RESEARCH ARTICLE OPEN ACCESS

Computational modeling of wind flow and its effect on a cubical building using Large Eddy Simulation (LES) technique

[a]Venugopal M.M., [b]S.K. Maharana

[a]Asst. Professor, [b]Professor, Dept. of Aeronautical Engineering, Acharya Institute of Technology, Bangalore-560107

Corresponding Author: Venugopal M.M.

ABSTRACT:

The wind flow and its effects over cubical buildings lying in close vicinity in urban areas generates flow interference effects causing problems related to pollution, pedestrian comfort and ventilation within the buildings thus promoting a lot of research interest in this area during the last few years. The wake and its characteristics of a building has drawn attention for various applications like dispersion of pollutants downwind of conventional or nuclear power plants, airport runway interference effect, take-off/landing limitation at heliports, pedestrian wind comfort, and wind loads on structures.Computational Fluid dynamics (CFD) is one of the promising technologies for investigating these important issues, which will not necessarily be solved by traditional wind tunnel technology. Large Eddy Simulation Technique (LES) shall be used to predict the flow physics- external flow around the building. The proposed work has been accomplished through numerical simulation using ANSYS-FLUENT 12.1 solver.The effects have been quantified in terms of the pressure distribution, turbulence in and around the building, three-dimensionality effect on the overall flow patterns

Keywords: Wind effects, CFD, Large eddy Simulation, Three-dimensionality effect

---

Date of Submission: 21-06-2018 Date of acceptance: 09-07-2018

I. INTRODUCTION

The wind effect is significant as far as pollution, pedestrian comfort and ventilation within the building are concerned. The present study makes an attempt to investigate wind flow and its effects on a prismatic building. It is important to stress that wind flow over a bluff body such as a building with sharp corner (Tamura et al. 1999) is complicated by the presence of streamline curvature effects, flow separation, re-attachment and flow recirculation etc. (Murakami S et al. 1990). Further complications arise due to the presence of atmospheric turbulence. Numerical modeling taking into account all these flow phenomena is an extremely difficult task and is limited by computational infrastructural limitations. Experimental data generation is certainly required for validation but realistic comparison between experimental and numerically predicted data depends on accurate simulation of the physics of the problem both experimentally as well as numerically.

In modern time, with the advent of high-speed computers, it is possible to tackle many problems on fluid dynamics with relative case. Thus the importance of fluid dynamics is rapidly growing. Since computational fluid dynamics (CFD) is capable of analyzing the pressure and velocity field of moving fluids, computational wind engineering (CWE) has been achieving much in this area. In the world of CFD it is possible to simulate hypothetical situation, which cannot be realized in wind tunnel experiments. In the framework of computational wind engineering, which deals with the application of CFD methodologies in the classical wind engineering and building aerodynamics problems, numerical solutions have been developed with the potential to overcome these limitations. However, such solutions have not yet fully demonstrated their ability even for simple building geometries exposed to strong horizontal winds. The wind flow when turbulent needs to be modeled properly. Turbulence modeling presents the biggest problem in predicting the flow field structure around bluff bodies with sharp corner. The initial enthusiasm on RANS based k-ε model died down in late eighties in favor of LES model because the latter can reproduce anisotropic flow field and unsteady vertical phenomena much better.

Due to non-availability of large computational infra-structure required to solve complex 3D problems through LES, many researchers have used 2D LES simulation. However it is recognized that 2D predictions do not compare well with experimental data since vortex stretching which plays a dominant role in 3D unsteady flow is absent in 2D simulation (Sakamoto et al. 1993). Rodi (Rodi 1993) also observed 2D LES calculations are quite inferior to 3D calculations which is best suited to cope with complex vortex shedding flows with large scale structures. This approach does most justice to physics at the price of high computing cost. Bouris and Bergeles (Bouris et al. 1999) performed 2D LES computations to obtain turbulent kinetic energy of both periodic and fluctuating components and compared the computed data with experiments along the centerline behind a square cylinder. They concluded that "from physical point of view one should aim at 3D LES with 2D LES being but a compromise imposed by computer limitation".

