CFD Simulation in Township Planning – A Case Study

[1],Nilesh S. Varkute, [2],R.S. Maurya
[1] Sardar Patel College Of Engineering, Mumbai, Maharashtra, India.
[2] Sardar Patel College Of Engineering, Mumbai, Maharashtra, India.

Abstract

There are increasing concerns regarding the quality of urban public spaces. Wind is one important environmental factor that influences pedestrians' comfort and safety. In modern cities there are increasing numbers of high constructions and complex forms which can involve problems of significant wind discomfort around these buildings. Architects and town planners need guidelines and simple design tools to take account of wind in their projects. In recent years the numerical investigation has emerged as powerful and sufficient tool for building optimization and a better township planning. Present work proposes the methodology to carry out exterior numerical on a cluster of existing structures and concludes with significant outcomes which are mostly neglected by the planners. Present work uses a three dimensional scale down model of buildings where steady incompressible flow analysis has been done. It has been implemented through ANSYS Fluent 12.0 using SIMPLE algorithm as solver. The effect of turbulence has been captured using k-ε model. Simulated work validates well with the available experimental result. The work progresses with single building simulation to a cluster of building with common external amenities. Some of the significant conclusion are changing ventilation pattern of air among buildings with wind direction, flow separation, wind shadow effect, location of amenities such as garden, playground etc.

Keywords: Building, CFD, Environment, High Rise Building, k-ε model, Optimization, Simulation, Township.

1. Introduction

With the development in technology taller and taller structures are being designed and constructed to care of the local need and desire. Such structures have a significant effect on the surrounding wind patterns due to the change in the local wind flows. This causes many environmental problems in nearby areas such as accelerated wind flow at the ground level impacting the comfort, and sometimes safety, of the users of the building and the pedestrians in the surrounding street canyons. Sometimes it creates permanent pollutant dispersion problem in the locality. Other significant effects of changed wind flow pattern are wind-sun shadow effect and locating amenities like playground, garden, swimming pool etc. For the city planners and building designers, prior knowledge of the air flow around builds play a significant role in finalizing their design. At the building scale, the theoretical challenges are inherent in the complex interplay of thousands of components, each with their own complex physical behavior and a multiplicity of interactions among them. Such kind of complexity can be easily handled numerically. Therefore numerical investigation for building has gained a well-respected role in the prediction, assessment, and verification of building behavior. Two conventional approaches i.e. Building energy balance model (BES) and Zonal airflow network (AFN) models are slowly losing their significance and now the focus is more towards CFD based simulation which is based on mass, momentum and energy conservation in the flow domain. But CFD simulation of building design is a challenging task due to large length scale with complex geometry, the uncertainty in assigning boundary condition and wide range of physical processes occurring in the surrounding.

