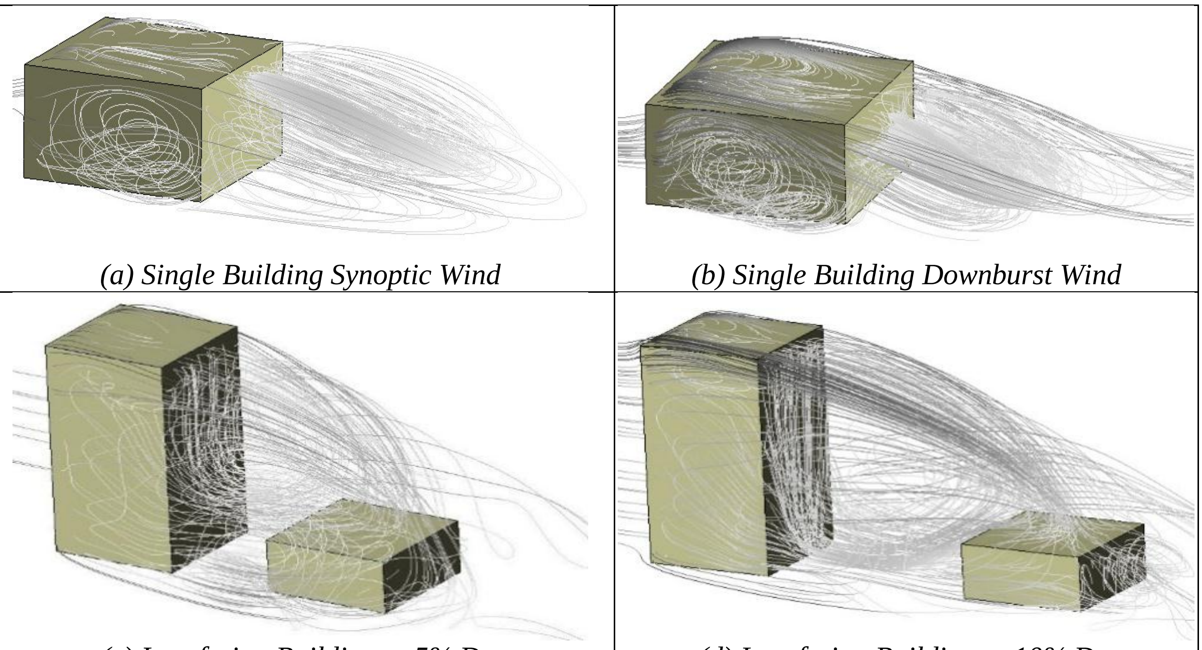
Fig. 11: *Cp Distribution along the Surface.*

Figure 11 shows the Cp distribution for all the building shapes considered in this study. In this case Cp is defined as:

$$Cp = (p - p_{\infty})/(0.5\rho_{\infty}U^{2}_{\infty})$$

For all the building shapes Cp drops from high positive value (nearly 1.0) to a lower positive value on the windward face. High positive Cp near windward corner is due to typical downburst profile, which shows high velocity near the ground and possible location of stagnation point near the corner of the windward side. Pressure drops on face roof due to flow separation. It is significant that for elliptic building with major axis parallel to the wind flow, Cp remains positive over almost the entire surface of the building except near the corner point, where pressure falls to nearly zero. For the circular prismatic building Cp fluctuations between positive and negative values on windward and leeward faces are due to velocity variations on these surfaces.