Thus large computational space, time and prohibitively high cost involved in 3D LES computation have prompted many authors to confine themselves to 2D LES computations.

Though it is quite challenging to carry out computation for 3D LES at attempt has been made to predict the flow properties and make a conclusion from the numerical prediction using ANSYS-FLUENT 12.1 CFD solver.

There has been an attempt to validate the different physical phenomena arising out of the wind flow over a single 3D (three dimensional) building model. Some of the parameters like velocity and pressure distribution over a single building have substantially motivated to study the flow over prismatic building in tandem with a sloping roof structure. There is a need to acquire an understanding of the flow around a single building model for different reasons mentioned above. This understanding shall be extended while studying the flow when two structures are kept in tandem. In this project it is attempted to see if all the physical phenomena named in the Fig.1 are existing after aCFD simulation and then exhibit the changes in the same when there is a sloping roof either in front or back of the building.

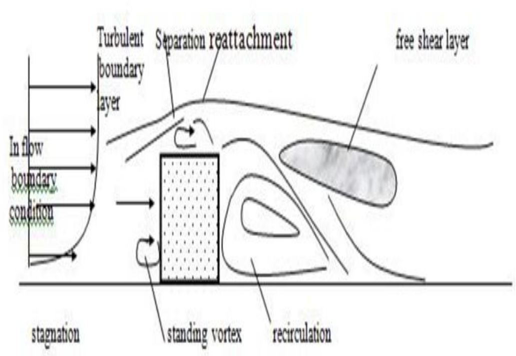


Fig.1: General understanding of flow field around a building model.

II. GOVERNING FLOW EQUATIONS

In large eddy simulations the large energy carrying scales are directly computed, and only the effects of small sub-grid ones are modeled. The large scale quantities (indicated by over bar) are defined by the filtering operation,

$$\bar{f}(x) = \int f(x')G(x, x')\,dx' \qquad (1)$$

in which G is the filtering function and integral is extended over the entire domain. Filter functions commonly used include the Gaussian, the sharp Fourier Cutoff and the top hat in real space. Applying filtering operation to the appropriate set of governing equations, one obtains the filtered set of governing equations as given below. For incompressible, isothermal flows there are filtered continuity and NS equations for LES (Murakami 1990, Qasim et al. 1992, Murakami et al. 1995) given, in dimensionless form, by

Governing equations of flow:

Continuity equations: $$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \qquad (2)$$

Momentum equations: $$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial}{\partial x_i}\left(\frac{\bar{p}}{\rho} + \frac{2}{3}k_{sgs}\right) + \frac{\partial}{\partial x_j}(\nu + \nu_{sgs})\bar{S}_{ij} \qquad (3)$$

$$\nu_{sgs} = (C_s h)^2 [\frac{1}{2} (\overline{S}_{ij})^2]^{\frac{1}{2}},$$

$$\overline{S}_{ij} = \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i},$$

Where,
$$k_{sgs} = \frac{\nu^2{}_{sgs}}{(C_k h)^2}$$

Where $\overline{u}_i$ and $\overline{u}_j$ are the velocity vectors, x_i and x_j are the component of spatial coordinates, $\overline{p}$ pressure,ν_{sgs} sub grid scale eddy viscosity, k_{sgs} sub grid component of kinetic energy, $\overline{S}_{ij}$ strain rate tensor, h mesh scale,ρ air density and ν free stream viscosity.

III. METHODOLOGY

Modeling:

Modeling of single building is done in CATIA V5 which is a well-known Computer Aided Design (CAD) tool. The flow domain is 105 (x-direction) x 105 (y-direction) x 60 (z-direction) in mm. The dimensions of the single building are 5x5x10 in mm. The grid size is uniform throughout the zone except near the solid surfaces. Below is given an isometric view of the whole arrangement of a single 3D building model inside its flow domain.

Meshing:

Meshing is done by utilizing the capability of ANSYS® ICEM CFD, a meshing tool that supports hexahedral structured grid and near the solid wall finer mesh is developed to capture exact flow physics near the wall surface of the building and at the domain walls are meshed coarse. The mesh size is 1mm. The number of cells is 667500, faces 2026400 and the nodes 691662.