The most important physical phenomena in the externally built environment are the atmospheric boundary layers, the unsteady flow, the separating bluff body flow and dispersion. In spite of such computational difficulties, CFD has become an influential tool in the building industry, especially with its capability to visualize air flow in and around the building. It is used to justify the selection of design option. Not only as a justification or confirmation tool, it has emerged as a shaping and molding tool also on the drawing board where the design is refined. Most of the work on building simulation is dedicated to optimizing internal environment of the rooms or interiors where location and orientation of the building were prime focus. Feustel and diamond [1] carried out an experimental and a numerical study to investigate air flow pattern and ventilation systems in the high-rise buildings. Using a building energy simulation program and a computational fluid dynamics program, Zhai [2] investigated the influence of building scales on building cooling energy consumption with and without natural ventilation. Investigations on natural ventilation design by Carriho-da-graca et al. [3], prediction of smoke and fire in buildings by Lo et al. [4] and Yeoh et al. [5], particulate dispersion in indoor environment by Quinn et al. [6], building element design by Manz [7], and space indoor environment analysis by Eckhardt and Zori [8] are some significant work on internal simulation on buildings. Often, the outdoor environment has a significant impact on the indoor environment. Therefore to solve the problems related to natural ventilation requires the study of both the indoor and outdoor environment together. Few such research work are simulations of outdoor airflow and pollutant dispersion by Sahm et al. [9] and Swaddiwudhipong and Khan [10], and combined indoor and airflow studies by Jiang and Chen [11]. Due to large spatial scale external building simulation could not progress at same pace as internal building simulation. But several efforts have been made by researchers in this direction also. Baskaran and Stathopoulos [12] presented the computer simulation of wind pressures on buildings. Cheng et al. [13] reported a systematic computational study of wind-induced natural ventilation and pollutant transport of re-entrant. Thepmongkorn et al. [14] investigated interference effects of neighboring tall buildings on wind-induced coupled translational and torsional motion. Investigation was carried out through a series of wind tunnel aero elastic model experiments. Baskaran and Kashef [15] used numerical techniques to predict wind flow conditions around a single building, between two parallel buildings and around a multiple building configuration. Bosch et al. [16] presented an experimental investigation on the flow past a square cylinder placed near a wall at Re = 22,000. Lakehal and Rodi [17] presented a 3D steady simulation of flow around a cube placed in developed-channel flow with various versions of the k- ε model. Rodi [18] compared LES and RANS calculation of flow around bluff bodies. Castelli et al.[19] investigated flow pattern around dome shaped structure by using proper boundary conditions like velocity inlet, pressure outlet, and symmetry in lateral and top direction, wall for ground and building wall surfaces. The review of literature indicates insufficient work in the area of external wind simulation around building structures which are useful for architects, city planner and building designers. The objective of the present work is to illustrate the capability and potential of modern CFD tool through a detailed case study of a cluster of high rise building located in Mumbai city.

Investigation would be helpful for the planners to take many critical decisions such as influence of high rise buildings and building clustering on surrounding settlements, locating garden area, play area, community centers, internal roads etc. to make their project better and effective from environmental view point. It contributes in structural analysis of the buildings also with the knowledge of wind drag data. A systematic investigation is proposed where following important parameters like effect of wind flow direction, building orientation, in between building spacing, building layouts are discussed. Based on available metrological data of Mumbai city, all simulation has been executed using wind speed of 10 m/s. Any kind of thermal effect arising due to the building is not a part of the investigation.

2. Problem Definition

Present problem is to carry out numerical investigation of air flow pattern around a cluster of existing high rise structures located in west coast mega city i.e. Mumbai of India. The bird eye view of this small township is shown in Fig.1. It consists of fourteen structures of different height and orientations. The objective is to know the effect of changing wind direction and predict the optimum location of garden, play area, community centers, and internal roads along with its influence on surrounding settlements. The investigation starts with the numerical study of single building, two buildings and multi building's inline and staggering layouts. The orientation of single building with L, B and H are length, breadth and height respectively is depicted in Fig.2. To investigate the channeling effect created by wind in between the buildings, two buildings of same size is proposed as shown in Fig.3.

