CFD ANALYSIS OF POLLUTION DISPERSION IN COMPLEX DOMAIN

Mr. Rahul Kumar[1], Mr. Devesh Kumar Singh[2], Mrs. Priyanka Jhavar[3], Prashant Singh[4]

[1]PG Scholar, Department of Mechanical Engineering, SSSIST Sehore, M.P., India
[2] PG Scholar, Department of Mechanical Engineering, BU Ajmer, Rajasthan, India
[3] Associate Prof., Department of Mechanical Engineering, SSSIST Sehore, M.P., India
[4] Assistant Prof., Department of Aeronautical Engineering, SSSIST Sehore, M.P., India

ABSTRACT

Over past few decades many CFD studies and model studies has been done to analyze the pollution dispersion in urban area. CFD simulation for pollution dispersion is gaining popularity because of increased computational capability. Two-dimensional simulation of pollution dispersion has been carried out in the present study. The computational domain is a two dimensional hill and a building in its downwind direction. The detailed study of different shapes of building and hills, effect of distance between hill and building are done. The attempt is made here to find the optimum distance from pollutant source so that building will face minimum level of pollution

Air quality is the major concern in the urban area. Urban areas are facing the severe problem of health issues and premature deaths due to increased level of pollutants because of traffic and industrial activities. Increasing number of industries and vehicles in urban area deteriorate the quality of air continuously.

Keyword: - *Ansys 14.0, Hill and Building Model, Reynolds Number, Pollutants.*

1. INTRODUCTION

Atmospheric phenomenon involves large range of scales in both space and time. It depends upon variety of meteorological elements such as turbulence, buoyancy, topographic effect and rotation of the Earth [3]. Understanding and predicting the regional distribution of meteorological variables, such as wind velocity, turbulence, temperature, humidity of these phenomena are quite necessary because they have strong relationships with flow field. Industrial planning and environmental problems have led to increased research in order to assist regulators and urban planners to improve air quality in cities.

Air quality in urban areas has gained increasing interest in recent years due to its significant influence in human health. The degradation of the indoor air quality by re-ingestion cause's respiratory diseases, heart and brain damage to the occupants of the building, four major tools are used to investigate the atmospheric phenomena.

1. Field study
2. Theoretical study
3. Experimental study
4. Numerical study

Urban areas are facing the severe problem of health issue and premature deaths due to increased level of pollutants due to traffic and industrial activities. Traffic related pollutants are CO, NOX and unburned hydrocarbon. Atmospheric pollutants cause both long term and short term effects on health of humans and environment. CO cause short term health impact and benzene cause cumulative long term health effect. NOX cause both long term and short term health issues. Long term exposure to high benzene level can cause cancer.

CFD simulation for pollution dispersion is gaining popularity because of increased computational capability. Over past few decades many CFD studies, model studies and experimental studies has been done to analyze the pollution dispersion in urban area. The prediction of pollution dispersion in urban environment is very tedious task due to interaction of flow with the structures. Analytical models, Lee & Park [4], are also developed based on simple flow and dispersion model.

2. COMPUTATIONAL DOMAIN

The computational domain is a two dimensional hill and a building in the wake area of the hill. The mathematical formulation of cosine hill is given by $y = H\ Sin^2(x_1\pi/2L)$. The computational domain is 60H long in stream wise direction and 22.5H high in transverse direction. The computational domain is chosen to be long so that the results do not have numerical errors due to boundary