Solution Procedure and mesh independence study

First the meshed model was imported to the solver where in the domain (front and rear surfaces) were defined with inlet velocity (at the front) and outflow respectively. Then the other surfaces of the single building structure were defined as walls and fluid interior defined as air at standard conditions. Since the interest was to study a turbulent flow around a building Large Eddy Simulation method was chosen from the options of the solver. Turbulent flows are characterized by eddies with a wide range of length and time scales. The largest eddies are typically comparable in size to the characteristic length of the mean flow. The smallest scales are responsible for the dissipation of turbulence kinetic energy. In LES, large eddies are resolved directly, while small eddies are modeled. Large eddy simulation (LES) thus falls between DNS (Direct Numerical Simulation) and RANS(Reynolds Averaged Navier Stokes) in terms of the fraction of the resolved scales. The rationale behind LES can be summarized as follows:

1. Momentum, mass, energy, and other passive scalars are transported mostly by large eddies.
2. Large eddies are more problem-dependent. They are dictated by the geometries and boundary conditions of the flow involved.

The convergence of the numerical solutions is obtained from the above mentioned problem using the residuals of the values of variables obtained after solving the continuity and the momentum equations. In this work, convergence occurs when the values of total residual in all the above-mentioned equations become smaller than 10^{-5}. All these values have reached their acceptable steady solutions during the simulation. The solutions are also independent of the mesh resolution.

Grid quality has been satisfactory after checking the skew as 0.2 and overall grid quality 0.92 (The highest value is 1.0). The gridding method of the tool has optimized the grid density appropriately to fit into flow domain and overall grid quality.

Boundary Conditions

The real driver for a particular solution of a set of governing equations is boundary conditions. The flow is viscous in this case. The specification of these boundary conditions is extremely important since it directly affects the stability and accuracy of the solution. Physical boundary conditions are applied on the geometric surfaces of the building model. Free stream far ahead of the building is considered.

Boundary conditions are as follows

i) Inlet to the domain : Velocity inlet, U_∞=1 m/s

ii) Outlet from the domain : Gauge pressure outlet, p= 0 pascal

iii) Wall of the domain : No slip wall boundary (u=v=w=0)

The boundary conditions and the initial conditions for the governing flow equations, Reynolds-averaged Navier-Stokes(RANS) are mentioned above. The approach velocity profile near the solid wall has been assumed to follow power law.

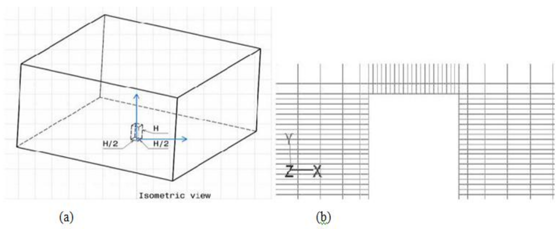


Fig.2 :(a) An isometric view of the flow domain of a single 3D building model and (b) the grid arrangement in the vicinity of the solid wall of the building

The meshing of the plane of the selected domains is indicated. This shows how has a solid surface of the building model as well as the other region of flow been arranged in the mesh. One observation is that the mesh near the solid walls is finer than that of elsewhere in the domain. One of the reasons is that coarse grids shall be saving computational time in the process. Turbulent flows are characterized by eddies with a wide range of length and time scales. The largest eddies are typically comparable in size to the characteristic length of the mean flow. The smallest scales are responsible for the dissipation of turbulence kinetic energy. In LES, large eddies are resolved directly, while small eddies are modeled. Large eddy simulation (LES) thus falls between DNS and RANS in terms of the fraction of the resolved scales. The rationale behind LES can be summarized as follows: momentum, mass, energy, and other passive scalars are transported mostly by large eddies. Large eddies are more problem-dependent. They are dictated by the geometries and boundary conditions of the flow involved.

The above methodology is extended while solving the flow over sloping roof structure in tandem with a prismatic building.