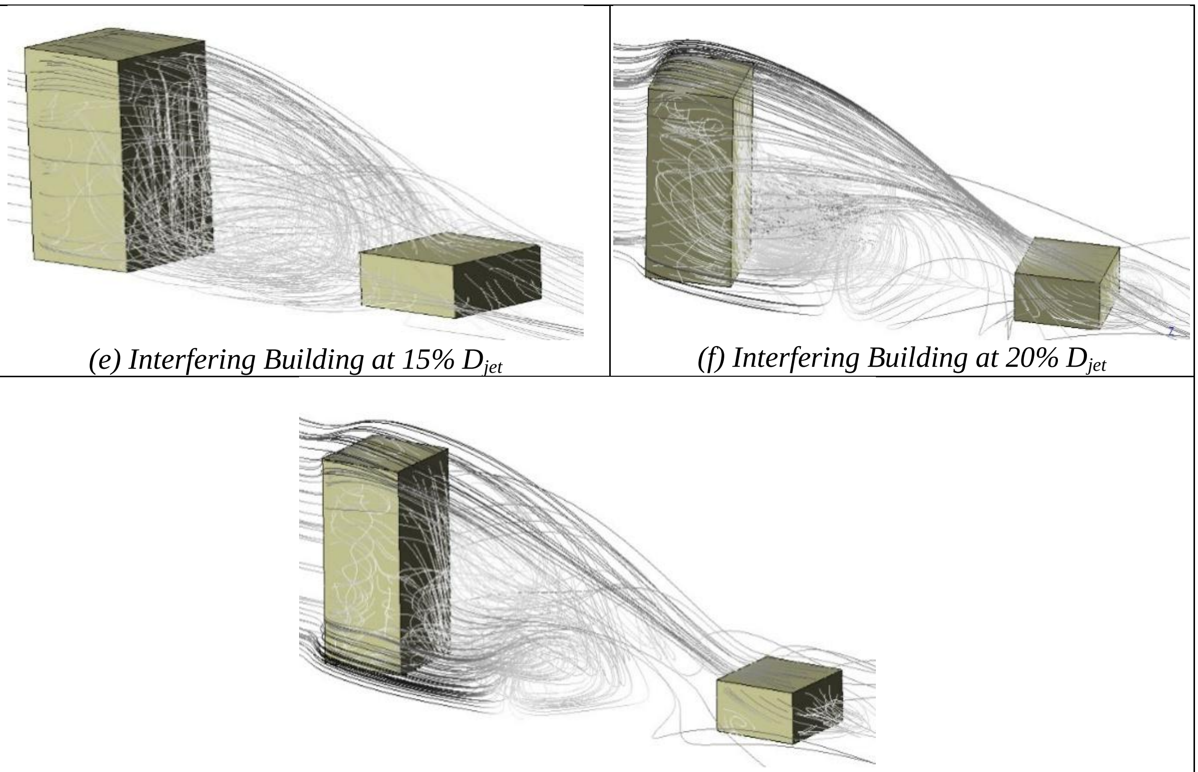
Figures 12, 13 and 14 show the flow dynamics for multiple buildings subjected to downburst wind loading.



(a) Single Building Synoptic Wind *(b) Single Building Downburst Wind*

(c) Interfering Building at 5% D_{jet} *(d) Interfering Building at 10%* D_{jet}

Fig. 12: *Interference Effect.*

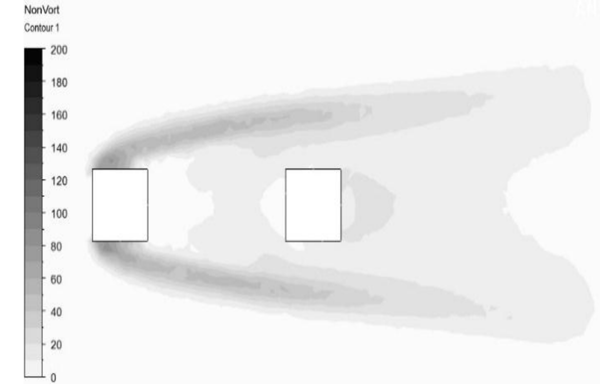


(e) Interfering Building at 15% D_{jet} *(f) Interfering Building at 20%* D_{jet}

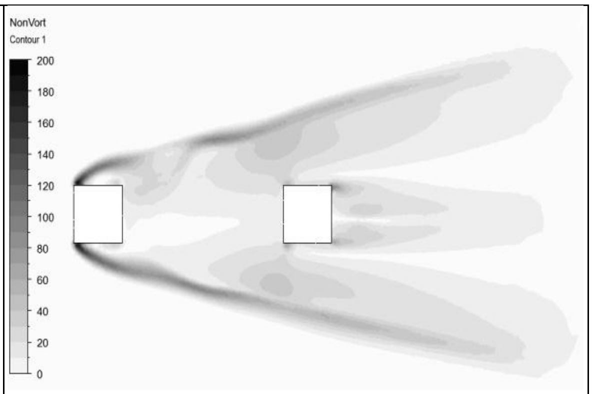
(g) Interfering Building at 25% D_{jet}

Fig. 13: *Interference Effect.*

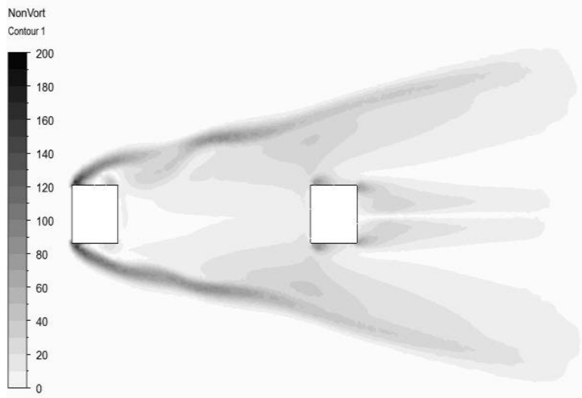
Journal of Aerospace Engineering & Technology
Volume 8, Issue 1
ISSN: 2231-038X (Online), ISSN: 2348-7887 (Print)



(e) Interfering Building at 15% D_{jet}



(f) Interfering Building at 20% D_{jet}



(g) Interfering Building at 25% D_{jet}

Fig. 14: *Interference Effect.*

CONCLUSIONS

The model has been used extensively to study wind flow behavior on buildings and their interference effect. The results have been presented in terms of 3D streamline plots, nondimensional vorticity contour and pressure coefficient (C_P) on the building surfaces. The streamline plot and nondimensional-vorticity plot show that the building shape has significant effect on flow characteristics and also on the coefficient of pressure distribution on the buildings.