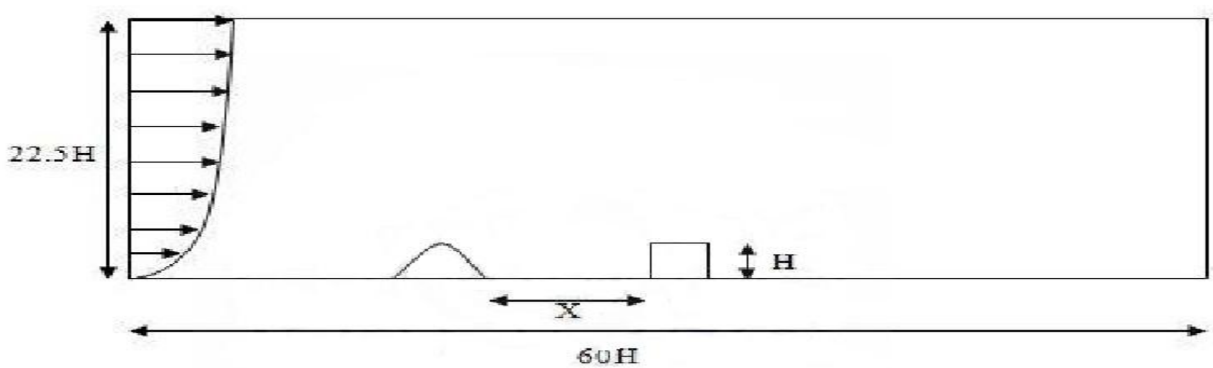

Fig - 1: Computational domain schematic; building height H at distance X from the end of two-dimensional hill of height H

The hill is 7H long for $y = H\ Sin^2(x_1\pi/2L)$ in stream wise direction. The height of the building is H for all cases. The computational cases are shown in Table 1.

The pollutant is coming from the inlet into the domain. The source of the pollutant is not shown in the computational domain. It is considered at some distance in the upwind direction of the inlet boundary. Although the pollutant plumes move in the shape of vortex but in present study average of the pollutant vortex plume is considered. The amount of pollutant entering in to the terrain is same as the case of plume. It is modeled as a constant source of pollutant from 0.75H to 1.25H with non-dimensional concentration value 1.0. Thus the non dimensional value of the pollutant at the inlet boundary is 1.0 at the vertical height of 0.75H to 1.25H. The distribution of pollutant in the domain is depends upon the flow field. The pollutant spread with the flow field and come in contact with hill and building. The study of pollutant dispersion for different location and heights of building with respect to the hill is done in the present study.

One building shapes are studied in the present study. The building shapes, which studied, are a square of length H. Distance X is the distance from end of the hill to the start of the building in the downstream direction.

There are three cases for each building height, Building at X=H, X=3H, and X=7H. Thus there are total three cases, which are studied, in the present study. As the distance X changes, different flow pattern is generated because interaction between flows field of hill and building changes. Effect of hill and building interaction is studied by taking various values of X.

The pollutant dispersion in the domain is highly dependent on the recirculation zones and reattachment lengths of the flow field. The reattachment length of flow is increased when building is present in downwind of the hill. The range of variation of X remains up to 7H from hill end to building starting point to capture the reattachment length. The geometrical information of these cases is shown below in Table 1.

Table 1: Geometrical information of studied computation cases

Mathematical Function of Hill		**Building Location (X)**	**Building Height**
Slope			
$y = H\ Sin^2 (x_1\pi/2L)$	a	H	H
	b	3H	H
	c	7H	H

3. RESULTS AND DISCUSSION

Result and discussion has been divided in three parts based on the building height. The first section has the result of $y = H\ Sin2\ (x_1\pi/2L)$ with a square building of height and width H at positions X=1, 3 and 7 with respect to the hill end. The width of the building is H in all cases. All the results are calculated at the Reynolds Number 13800. The square building of height H is placed at X= H, 3H and 7H from the hill end. In the present section the result of hill function $y = H\ Sin^2 (x_1\pi/2L)$ is studied with different building location. The stream wise velocity contours with the streamlines are shown in figure 2.