IV. RESULT AND DISCUSSION

4.1 Building model is at an angle (0^o) to the geometric axis.

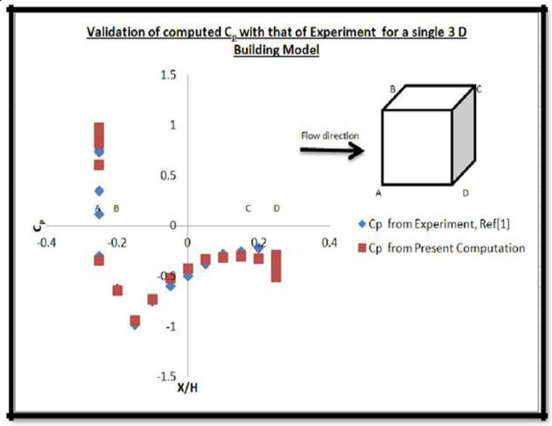


Fig.4.1 Validation of computed C_p with experimented 3D single rectangular building [1]

From the Fig.4.1 it is observed that at windward side of the building computed C_p has little difference with the experimental values. This shall be investigated during final stage. In other surfaces like top and leeward side of the building the C_p value tallies with that of experiment.

From the velocity vector plot, which is taken at the centre plan of the building, over the single 3D building model it is observed that the flow separates at the sharp corner facing the wind where the velocity is of 1.2 times the inlet velocity. Then the velocity gets reduced to a minimum value of .0237 m/s at the wake zone, bottom leeward areas. This value is almost 1/40th of the inlet velocity. These areas are clearly being validated by the value of dynamic pressures shown in the Fig 4.1. The vortices which are produced from solid wall and are present in the wake zone show that the centre value of a single vorticity is almost double of that of the outermost value. This signifies that there is a highly turbulent region present in the wake of the building model. The Fig 4.2 is for x-velocity vector plots. This plot is taken at the mid plane of the building model. The plots justify the physics of the flow where the separation is distinguished at the sharp edges and standing vortices at the bottom of the model. The Fig.4.3 is contour of the z-vorticity perpendicular to x-y plane. The vorticity has shed from solid surface of the building and being carried along the flow. The strength is high inside the wake because the rotation of flow happens more prominently due to the difference of forces (and the moment) near to the solid surface and the wake zone. The Fig.4.4 is the contour for dynamic pressure. This component of pressure depends upon the velocity of the flow. The lower value of dynamic pressure is justified by the higher value of kinetic energy inside the flow.

Sub-grid turbulent viscosity is demonstrated by the Fig.4.5. The value of turbulence in the middle of the low pressure zone or the wake zone is extremely higher than the value of turbulent viscosity near the wall. This clarifies that the building is under a severe pressure gradient zone which might lead to the hazardous situation. The Fig. 4.6 is the top view of the flow domain that shows the contour plots of X-velocity. Both leeward sides of the building models has higher value of velocity of u away from the solid surface which is attached to a low speed wind flow. The flow from the both sides comes (after getting separated) towards reattachment after the wake zone. Pathlines taken in the Fig.4.7 demonstrates how the flow moves from inlet to the exit of the flow domain with the turbulence because of the presence of a solid body.

The Fig. 4.8(a) and (b) are vector plots . The front and top views of the velocity vectors convey important information like the direction of the flow and magnitude of velocity near and far from the body. They also signify how the flow develops, separates, recirculates and reattaches.

The Fig.4.9 (a) and (b) show the contour plots of velocity at two different time steps (500 and 2000) respectively. These show that the flow is unsteady and changes with time. The flow pattern has become different because of the chaotic motion of vortices as shown in the Fig.4.10(a)-(b), change of directions of velocity and fluctuation of other flow variations.