It is not a part of CFD analysis

Figure 1. Bird view of township

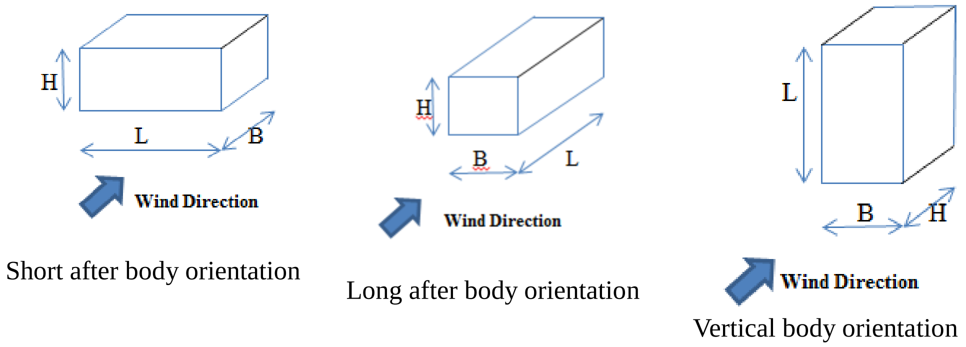


Figure 2. Orientation of building

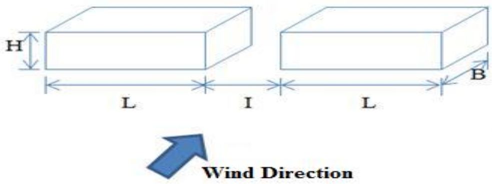


Figure 3. Two Building Model

3. Modeling of Problem

3.1. Mathematical Model

The average wind speed of Mumbai region is about 10 meters per second based on the available metrological data. The movement of air around the building can be modeled as Newtonian fluid with turbulent, viscous, incompressible and steady flow in nature. The flow behavior is governed by mass and momentum conservation with appropriate turbulence model where the body force can been neglected.

3.1.1. Governing Equations

The governing equation for Newtonian, incompressible and steady flow without turbulence is presented here, To turbulence is captured by using standard K-ε Model which leads to two additions transport equation i.e. kinetic energy(k) and turbulence diffusivity (ε).

Mass Balance,

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad \ldots (1)$$

X- Momentum Equation,

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \qquad \ldots (2)$$

Y- Momentum Equation,

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right) \qquad \ldots (3)$$

Z- Momentum Equation,

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \qquad \ldots (4)$$

Turbulence Equation: K-ε Model

Air flow simulation around the building needs scale down of the actual building size for the sake of computational efficiency. Being external flow investigation, a judicious selection of appropriate computational domain considering the geometrical scale of the problem is must along with suitable boundary condition.

a. The inlet boundary condition is based on finite velocity which is obtained from the average wind speed of the location collected from metrological department of India based on past 10 years data.

b. Model of air behaving like a viscous fluid, no-slip wall boundary condition is used for ground part of the computational domain.
c. Remaining parts of the computational domain is assumed to be at ambient condition i.e. zero gauge pressure.

3.2. Numerical Implementation

Numerical simulation of the cases presented in the previous section is done through the ANSYS Fluent 12.0. It starts with the selection and modeling of the computational domain and its meshing. Physics of the problem demands sufficiently lengthy downstream length to care of vortex formation behind the structure. To capture presence of velocity gradient near the structures sufficient fine meshing is required. The selection of proper solver settings is essential for successful simulation. For pressure velocity decoupling SIMPLE algorithm is used and turbulence is captured by standard k-ε model. The convective terms present in momentum, kinetic energy and turbulent dissipation is treated by using first order upwind scheme, to interpolate pressure cell based pressure data is averaged at the faces and the pressure gradient is estimated using least square cell based technique.

4. Results and Discussion

This section deals with the result and discussion associated with air flow distribution around single body, two body, multi-body and actual existing cluster of building respectively. The work has been validated with experimental result. For the sake of completeness of the investigation the parameter like air flow distribution, direction of wind, recirculation zone and drag forces have been considered for the analysis and conclusion.

