Indian Journal of Science and Technology, Vol 10(8), DOI: 10.17485/ijst/2017/v10i8/109197, February 2017

ISSN (Print) : 0974-6846
ISSN (Online) : 0974-5645

Investigation of Turbulence for Wind Flow over a Surface Mounted Cube using Wall Y+ Approach

Bibhab Kumar Lodh¹, Ajoy K Das¹ and N. Singh²

¹Department of Chemical Engineering, NIT Agartala, Barjala, Jirania, West Tripura - 799055, Tripura, India; bibhab123@rediffmail.com, akdas_72@yahoo.com
²Department of Aerospace Engineering, IIT Kharagpur, Kharagpur - 721302, West Bengal, India; nsingh@aero.iitkgp.ernet.in

Abstract

Objective: To find the best mesh based on wall y+ approach and to model turbulence flow numerically over a surface mounted cube so that appropriate mesh can be selected and thereby the best grid configuration can be found. Methods: The study uses three mesh configuration based on wall y plus to capture the turbulence and its complex characteristic for moderate high Reynolds number. The study depends on Reynolds number 40,000 which is based on height of the cube and bulk velocity and the numerical and analysis results are compared with the recognised published work on the flow past ground mounted prismatic body placed in a fully developed turbulent channel flow. The turbulence model used in this work is large eddy simulation incorporating Smagorinsky scheme. The investigation of near wall treatments is done for maximum wall y+=5 covering the viscous sub layer, maximum y+=23 the buffer region and maximum y+=40 resolving the log-law region. OpenFoam platform under Linux OS is used for the study. Findings: After completion of three numerical simulations based on three grids various results in terms of fluid parameters are discussed such as drag and lift, velocity profile, pressure profile, Q criteria etc. It has been found that the drag force varies from 1.13 to 1.38 and lift coefficients varies from 0.8 to 1.0. Strouhal number varies from 0.45 to 2.17. Maximum and minimum pressure values are also been found and is discussed in the result & discussion section of this paper. Application: The method described in the present study is very helpful in finding the best suitable grid for CFD analysis. As it is very well known fact that, if the grid is not good enough for any CFD study, the result of the simulation will be wayward. Finding the best grid or mesh is a time consuming process and also involves high cost. This method can minimize the cost and also saves lots of computational time.

Keywords: Buffer, Drag Coefficient,External Flow, LES, Lift Coefficient, Pressure Coefficient, Reynolds Number, Smagirinsky, Turbulence, Viscous, Yplus

1. Introduction

In Computational Fluid Dynamics (CFD) study the successful generation of best mesh for a problem domain is an important and integral part and a significant amount of project time and effort takes by this process. To determining the most appropriate mesh configuration various methods are used and among all Grid Independence Test is very fruitful and common way where different meshes are tested till the solution is independent of further mesh refinementsso that the numerical results can be compared with the bench mark tests and/or experimental published data. This in itself is a time-consuming process. Turbulence flows are very complex flow phenomenon and are highly affected due to the bounding walls because of the no-slip condition. Because of this large gradients are being generated in the variables in the viscous affects dominated region which is called none other than the viscous layer. The work were carried by many researchers on the various aspects to find the best mesh for the numerical investigations. One of these kinds of study was done by [1] for 2D wall bounded flow regime in the area of y+. But in cases there are unavailability of reliable experimental. here after [2] worked on a three-dimensional (3D) problem based on low Reynolds number of 1,870 taking cube height as
Investigation of Turbulence for Wind Flow over a Surface Mounted Cube using Wall Y+ Approach

the reference length and fluid bulk velocity and found that Spalart-Allmaras turbulence model with a mesh resolving near viscous sub layer is quite accurate with the benchmark experimental data. In their study two y+ ranges i.e., viscous sub layer and buffer layer were studied and the turbulent zone i.e. log-law region was not covered. They had suggested to use the Reynolds Averaged Navier-Stokes (RANS) models and also near-wall treatments as an inbuilt function in Fluent. In [3] had continued the work further in which a higher Reynolds number was included and the log-law region is also being analysed which in turn allows to analyse the total behaviour of turbulence numerical models and also near-wall treatment for all three wall y+:

(a) $y^+ \leq 5$: The viscous sub layer region in which velocity profile is supposed to be laminar and the wall shear is being dominated by viscous stress.