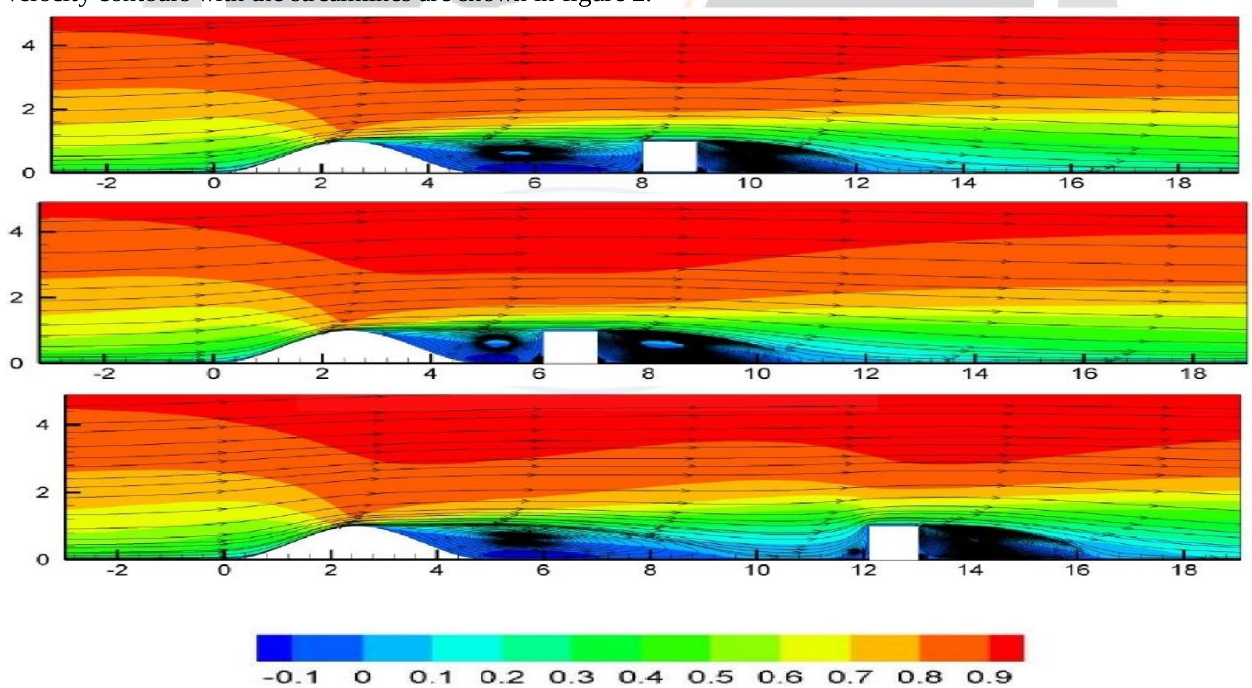

Fig -2:- Stream wise velocity contours of flow field for building height H. The building is placed at distance X from the hill end; (a) X= H, (b) X= 3H, and (c) X= 7H.

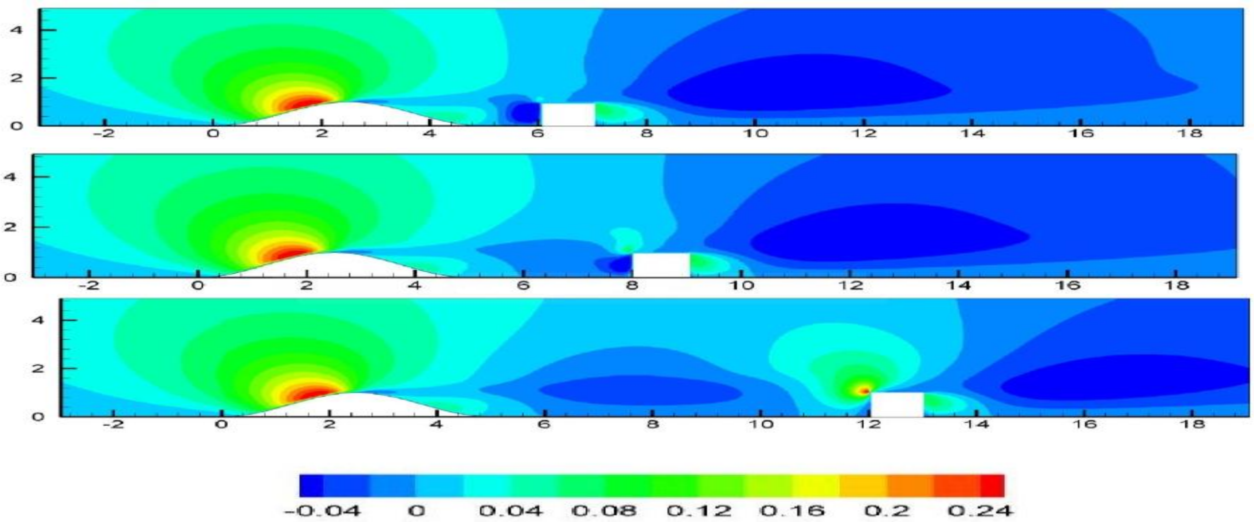

Figure 3: Transverse velocity contours of flow field for building height H. The building is placed at distance X from the hill end; (a) X= H, (b) X= 3H and (c) X= 7H.