4.1 Single Building Simulation

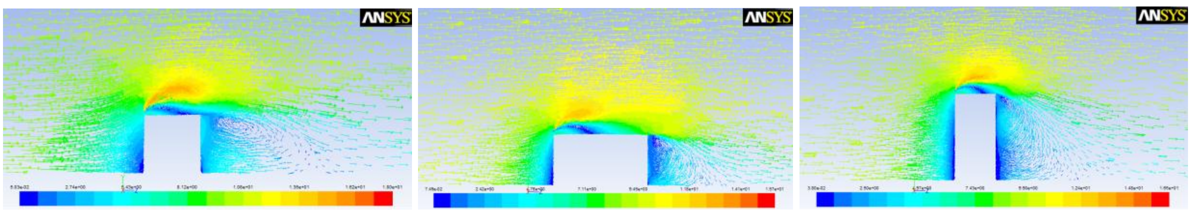
The investigation is performed on a building of dimension L=40 m, B=20 m, H=20 m with short, long and vertical body orientation which is depicted in Fig.2. Figure 4 shows a movement of air around building under different orientations through velocity vectors. The flow can be observed to be getting stagnated as it approaches the building. A separation of flow from the front edges of the building and development of a weak velocity recirculation zone behind the building can be observed in all three orientations. With the length of recirculation zone, and total drag force data for each building orientation is presented in Table 1. The total force component along the specified force vector $\vec{a}$ on a wall zone is computed by summing the dot product of the pressure and viscous forces on each face with the specified force vector. The terms in this summation represent the pressure and viscous force components in the direction of the vector $\vec{a}$.

$$F_a \text{(Total Force)} \quad = \quad \vec{a} \cdot \vec{F}_p \text{ (Pressure Force Component)} + \vec{a} \cdot \vec{F}_v \text{(Viscous Force Component)}$$

Where $\vec{F}_p$ and $\vec{F}_v$ are specified, pressure and viscous force vector respectively.

Table 1. Comparison between orientations of building

Parameter/Orientation	Short after body orientation	Long after body orientation	Vertical body orientation
Recirculation Zone length (m)	37	24	31.5
Total Drag Force (N)	49912.35	25015.21	52273.75



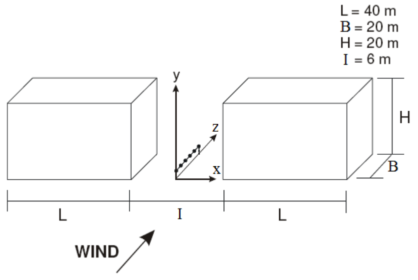
(a) (b) (c)

Figure 4. Velocity vectors (a) short after body, (b) long after body and (c) vertical body orientation

A low value of recirculation zone length and drag force acting on the structure can be observed for long body orientation of the structure. This can be attributed to the nature of flow separation occurring in all three orientations. Reattachment of flow in early length part and second separation at extreme downstream leads to lesser pressure and viscous drag. So long body orientation is best from good in design point of view.

4.2. Two Buildings Simulation

The simulation investigates the optimum nearness between two buildings for human comfort and good city planning. The model considered for investigation is depicted in Fig.5 which has been experimentally investigated by Baskaran and Kashef [10]. The simulated results for velocity at mid-section in between the building spacing along z- direction of flow is validated with experimental data and presented in Fig.6 for in-between building gap of 6 meter. Here U and V are local and average incoming wind speed. It is found that the magnitude of flow velocity between two buildings is increases first and then it decreases. Numerical result matches well with the experimental work with maximum error of 5%.