(b) $5 < y^+ \leq 30$: The buffer region in which both the turbulent shears as well as viscous stress dominate.

(c) $30 < y^+ \leq 300$: Fully turbulent zone or log-law region which correspond the region where only turbulent shear dominates.

The wall y+ [equation (1)] is a dimensionless number and it is used to determine the influences in the near wall cells whether they are laminar or turbulent and indicates the part of the turbulent boundary layer that it resolves.

$$y^+ = \frac{u_{ref}y}{\nu_{air}} \quad (1)$$

One of the most common and treated as the benchmark problem in the analysis of turbulent flow over a bluff body is a three dimensional cubical obstacles immersed in a turbulent channel. Despite of its simple geometry the flow phenomenon generates very complex flow structures and a number of irregular vortices and eddies which comprises of numerous sizes and configurations which are relevant to many important engineering application such as wind load on small or big architectural structures and cooling of electronics chip etc. Many famous researchers have worked in this benchmark problem such as [4] and [5] performed numerous experimental studies on cubical resistance mounted in the ground surface of a channel where a fully developed turbulent flow persists based on high 40,000 Reynolds number depending on cubical resistance height and bulk fluid velocity. A numerous number of numerical simulation and analyses exist on the same problem, such as two-layer turbulence models studied by [6], large eddy simulation (LES) by [7] and some kind of unsteady flow numerical simulation using Reynolds Averaged Navier-Stokes (RANS) employed by [8]. Unsteady Reynolds-Averaged Navier–Stokes equations (URANS) based model was performed by [9] in combination with the semi empirical Spalart–Allmaras turbulence model and found the horse shoe type vortex behind the cube and also found two dominant flow characteristic based on strouhal number. In [10] have validated their CFD results for wind flow past a wall mounted cubical obstacles in a channel where turbulent flow prevails and found the pressure coefficient which matches with our results. In this particular paper the study has been done to verify the same problem and compare the results for three mesh size based on wall y+ using Large Eddy Simulation in OpenFoam.[11]

2. Model Description

The computational domain shown in Figure 1 replicates the domain used by [1] in their numerical analysis which was same as the experimental setup of [2]. In their experimental setup, at the inlet of the domain turbulent flow was set with fully developed condition with $Re_H = 40,000$ which is based on height of the cube. No-slip boundary conditions were applied on all cubes faces. In the computational domain the top and the side walls were defined as symmetry since they are quite a distance apart from the cube and hence the chance of influence the flow characteristics are very minimal.

Figure 1. Computational Geometrical Domain.

Incompressible, 3D transient flow using Large Eddy Simulation (LES) with Smagorinsky scheme was implemented for the study. The unsteady flow behaviour
which has been performed in the present study is done with the LES methods since LES proved to be good agreement with real time experimental result. Whereas the RANS approach could not produce the complex vortex structure within the near wall vicinity because of the boundary layer development in front of the cube. The blockMeshdict utility is used in OpenFoam environment to create the initial geometry grid defining the problem and discretize the domain and there after the snappyHexMesh was employed to refine the initial grid in three stages to attach the cube model which is made in FreeCad while OpenFoam3.1 is employed to discretize and solve the governing equations given in. Figures 2, 3 and 4 show the final grids for the computational study with different mesh configurations used for the present study. The details of all the three meshes are given in the Table 1.

![Figure 2. Mesh 1 (Fine).](image)

![Figure 3. Mesh 2 (Coarse).](image)

![Figure 4. Mesh 3 (Coarser).](image)

The boundary condition and the respective values of the parameters at the boundary are tabularized in Table 2. The inlet wind velocity is taken as (0.6, 0, 0) i.e. the bulk fluid velocity $U_b = 0.6$ m/s in horizontal direction and the size of the cube is taken as $H=1.0$ m and hence the $Re_H = H U_b / \nu_{air} = (1.0*0.6)/1.5*10^{-05} = 40000$. The PISO algorithm was used which is a transient solver to run the simulation. The time step was taken as $\Delta t = 10^{-4}$ and the end time was 0.7. The time step was taken very small so that the Courant number should be less than unity which ensures the good stability of the solver. The numerical simulation time for the three meshes were 90 hours for mesh 1 (Fine), 48 hours for mesh 2 (coarse) and 17 hours for mesh 3 (coarser) respectively for dual core Pentium I3 processor with 2 GB of RAM.