REFERENCES

1. Alahyari A, Longmire EK. Dynamics of experimentally simulated microbursts. *AIAA J.* 1995; 33 (11): 2128–2136p.
2. Fujita TT. Tornadoes and downbursts in the context of generalized planetary scales. *J Atmos Sci*. 1981; 38: 1511–1534p.
3. Lundgren TS, Yao J, Mansour NN. Microburst modeling and scaling. *J Fluid Mech*. 1992; 239: 461–488p.
4. Cooper D, Jackson DC, Launder BE, *et al.* Impinging jet studies for turbulence model. 1993.
5. Didden N, Ho CM. Unsteady separation in a boundary layer produced by an impinging jet. *J Fluid Mech.* 1985; 160: 235–256p.
6. Knowles K, Myszko M. Turbulence measurement in radial wall-jets. *Exp Thermal Fluid Sci*. 1998; 17: 71–78p.
7. Letchford CW, Chay MT. Pressure distributions on a cube in a simulated thunderstorm downburst, Part B: moving downburst observations. *J Wind Eng Ind Aerodyn*. 2002; 90: 733–753p.
8. Chay MT, Letchford CW. Pressure distribution on a cube in a simulated thunderstorm downburst—Part A: stationary downburst simulation. *J Wind Eng Ind Aerodyn*. 2002; 90: 711–732p.
9. Sengupta A, Sarskar PP. Experimental measurement and numerical simulation of an impinging jet with application to thunderstorm microburst winds. *J Wind Eng Ind Aerodyn.* 2007.
10. Proctor FH. Numerical simulations of an isolated microburst. Part I: Dynamics and Structure. *J Atmos Sci*. 1988; 45: 3137–3160p.

11. Craft TJ, Graham LJW, Launder BE. Impinging jet studies for turbulence model Assessment-II: an examination of the performance of four turbulence models. *Int J Heat Mass Transfer*. 1993; 36(10): 2685–2697p.
12. Selvam RP, Holmes JD. Numerical simulation of thunderstorm downdrafts. *J Wind Eng Ind Aerodyn*. 1992; 41–44: 2817–2825p.
13. Holmes JD, Oliver SE. An empirical model of a downburst. *Eng Struct*. 2000; 22: 1167–1172p.
14. Kim J, Hangan H. Numerical simulation of impinging jets with application to downbursts. *J Wind Eng Ind Aerodyn*. 2007; 95: 279–298p.
15. Mason MS, Wood GS, Fletcher DF. Numerical Simulation of Downburst winds, *J Wind Eng Ind Aerodyn*. 2009; 97: 523–539p.
16. Mason MS, Fletcher DF, Wood GS. Numerical Simulation of idealized three dimensional Downburst wind fields, *J Wind Eng Ind Aerodyn*. 2010; 32(11): 3558–3570p.
17. Chen L, Letchford CW. Multi-scale correlation analyses of two lateral profiles of full scale downburst wind speed. *J Wind Eng Ind Aerodyn*. 2006; 94(9): 675–696p.
18. Assessment-I. Flow-field experiments. *Int J Heat Mass Transfer*. 1993; 36(10): 2675–2684p.
19. Fletcher CAJ. *Computational Techniques for the Fluid Dynamics*. Springer-Verlag Publication, 1987; 2.
20. Hjelmfelt MR. Structure and life cycle of micoburst outflows observed in Colorado. *J Appl Met*. 1988; 27: 900–927p.
21. Parsons DB, Weisman ML. A Numerical study of a Rotating Downburst, *J Atmospheric Sci*. 1993; 50(15): 2369–2385p.
22. Sakamota S, Murakami S, Mochida A. Numerical study on flow past 2D square cylinder by Large Eddy Simulation Comparison between 2D and 3D computations, *J Wind Eng Ind Aerodyn*. 1993; 50: 61–68p.

Cite this Article

Kalyan Kumar Das, Hirak Ranjan Das. Numerical Investigation of Macro Flow Dynamics of Downburst Wind over Buildings and the Interference Effect. *Journal of Aerospace Engineering & Technology*. 2018; 8(1): 11–20p.