The pollutant concentration in any terrain is driven by flow field. The pollutant removal depends upon convection and diffusion. Transverse and stream wise velocity advents the pollutant particle to the fresh air with itself. Turbulent flows are highly diffusive flows; the other mechanism of pollutant removal is turbulent diffusion which diffuses the pollutant to the fresh air. Figure 2 and Figure 3 shows the stream wise and transverse velocity contour.

The turbulent kinetic energy contours and turbulent viscosity contours are shown in Figure 4 and Figure 5. There concentration contours are shown in Figure 6.

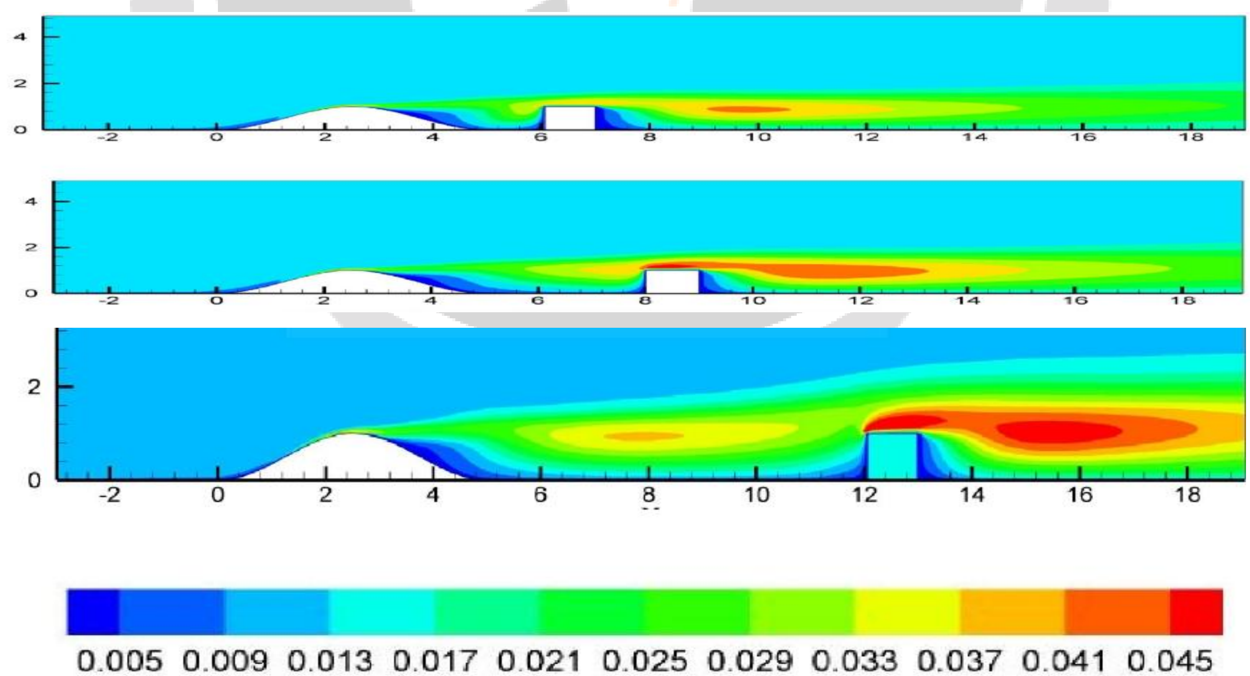

Figure 4: Turbulent kinetic energy contours of flow field for building height H. The building is placed at distance X from the hill end; (a) X= H, (b) X= 3H and (c) X= 7H.