### 3. Results

The $y+$ values with respect to the cube near wall for the three considered meshes are ≈ 5, 23, and 40 (as given before) and they correspond the resolution in the region as viscous sublayer, buffer layer and log-law zone, respectively which shown graphically in Figure 5.

From the Figure 5 it can be seen that for the finer mesh (Mesh 1) the $Y+$ varies from 5 to 2, for coarse mesh(2) $Y+$ varies from 23 to 6 and for coarser mesh (Mesh 3) $Y+$ varies from 40 to 31 and all the shown ranges are within the range.

| Mesh No | Mesh type | Initial Grid (X*Y*Z) | Total no of cells after 3 stage of refinement | Wall adjacent cell with respect to cube |
|---------|-----------|-----------------------|---------------------------------------------|----------------------------------------|
| 1       | Fine      | 32*18*10              | 20,37,360                                   | 0.00356H                               |
| 2       | Coarse    | 30*14*8               | 12,16,280                                   | 0.00426H                               |
| 3       | Coarser   | 22*8*6                | 4,00,286                                    | 0.00553H                               |
Investigation of Turbulence for Wind Flow over a Surface Mounted Cube using Wall Y+ Approach

Table 2. Boundary Conditions

| Parameters/Domain boundary | Lower Wall | Cube | Inlet | Outlet | Front | Back | Upper Wall |
|----------------------------|------------|------|-------|--------|-------|------|------------|
| P                          | Zero Gradient | Zero Gradient | Zero Gradient | Fixed Value Uniform 0 | Symmetry | Symmetry | Symmetry |
| U                          | Fixed Value (0.6,0,0) | Fixed Value (0.0,0) | Fixed value, Uniform (0.6,0,0) | Inlet Outlet Fixed value (0.6,0,0) | Symmetry | Symmetry | Symmetry |
| nuSgs                      | Zero Gradient | Zero Gradient | Zero Gradient | Zero Gradient | Symmetry | Symmetry | Symmetry |
| k                          | kqRWallFunction, Uniform 0.24 | kqRWallFunction, Uniform 0.24 | Fixed Value Uniform (0.24) | Inlet Outlet Uniform (0.24) | Symmetry | Symmetry | Symmetry |
| nut                        | nutUSpalding WallFunction, Uniform 0 | nutUSpalding WallFunction, Uniform 0 | Calculated Uniform 0 | Calculated Uniform 0 | Symmetry | Symmetry | Symmetry |
| nuTilda                    | Fixed Value Uniform (0) | Fixed Value Uniform (0) | Fixed Value Uniform (0.05) | Inlet Outlet Uniform (0.05) | Symmetry | Symmetry | Symmetry |

Figure 5. Cube Yplus.
Mean stream wise velocity is presented for different non dimensional X distances with X=0 which is being set at the front of the cubical obstacle.

Figure 6. Comparison of Velocity Profile (y/H vs U/Ub) for Three Meshes at x/H=-1.

Figure 7. Comparison of Velocity Profile (y/H vs U/Ub) for Three Meshes at x/H=-0.01.

Figure 8. Comparison of Velocity Profile (y/H vs U/Ub) for Three Meshes at x/H=0.5.

Figure 9. Comparison of Velocity Profile (y/H vs U/Ub) for Three Meshes at x/H=1.05.