All these plots are for the wind flow whose direction is in parallel with the geometric axis of the building(that means 0^o orientation). An attempt is made to observe the flow pattern when the angle or building orientation is changed to 22.5^o. There could be multiple orientations of the building or multiple directions of the wind flow. The basic intent is to see when the wind direction changes for various reason what happens to theflow patterns and their effects of the building model. Here, to start with, a basic model is adopted. The Fig.4.11 is showing the pathlines for the building model orientated at 22.5^o. The flow is characterized by the presence of horse-shoe vortices and certain flow patterns near the solid surface. It is to be observed that the flow patterns have changed significantly and hence the load distribution on the building model is expected to vary. If the flow patterns shown in The Fig 4.13 and Fig.4.11 are compared then it shall be noted that the magnitude of velocity is higher at the leewards of building model(at 0^o orientation) than that in building model(orientated at 22.5^o). The quantification of this difference shall be done in the future work. The orientation has an impact on the formation of separation over the building top surface and at the leewards. The vector plot shown in the Fig. 4.12 also justifies the same logic at the location of leeward.

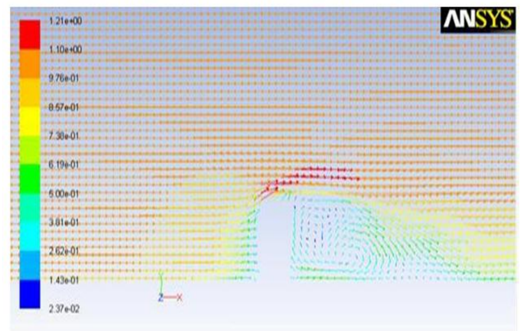


Fig. 4.2 Vector plots of X velocity (m/s) atcentre plane

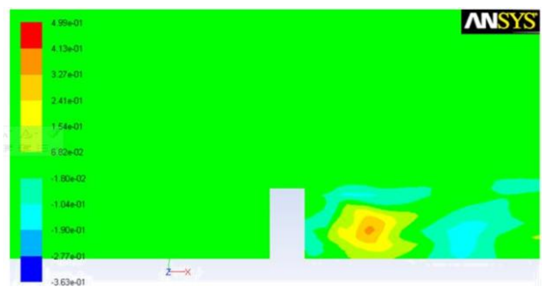
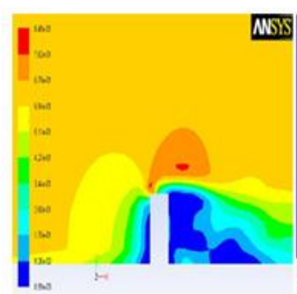


Fig. 4.3 Contours of z- vorticity at centre plane



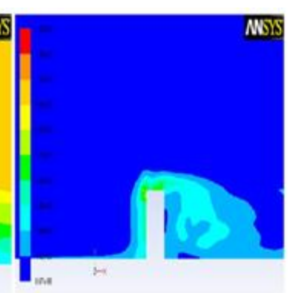


Fig.4.4 Fig. 4.5

Fig. 4.4 Contours of dynamic pressure (Pascal)
Fig. 4.5 Contours of sub-grid turbulent at centre planeviscosity (kg/m-s)

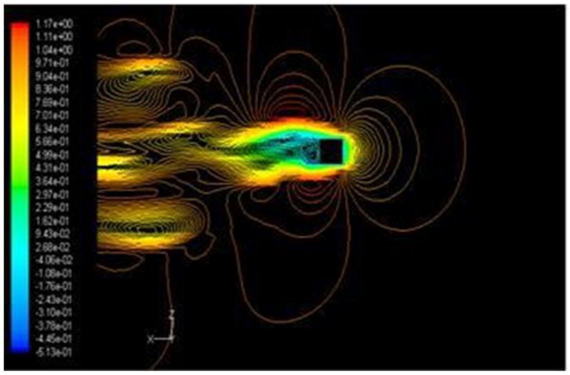


Fig. 4.6 Contours of X velocity (m/s) at horizontal plane of 3D single building

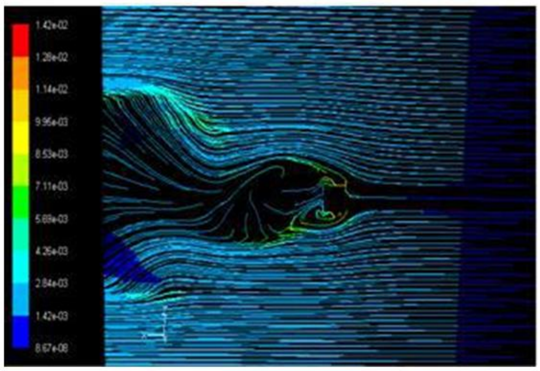


Fig. 4.7 Path lines around 3D single building at horizontal plane of 3D single building