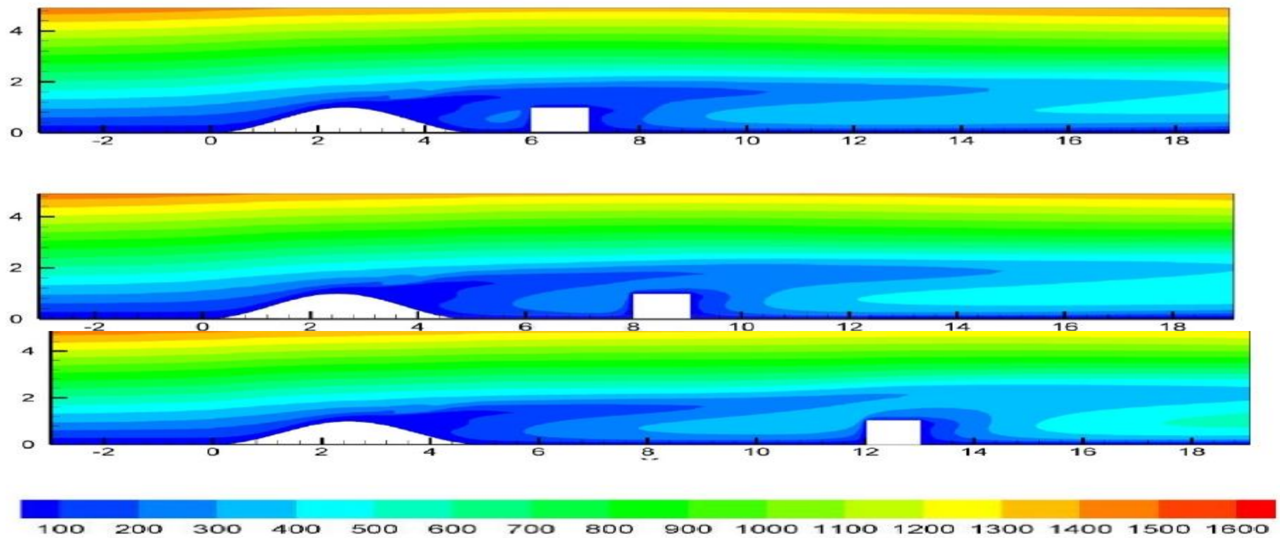

Figure 5: Turbulent viscosity contours of flow field for building height H. The building is placed at distance X from the hill end; (a) X= H, (b) X= 3H and (c) X= 7H.