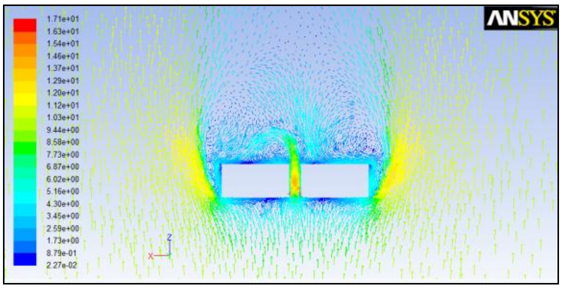


Figure 5. Model and velocity vector

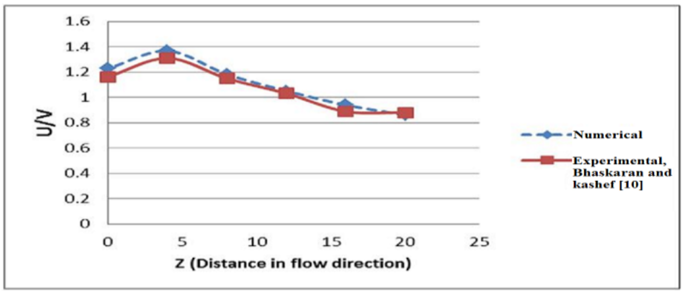


Figure 6. Validation graph for two building model

A numerical investigation of in-between building gap (I) on velocity along the downstream at height of 2 meter from the ground is presented in Fig.7. The building gap (I) is varied from 6 meters to 20 meters. This variation can be attributed to the venturi effect created in between the building spacing. The maximum and average both wind velocity can be observed to be rising and then decreasing. The magnitude of several parameters is presented in Table 2.

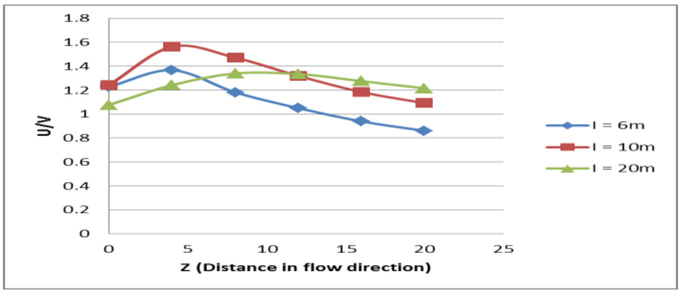


Figure 7. Comparison of channeling effect between two buildings for various spacing

Table 2. Parametric comparison of channeling effect between two buildings

Parameter	I = 6 m	I = 10 m	I = 20 m
Maximum Velocity rise along the flow (m/s)	13.67	15.64	13.40
Flow pattern behind buildings	Non-uniform	Uniform	Uniform
Total Drag Forces (N)	139791.68	113159.12	132608.37

4.3. Existing Cluster of High Rise Structures

The bird eye view of existing structures as depicted in Fig.1, consisting of many tall structures with few small and many amenities such as playground, garden and swimming pool etc. In numerical simulation some of small structures have been removed to find air flow distribution only due to tall structures (50 meter high). Simulation presents the investigation for natural ventilation around, location of external amenities and drag data due to changing wind direction. The computational domain as shown in Fig.8 is chosen for the simulation is with: Upstream distance = 400 m, Downstream distance = 1180 m, Side distances = 400 m and top distance = 250 m. The domain meshed with 3.3 lakhs tetrahedral mesh as computational cells with maintaining fine meshes near the structures and ground. The simulation has been carried out under north, west, SW and south wind directions.

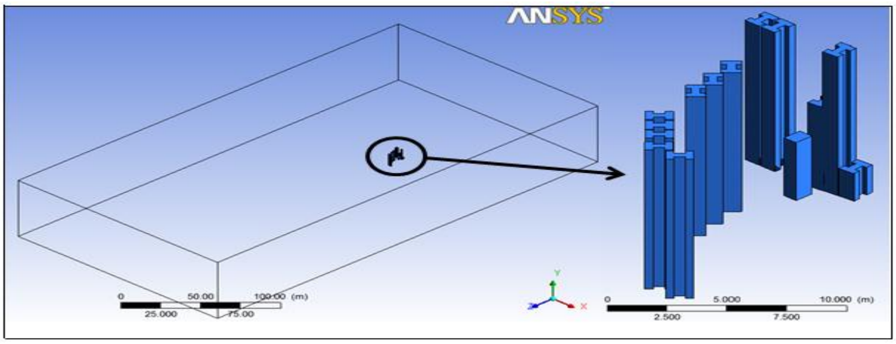
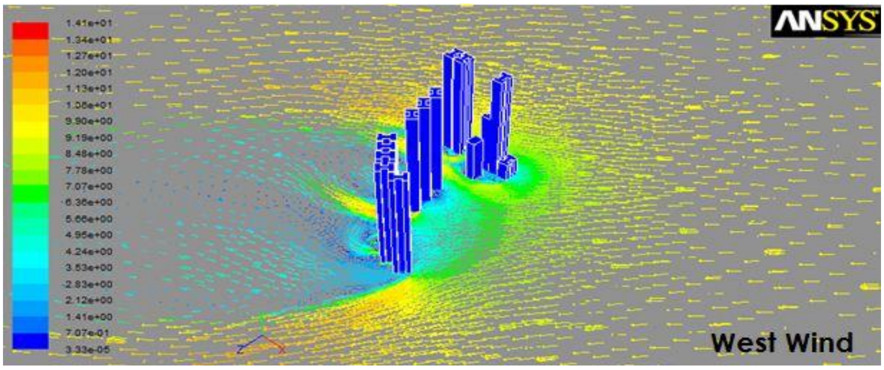