Figures 6 and 7 compare the mean stream wise velocity profiles in the line of symmetry X/H= -1 and X/H= -0.01 which are in the upstream section of the domain. From the figures it can be observed that in figure 6 there is no variation in the profile for the three meshes since the point is quite far from the obstacle and hence there is no such effect of the obstacle but in figure 7 which is very close to the cube the profile are disturbed and in case of fine mesh (mesh 3) representing the red triangles are more chaotic since the because of the fine grid the resolution the capture is much more prominent than the two types of coarse meshes. Figure 8i.e., in the line of symmetry X/H=0.5 which is starting from the top of the cube and can be understood from the Y/H axis which varies from 1 to 2 instead of 0 to 2, in case of the fine mesh the boundary layer formation is much more enhanced than the other two meshes.Figure 9 corresponds to velocity profile in the symmetry line of X/H=1.05 which is very close to the cube back face and in the downstream zone of the geometry. Here again it can be observed that for the fine mesh the velocity profile is much more developed than the two coarse meshes and because of the inference of the cube obstacle the profile is disturbed upto a length of Y/H =0.7 which corresponds to the generation of vortex at the back side of the obstacle.

When comparing Figure 9 with Figure 7 which is almost a mirror image symmetry line in the opposite side of the cube, it can be visualised that at the front both the curves shapes are almost similar. The only difference can be observed at the bottom of the curve which is due to the small vortex generated at the bottom of the front face of the cube because of the change in fluid direction.
Figures 10 to 13 correspond to the velocity profile (mean streamwise) at the line of symmetry \( X/H = 2, 3, 4 \) and 5 respectively. Since all those symmetry lines are far away from the obstacles it can be seen that the inference is not affecting the velocity profile and which in turn does not affect the development of boundary layer. In all the four figures it can be observed that in case of fine mesh the velocity profile develop earlier than the other two meshes. As we move further it is prominent that the finer mesh resolution and capacity of capture is more rigorous than the coarse meshes. In case of coarse meshes there is almost no difference due to the no of cells in the domain. As it is well cited that the boundary layer corresponds to a value of \( U/Ub = 0.99 \) and in Figure 12 it can be observed that the thickness is 0.8 & 0.6 respectively for the fine and coarse meshes and 1.8 in the Figure 10. Hence the inference can be drawn that the boundary layer thickness is higher for the finer mesh (Mesh 1) and as we move in more downstream section the thickness decreases. Further close to the cube wall for all the meshes the boundary thickness is almost same and more than the downward portion.

**Figure 10.** Comparison of Velocity Profile (\( y/H \) vs \( U/Ub \)) for Three Meshes at \( x/H = 2 \).

**Figure 11.** Comparison of Velocity Profile (\( y/H \) vs \( U/Ub \)) for Three Meshes at \( x/H = 3 \).

**Figure 12.** Comparison of Velocity Profile (\( y/H \) vs \( U/Ub \)) for Three Meshes at \( x/H = 4 \).

**Figure 13.** Comparison of Velocity Profile (\( y/H \) vs \( U/Ub \)) for Three Meshes at \( x/H = 5 \).

**Figure 14.** Comparison of Pressure Coefficient (\( Cp \) vs. Distance over Cube) for Three Meshes.
Figure 14 gives pressure coefficient comparison for three meshes and it can observed that in case of mesh 1 & 2 the graphs are almost similar but for the third mesh which corresponds to the turbulent zone with y+ more than 30 gives a higher negative Cp. This is may be due to the reason that for a higher y+ value the viscous sublayer and the buffer layer are not resolved. In case of cube wall where the viscous region dominates gives a little wayward result. It was noticed that the location of maximum positive pressure coefficient (CpW) was at a distance 0.55H from the ground for fine mesh (mesh1) that is nothing but the stagnation point and the registered value was 1.1579, on roof top the maximum negative pressure (CpR) value was -1.75 (Mesh 1) and as for the leeward façade the maximum negative pressure (CpL) value was -0.27. The values for other two meshes given in Table 3.

**Figure 15 a,b,c.** Streamline Representation for Mesh 1.

**Figure 16 a,b,c.** Streamline Representation for Mesh 2.
Figures 15a,b,c, 16a,b,c& 17a,b,c give the streamline representation coloured by velocity in various camera position for the better visualisation for fine(mesh1), coarse(mesh 2) and coarser (mesh 3) respectively. The red colour shows the high velocity whereas the blue shows the lowest velocity. Comparing the figure no 15b, 16b and 17b it can be seen that there is symmetrical horse shoe like vortices generating in the backward side of the cube. The vortex size and shape in figure no 15b is more prominent because of the highest resolution of mesh comparing to figure no 16 and 17b. Figure no 15a, 16a and 17a show the stagnation points at the front windward face and also the length of reattachment at the roof of the cube. As the mesh become coarser the stagnation point length seems to be increase as shown in the figures but the reattachment length does not vary too much with the mesh. Figure 15c, 16c& 17c show the reattachment lengths in the backward side of the cube and it can be seen that with the higher resolution of the mesh the reattachment length varies as shown in the Figures (The results are tabulated in Table 3).