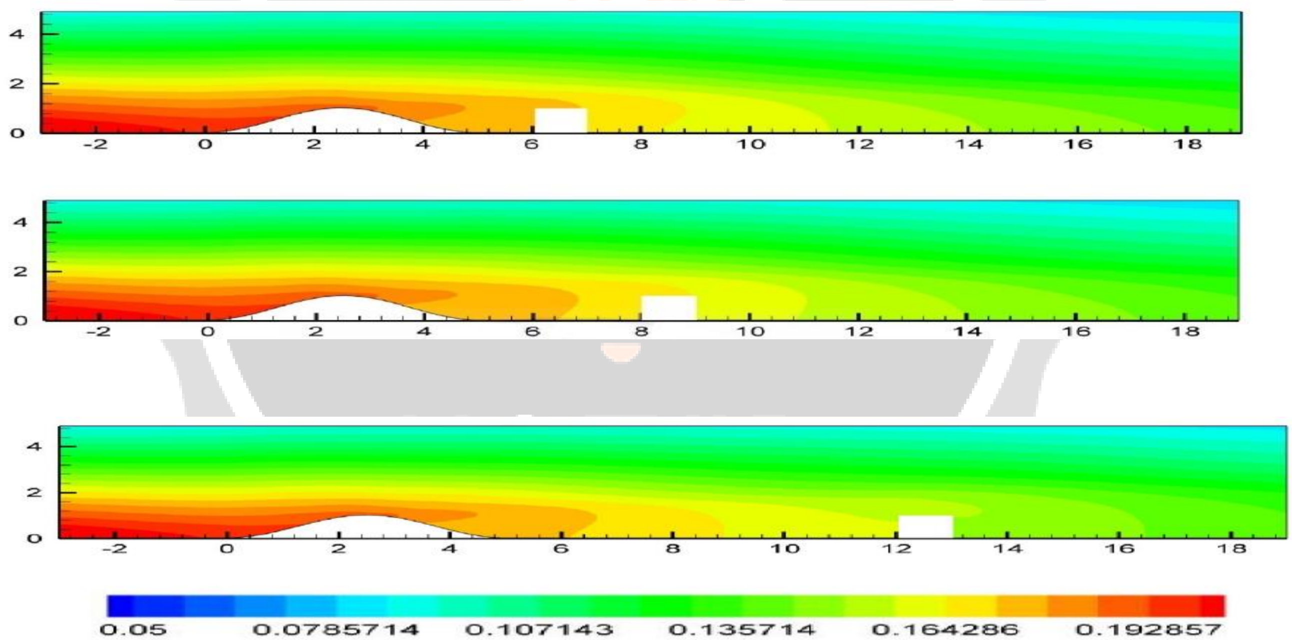

Figure 6: Pollutant concentration contours for building height H. The building is placed at distance X from the hill end; (a) X= H, (b) X= 3H and (c) X= 7H

The turbulent kinetic energy has maximum values at the corner of the building and at the upwind side of the hill crest because there is highest velocity shear at these locations. Production is high at these points which results high turbulent kinetic energy at these locations. The shear layer profile can be understood by seeing the streamlines. The value of turbulent kinetic energy is high in shear layer. The spread of high turbulent kinetic energy zone in the downwind of leeward wall increases as the distance X increases because of increase in reattachment length in the downwind of the building.

The turbulent viscosity depends upon the mean dissipation rate and turbulent kinetic energy. It has high values near the windward walls due to high turbulent kinetic energy in this area.

On seeing the results of concentration profile carefully the concentration at the windward wall is decreasing for increment in distance X. The leeward wall also has the same nature with the distance X and concentration. The plot of wall concentration with wall height for windward wall and leeward wall are shown in Figure 7 and Figure 8 respectively.

The change in concentration at the windward wall is 12% when Case (c) concentration is taken as reference.

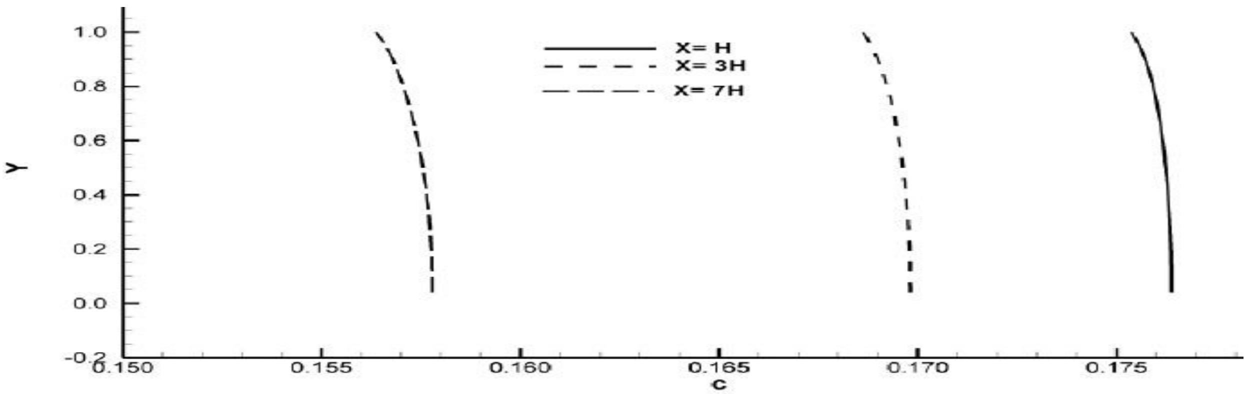

Figure 7: Pollutant concentration profiles of the windward wall at different building locations are shown for building of height H.