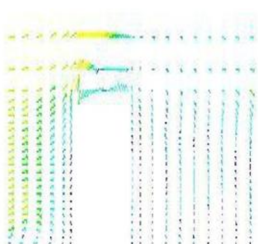


Fig. 4.8 (a) Velocity vector plot in x-y plane

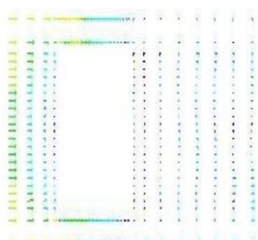


Fig. 4.8 (b) Velocity Vector plot in x-z plane

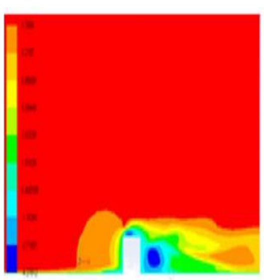


Fig. 4.9 (a) Velocity contours at time-step 2000

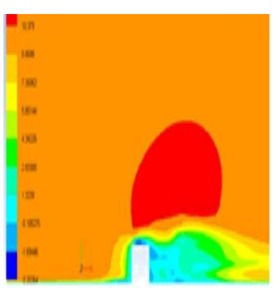


Fig. 4.9 (b) Velocity contours at time-step 500

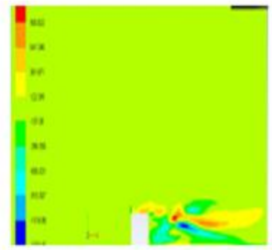


Fig.4.10 (a)

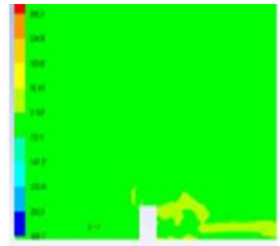


Fig.4.10 (b)

Fig. 4.10 (a) Vorticity contours at time-step500
Fig. 4.10 (b) Vorticity contours at time-step2000