Figure 8. Computational domain for cluster of structures

Figure 9 shows the effect of wind direction on velocity distribution around the structures. The formation of vortex and wind shadow zone can be observed under each wind direction simulation. A wake region and stagnant zone develops behind the building. This is not good for pleasant ambience. The town planners need to be careful in minimizing such wind effects. In the west coastal region where Mumbai is located, have wind direction throughout the year is either north or west direction. Therefore all external amenities must be located on the upstream side for better ventilation and pleasant ambience. For the case under investigation the playground, swimming pool and landscape garden is properly located. There is no wind shadow effect of one structure over others. In case of southward or south west wind has sever wind shadow effect and the layout of the building is not good for these two wind directions. Figure 10 shows the zoomed view of air flow in-between the structure. It illustrates the qualitative results which is good enough for building planners to take critical decision such as exterior layout of building, building spacing, stagnant pockets created by wind flow. In present vector plot it can be seen that except few places where the structures are extremely nearby there is a development of stagnant zone which is not good from building planning point of view.



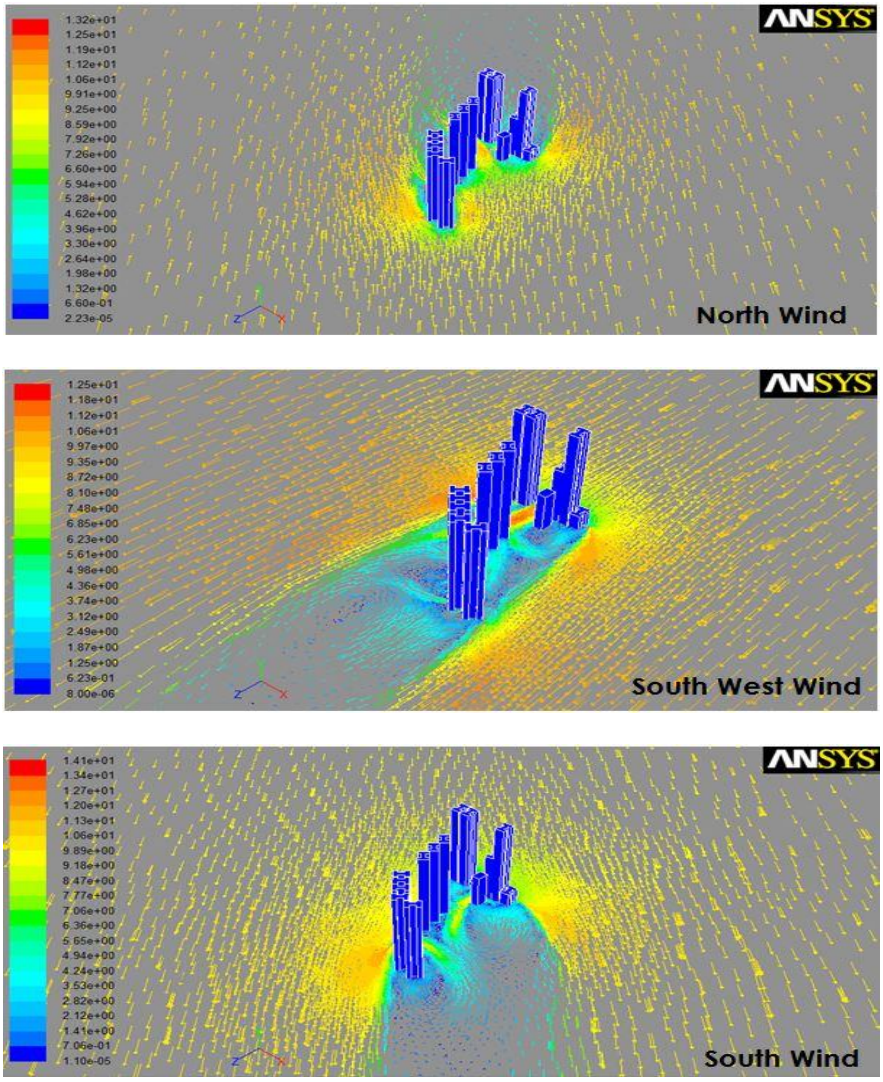


Figure 9. Velocity vector in different wind direction

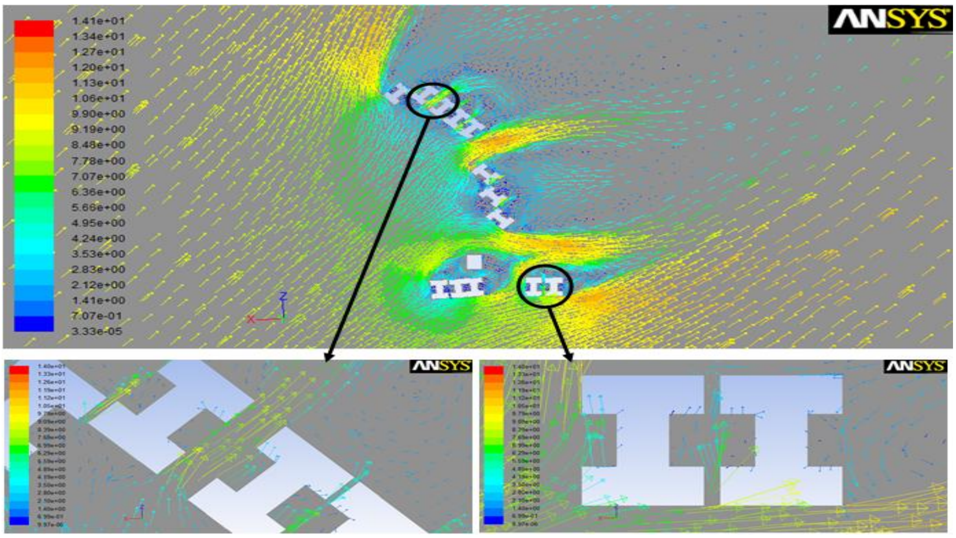


Figure 10. Wind velocity in-between two buildings

5. Conclusion

A three-dimensional numerical model is developed to analyze the flow pattern around buildings. From the preceding discussions, the following conclusions can be made:

a. Computation fluid dynamics is a powerful tool for the investigation of building air flow applications. It provides detailed predictions of air velocities around buildings.
b. Gap between two buildings is significant and needs to be investigated before planning.
c. Due to the detailed study of flow pattern around the buildings we are able to capture correct location of air conditioning systems, heating installations etc.
d. A good natural ventilated location around an existing can be easily located with the help of CFD analysis.