Figures 18 -20a,b,c give the Q criteria and vortex generation with the wall bounded streamline coloured by pressure. In the figures 18b, 19b & 20b it can be seen that significant vortex generates at the roof of the cube and at the back portion whereas small vortex generates at the bottom front and at the bottom back side of the cube which in fact confirms the work done by6.
Figures 21a, b and c show the pressure contour coloured by pressure in the z normal of the domain where the slice is coloured by velocity. Comparing the figures it is apparent that due to the high no of cells and better grid the Figure 21a gives the best captures of pressure contour. Hence it can be said that the pressure varies sufficiently large with the grid.

Figure 22 gives the comparison of drag coefficient and it can be observed that for the mesh1 the drag coefficient is high and almost 1.38 and for the other two i.e. mesh 2 & 3 they are almost same. The curves are fitted in MATLAB and observed that for the fine mesh the drag coefficient is a third order polynomial function of time whereas for other two meshes (2 and 3) it is a fifth order polynomial function of time. But the curves fitted better for the low resolution grid since the R² value are much higher as 0.992 & 0.951 for mesh 2 & 3 compare to 0.878 for mesh 1. Hence it can be predicted that for medium size of grid point or the grids in the buffer zone of Yplus the drag coefficient can be predicted by data fitting whereas for higher grid points i.e., in the viscous zone of Yplus value the drag may not be predicted correctly.

Figure 23 show the lift coefficient comparison and it can be seen that for mesh 1 & 2 curve fitted nicely with high R² values of 0.929 & 0.993 respectively whereas for the more coarser mesh it gives a poor result with R² value as low as 0.5.

For finding the strouhal number vortex shedding frequency is necessary and hence Fast Fourier Transformation (FFT) of the lift coefficient data has been performed to get the dominant frequency. The vortex shedding frequency is 1.3 for mesh 1 & 2 and 0.27 for mesh 3 and generate the corresponding strophe number (FD/Ub) as 2.17 for mesh 1 & 2 and 0.45 for mesh 3. In found the values of reattachment lengths for the various turbulence models and the values are in good agreement with our LES results. In worked on wind flow over cube in turbulent channel flow and they have found the reattachment length over the cube roof which closely resembles our findings. Their founded stagnation point also gives a very good agreement with our findings.

Tabulated results
4. Conclusions

Behaviour of flow of wind past a ground mounted cubical obstacle in turbulence flow domain with mapped inlet flow was investigated using CFD simulation and the results were compared to published results from various researcher to identify the reliability of using a certain set of simulation variables for investigating wind flow. The simulation was carried out for three different meshes based on Yplus cube near wall cell height. Large Eddy Simulation (LES) is employed with smagorinsky scheme to run and results were obtained and compared to the reviewed results using different tools for assessing similar wind flow problems. The inference can be drawn that qualitatively and quantitatively the results are in good agreement and consistent with other reviewed results as all the flow features were captured in the CFD simulation. In addition all the values of the specific lengths of the flow were within the ranges of the reviewed results. At the end it can be summarize that the LES can be a suitable alternative for the turbulence modelling for flow over bluff body and the y+ approach to find suitable grid can be proved fruitful for the good CFD result.

5. Acknowledgement

The author would like to acknowledge the second and third author of this paper viz. Dr Ajoy Kumar Das & Dr. N. Singh for their constant encouragement and support without which this work would not have seen the day light. The author also would like to give his sincere thanks to the Department of Mechanical Engineering and Department of Chemical Engineering NIT Agartala and all their staffs for their support towards the completion of the work and to allow the author to use the resources. At last the author would also like to show his gratitude to his family for constant motivation and support towards the completion of this work.