4.2 Building model is at an angle (22.5^o) to the geometric axis.

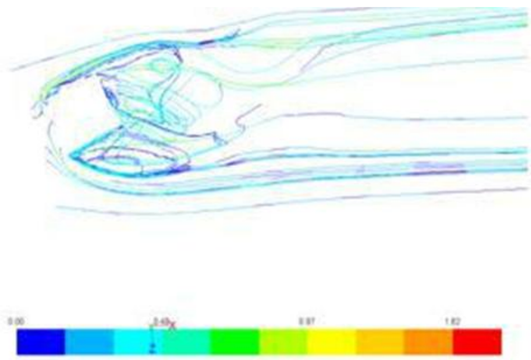


Fig. 4.11 Pathlines over building model oriented at 22.5°

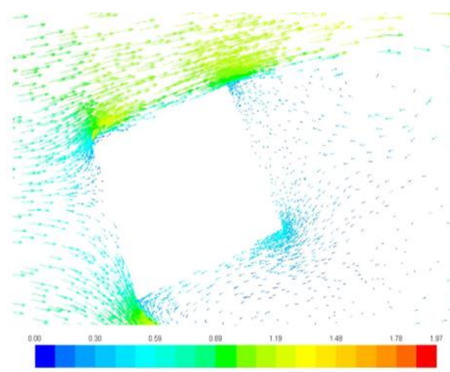


Fig. 4.12: Velocity vector for building model oriented at 22.5°

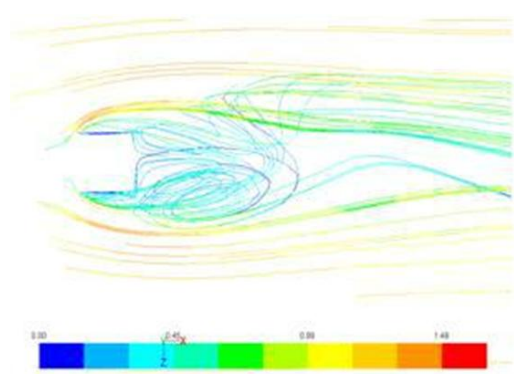


Fig. 4.13 Pathlines over building model oriented at 0°

V. CONCLUSION

After rigorous validation and testing of the numerical prediction of flow variables for a single building during the study we can predict the accuracy of flow behavior related to wind loading. By qualitative comparison of the flow patterns over the cubical building at two different orientations it is to be observed that the flow patterns have changed significantly for orientation 22.5° and hence the quantity of the wind effects in terms of the pressure distribution and turbulence intensity on the building model is expected to vary prominently. If the flow patterns shown in Fig 4.13 and Fig. 4.11 are compared then it shall be noted that the magnitude of velocity is higher by around 30-40 % from eachother at a specific location at the leewards of each building model. The validation of quantification of this difference shall be done in the future work with more strategic orientations. The orientation has an impact on the formation of separation over the building top surface and at the leewards. The vector plot shown in Fig. 4.12 also justifies the same logic at the location of leeward.

REFERENCES

[1]. Murakami, S. "Computational wind engineering", J. Wind Eng., 36: (P517-538) 1990

[2]. Tamura T and Miyagi T (1999) The effect of turbulence on aerodynamic forces on a square cylinder with various corner shapes. Journal of Wind Engineering and Industrial Aerodynamics 83: 135-145.

[3]. Murakami S, Mochida A and Hayashi Y (1990) Examining the k-□ model by

[4]. means of wind tunnel test and large- eddy simulation of the turbulence structure around a cube, Journal of Wind Engineering and Industrial Aerodynamics 35: 87-100.

[5]. Sakamoto S, Murakami S and Mochida A (1993) Numerical Study on Flow past 2D Square Cylinder by LES: Comparison between 2D and 3D Computations,

[6]. Journal of Wind Engineering and Industrial Aerodynamics 50: 61-68.

[7]. Rodi W (1993) On Simulation of Turbulent Flow past Bluff Bodies, Journal of Wind Engineering and Industrial Aerodynamics 46-47: 3-19.

[8]. Bouris D and Bergeles G (1999) 2D LES of vortex shedding from a square cylinder , Journal of Wind Engineering and Industrial Aerodynamics 80: 31-46.

[9]. Maharana SK, Ghosh AK (2004) A numerical study of flow over a sloping roof structure in tandem arrangement. Journal of Wind and Engineering 1: 53-59.

[10]. Murakami S(1990) Computational Wind Engineering, Journal of Wind Engineering and Industrial Aerodynamics 36: 517-538.

[11]. Qasim A, Maxwell TT, and Parameswaran S (1992) Computational prediction of flow over a 2-D buildings. Journal of Wind Engineering and Industrial Aerodynamics 41-44: 2839-2840.

[12]. Murakami S and Mochida A (1995) On Turbulent Vortex Shedding Flow past 2D Square Cylinder Predicted by CFD. Journal of Wind Engineering and Industrial Aerodynamics 54-55: 191-211.

[13]. Murakami S, Mochida A (1988) 3-D Numerical Simulation of Airflow Around a Cubic Model by means of the k-□ Model, Journal of Wind Engineering and Industrial Aerodynamics 31: 283-303.

[14]. Murakami S, Mochida A, Hayashi Y, and Sakamoto S(1992) Numerical study on velocity-pressure field and wind forces for bluff bodies by k-□ , ASM and LES, Journal of Wind Engineering and Industrial Aerodynamics 44: 2841-2852

[15]. Ng CW, Ko NWM (1995) Flow interaction behind two circular cylinders of equal diameter-a numerical study, Journal of Wind Engineering and Industrial Aerodynamics 54-55: 277-287

M.M. Venugopal "Computational modeling of wind flow and its effect on a cubical building using Large Eddy Simulation (LES) technique "International Journal of Engineering Research and Applications (IJERA) , vol. 8, no.7, 2018, pp.20-26