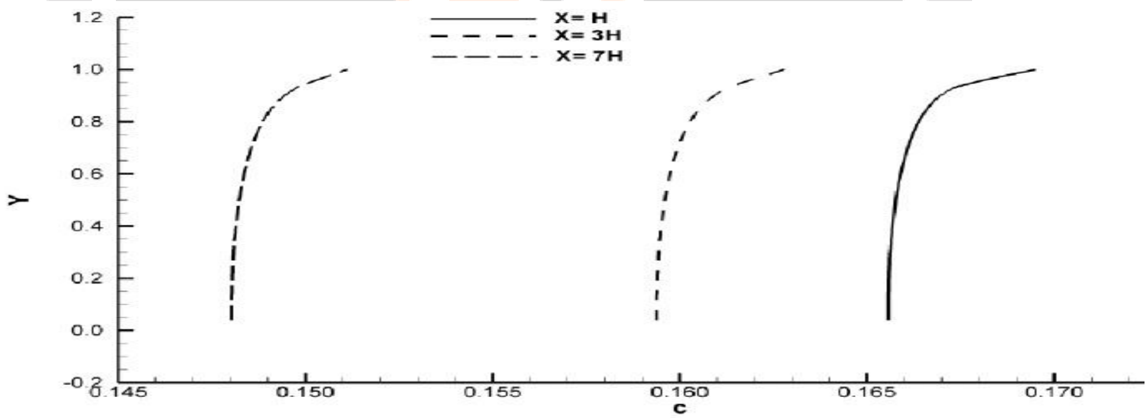

Figure 8: Pollutant concentration profiles of the leeward wall at different building locations are shown for building of height H.

Table 2: Values of average wall concentration on the leeward and windward wall are shown for building height H

Case	Building Height	Distance (X)	Windward	Leeward
a	H	H	0.176	0.166
b	H	3H	0.169	0.159
c	H	7H	0.157	0.148

4. CONCLUSIONS

In the present study, pollution dispersion for cosine hill shape $y = H\ Sin^2(x_1\pi/2L)$ and rectangular building shapes are studied. The location of building is changed with respect to hill and pollutant source. The Reynolds number for the entire study is taken constant 13800.

- The pollutant source is present at the inlet from 0.75H to 1.25H value.
- The pollution level is find minimum for all building shapes when the building is farthest from the hill. The effect of hill height on this farthest location is checked also.
- The concentration of pollutant, velocity and turbulent kinetic energy has high gradient at the height of the building.
- The recirculation zone between building and hill is spreading and at last a reattachment point appears at distance 9H from hill crest.

6. REFERENCES

[1]. C. M. Rhie, W.L.C., Numerical study of the turbulent flow past an airfoil with trailing edge separation. A I AA , 1983. 21(11): p. 1525-1532.

[2]. K. Abe, T.K., Y. Nagano, A new turbulence model for predicting fluid flow and heat transfer in separating and reattaching flows-l.

[3]. Ted Stathopoulos, B.H., Ali Bahloul, Analytical evaluation of dispersion of exhaust from rooftop stacks on buildings, in Chemical Substances and Biological Agents. 2008, IRSST: Montreal, Canada.

[4]. A. K. Saha, G.B., K. Muralidhar, Numerical study of the turbulent unsteady wake behind a partially enclosed square cylinder using RANS. Computer Methods in Applied Mechanics and Engineering, 1999. 178: p. 323-341.

[5.] Ugo Piomelli, E.B., Wall-Layer models for large eddy simulations. Annual Review of Fluid Mechanics, 2002.

34: p. 349-374.

BIOGRAPHIES

	Mr. Rahul Kumar PG Scholar. Department of Mechanical Engineering Sri Satya Sai Institute of Science and Technology, Sehore, M.P., India Contact no. +91-9680541033 Email: rahul.aero001@gmail.com
	Mr. Devesh Kumar Singh PG Scholar Department of Mechanical Engineering Bhagwant University, Ajmer, Rajasthan, India Contact no. +91- 7737649429, Email: dsk014@gmail.com
	Mrs. Priyanka Jhavar Associate Prof. Department of Mechanical Engineering Sri Satya Sai Institute of Science and Technology, Sehore, M.P., India Contact no. +91-9424875570 Email: priyanka.jhavar10@gmail.com

	Mr. Prashant Singh Assistant Prof. Department of Aeronautical Engineering Sri Satya Sai Institute of Science and Technology, Sehore, M.P., India Contact no. +91-8349687471, +91-9770661882 Email: prashant.singhkalhans@gmail.com