6. References

1. Salim SM, Cheah SC. Wall Y+ Strategy for Dealing with Wall-bounded Turbulent Flows. Hong Kong: IMECS 2009 Proceedings of the International Multi Conference of Engineering and Computer Scientists. 2009 Mar; 2:1-6.
2. Ariff M, Salim SM, Cheah SC. Wall Y+ Approach for Dealing with Turbulent Flow over a Surface Mounted Cube: Part I - Low Reynolds number. Australia, Melbourne: CSIRO: Proceedings International Conference on CFD in the Minerals and Process Industries. 2009 Dec.
3. Ariff M, Salim SM, Cheah SC. Wall Y+ Approach for Dealing with Turbulent Flow over a Surface Mounted Cube: Part 2 – High Reynolds number. Australia, Melbourne: CSIRO: Seventh International Conference on CFD in the Minerals and Process Industries, Melbourne. 2009 Dec; p. 9-11.
4. Martinuzzi R, Tropea C. The Flow around Surface Mounted Prismatic Obstacles Placed in a Fully Developed Channel Flow. Transaction ASME Journal Fluid Engineering. 1993; 115(1):85-91. Crossref
5. Hussein HJ, Martinuzzi RJ. Energy Balance for Turbulent Flow around a Surface Mounted Cube Placed in a Channel. Physics of Fluids. 1996; 8(3):764-80. Crossref
6. Lakehal D, Rodi W. Calculation of the flow past a surface of a mounted cube with 2-layer turbulence model. Journal of Wind Engineering and Industrial Aerodynamics. 1997; 67-68:67-68. Crossref
7. Shah KB, Ferziger JH. A fluid mechanicians view of wind engineering: Large eddy simulation of flow past a cubic obstacle. Journal of Wind Engineering and Industrial Aerodynamics. 1997; 67-68:211-24. Crossref
8. Iaccarino G, Ooi A, Behnia M. Reynolds averaged simulation of unsteady separated flow. International Journal of Heat and Fluid Flow. 2003; 24:147-56. Crossref
9. Isaev SA, Lysenko DA. Calculation Of Unsteady Flow Past A Cube On The Wall Of A Narrow Channel using Urans and The Spalart–Allmaras Turbulence Model. Journal of Engineering Physics and Thermo Physics. 2009; 82(3):488-95. Crossref
10. Abobela I, Hamza N, Dudek S. Validating CFD Simulation Results: Wind flow around a surface mounted cube in a tur-
bulent channel flow. Peru, Lima: PLEA2012 - 28th Conference, Opportunities, Limits & Needs Towards an environmentally responsible architecture. 2012 Nov; 7-9.

11. Open Foam. Date Accessed: 2016: Available from: http://cfd.direct/openfoam/user-guide/turbulence/#x36-2640007.2.2.

12. Krajnovic S, Davidson L. Large-eddy simulation of the flow around a surface-mounted cube using a dynamic one-equation sub grid model. Beggel House, Inc.: New York: The First International Symposium on turbulence and Shear Flow Phenomena. 1999.

13. Pope SB. Cambridge University Press: Turbulent Flows, Second Edition. 2000.

**Nomenclature**

- $H$ Cube height (m)
- $u_{\text{fluid}}$ Instantaneous Velocity (ms$^{-1}$)
- $U_b$ Bulk Fluid Velocity (ms$^{-1}$)
- $X$ Distance along horizontal in streamwise direction (m)
- $XR_{\text{roof top reattachment length}}$ (H)
- $XF_{\text{back face reattachment length}}$ (H)
- $Y$ Distance along normal to direction of the wall (m)
- $Z$ Spanwise parallel direction (m)
- $Re_H$ Reynolds Number (Dimensionless) which is given by (= $Hu_b/\nu_{\text{air}}$) (Based on cube height)
- $y^+$ Dimensionless distance to the wall
- $St$ Strouhal number
- $C_D$ Drag coefficient
- $C_L$ Lift coefficient
- $C_{PW}$ Maximum positive pressure coefficient in the windward facade
- $C_{PR}$ Maximum negative pressure coefficient in the cube roof
- $C_{PL}$ Maximum negative pressure coefficient in the leeward facade