References

[1] H.E. Feustel and R.C.Diamond. Air flow distribution in a high-rise residential Building. Lawrence Berkeley National Laboratory, Berkeley, 1996.
[2] Z. Zhai. Numerical study of optimal building scales with low cooling load in both hot and mild climate regions. In Proceedings of the 17th Annual North American Waste-to-Energy Conference, May 18-20, Chantilly, Virginia, 2009.
[3] G. Carrilho da Graca, Q. Chen, L.R. Glicksman, and L.K. Norford. Simulation of wind-driven ventilate cooling systems for an apartment building in Beijing and Shanghai. Energy and Buildings, 34(1):1–11, 2002.
[4] S.M. Lo, K.K.Yuen, W.Z. Lu, and D.H. Chen. A CFD study of buoyancy effects on smoke spread in a refuge floor of a high-rise building. Journal of Fire Sciences, 20(6):439–463, 2002.
[5] G.H.Yeoh, R.K.K.Yuen, S.M. Lo, and D.H. Chen. On numerical comparison of enclosure fire in a multi-compartment building. Fire Safety Journal, 38(1):85–94, 2003.
[6] A.D. Quinn, M. Wilson, A.M. Reynolds, S.B. Couling, and R.P. Hoxey. Modelling the dispersion of aerial pollutants from agricultural buildings—an evaluation of computational fluid dynamics (CFD). Computers and Electronics in Agriculture, 30(1): 219–235. 2001.
[7] H. Manz. Numerical simulation of heat transfer by natural convection in cavities of facade elements. Energy and Buildings, 35(3):305–311, 2003.
[8] B. Eckhardt and L. Zori. Computer simulation helps keep down costs for NASA's 'lifeboat' for the international space station. Aircraft Engineering and Aerospace Technology: An International Journal, 74(5):442–446. 2002.
[9] P. Sahm, P. Louka, M. Ketzel, E. Guilloteau, and J.F. Sini. Inter-comparison of numerical urban dispersion models—part I: Street canyon and single building configurations. Water, Air and Soil Pollution: Focus, 2(5–6):587–601, 2002.
[10] S. Swanddiwudhipong and M.S. Khan. Dynamic response of wind-excited building using CFD. Journal of Sound and Vibration, 253(4):735–754, 2002.
[11] Y. Jiang and Q. Chen. Effect of fluctuating wind direction on cross natural ventilation in building from large eddy simulation. Building and Environment, 37(4):379–386, 2002.
[12] Baskaran and T. Stathopoulos. Prediction of wind effects on buildings using computational methods review of the state of the art. Canadian Journal of Civil Engineering, 21:805-822, 1994.
[13] C.K.C. Cheng, K.M. Lam, Y.T.A. Leung, K. Yang, H.W Danny and C.P. Cheung Sherman. Wind-induced natural ventilation of re-entrant bays in a high-rise building. Journal of Wind Engineering and Industrial Aerodynamics, 99:79–90, 2011.
[14] S. Thepmongkorn, G.S. Wood, K.C.S. Kwok. Interference effects on wind-induced coupled motion of a tall building. Journal of Wind Engineering and Industrial Aerodynamics, 90:1807–1815, 2002.
[15] A.Baskaran, and A. Kashef. Investigation of air flow around buildings using computational fluid dynamics techniques Engineering Structures, 18(11):861-875,1996.
[16] G. Bosch, M. Kappler, W. Rodi. Experiments on the flow past a square cylinder placed near a wall. Experimental Thermal and Fluid Science, 13:292-305, 1996.
[17] D. Lakehal, W. Rodi. Calculation of the flow past a surface-mounted cube with two-layer turbulence models. Journal of Wind Engineering and Industrial Aerodynamics 67-68:65-78, 1997.
[18] W. Rodi. Comparison of LES and RANS calculations of the flow around bluff bodies. Journal of Wind Engineering and Industrial Aerodynamics, 69-71:55-75, 1997.
[19] M.R.Castelli, S.Toniato and E. Benini. Numerical analysis of wind loads on a hemi cylindrical roof building. World Academy of Science, Engineering and Technology,56, 2011.