Comparison of RANSE and LES for Wind Structural Interaction

L.H.S.U. Balasooriya, B.R.G.A. Krishantha and K.K. Wijesundara

Abstract: Popularity of tall buildings increases due to technological advances, but Code-based designs are conservative and not reliable for structures beyond 200 m height. Conducting a wind tunnel test is expensive and the resources are limited. Hence, numerical modelling is an alternative where Reynolds Average Navier Stokes Equation (RANSE) and Large Eddy Simulation (LES) are such numerical techniques.

In recent Computational Fluid Dynamic (CFD) studies, it could be observed that results have considerable deviations in flow separating and high turbulent areas. Hence a structured mesh was used here to perform mesh refinement in such critical locations to refine only the required areas.

The objective of this study is to compare the RANSE and LES in interpreting the wind structural interactions using a structured mesh arrangement. This study will be limited to tall buildings of height less than 200m, rectangular in shape. Hence Commonwealth Advisory Aeronautical Council (CAARC) standard building model was used as the subject and Simulation results will then be compared with the values of the wind tunnel test available in the literature.

It could be observed that the results obtained by RANSE simulation for a structured mesh has a deviation less than 10%. But natural variations of the wind are more clearly indicated with LES with a deviation even less than 3% in turbulent regions.

Keywords: Wind structural interaction, Tall building, RANSE, LES, Structured mesh, Optimization

1. Introduction

Due to the high urbanization, increasing population, and limitation of space, the demand for high-rise buildings is increasing day by day. Due to the higher flexibility of taller buildings, they would become more sensitive to wind loading. Therefore, the prediction of wind structure interactions quantitively would become one of the major concerns in design.

The code of practices in the estimation of wind loads on structures are not reliable in economical designing tall buildings due to the limitation imposed in the code-based procedures [1]. As an alternative, wind tunnel tests are widely used to predict the wind-induced action. However, a complex shape of the building and complex wind patterns, make the wind tunnel simulation time consuming and expensive to conduct. Therefore, a need of reliable effective numerical simulation to predict the wind flow around a tall building arises. Furthermore, for tall and complicated structures, the existing wind codes would produce more over-predicted or conservative results.

If it is possible to find out the most accurate responses around the perimeter of the building it is possible to make the building designs more optimized. Xu et al**.** [2] show that the conventional wind tunnel testing, high frequency base balance (HFBB), systematic multiprocessors (SMPSS) and advanced wind tunnel test techniques (force balance test techniques, vibration test techniques) are not fully capable of predicting the wind induced response.

The natural winds are highly turbulent so a complicated environment is created around the building by wakes and flow separations which result increating different pressure zones around the building that would impose loads on the structure and its facades [3].

There are three main wind induced responses of a tall building, along wind response, cross wind response and torsional response. Along wind

Mr. L.H.S.U. Balasooriya, Student Member of IESL, Department of Civil Engineering, Faculty of Engineering, University of Peradeniya,
Email:e16038@eng.pdn.ac.lk
https://orcid.org/0000-0005-8670-8353
Mr. B.R.G.A. Krishantha, Student Member of IESL Department of Civil Engineering, Faculty of Engineering, University of Peradeniya, Sri Lanka.
Email:e16189@eng.pdn.ac.lk
Eng.(Prof.) K.K. Wijesundara, AMIE (Sri Lanka), B.Sc. Eng. (Hons.) (Peradeniya), M.Sc. (Pavia), Ph.D. (Pavia), Professor of Civil Engineering, Department of Civil Engineering, University of Peradeniya.
Email:kushanw@pdn.ac.lk
https://orcid.org/0000-0002-4174-8707

ENGINEER

action is induced by the fluctuating component and the mean component in wind actions. The cross-wind actions are induced by vortex shedding, incidence turbulent mechanism and higher derivative of cross winds [4]. Torsional effect can be created by above two wind-induced effects due to the lateral load-resistant characteristics of the building such as the asymmetrical arrangement of the shear core in the geometry of the building. Combination of along wind response together with cross wind can also induce a torsion in the building structure.

For such a complicated phenomenon, conducting a wind tunnel test requires a lot of effort in creating the scaled-down model of the prototype to have proper similitude between model and prototype [5].

This shows the requirement of a reliable numerical simulation to simulate the existing worst conditions and get accurate results for such complex scenario. In modern wind engineering, Reynolds Average Navier Stokes Equations (RANS)and large eddy simulation are commonly used, and they can be used for simulations in computational fluid dynamic (CFD) software such as ANSYS Fluent. In early 2000s, CFD was not popular due to the unavailability of computers with the required power to conduct complex simulations.

With the development of computational capabilities, it is possible to implement RANSE and large eddy simulations to obtain numerical solutions using CFD codes which are widely used in the modern field. CFD analysis may be a lot easier than the wind tunnel tests, but it consumes a considerable amount of computational power even with modern-day computers thereby making it computationally expensive. Hence the requirement for optimizing the computational power demand required by a numerical model should be paid attention.

For unstructured meshes in previous studies, it could be observed that there are large deviations in results at the flow separating regions and wake regions. Hence, for RANSE, the results are quite unreliable at higher turbulent regions and flow separating sharp edges [1,12,13]. This is because the kinetic energy of wind is not resolved properly by the mesh arrangement, hence it creates progressive errors till the final results. Hence by implementing a structured mesh by considering

the way of air flow behaviour the layered domain has to be created to model all the shear layers accurately in order to obtain more accurate results in high turbulent areas.

This study mainly focuses on improving the RANSE simulation and comparing it with the large eddy simulations using a structured mesh arrangement to address drawbacks that have been in simulations in RANSE recent past. For validation, these results are compared with the wind tunnel test data taken from Melbourne in 1980 [6]. Since this wind tunnel test has been carried out for several input conditions for a CAARC standard building model which simulates different cases. Therefore, it is popular among many CFD researchers for validating their numerical models.

2. Reynolds Averaged Navier Stokes Equation

Time averaging the Navier Stokes equation for incompressible fluids, an average flow representation of the actual flow is done by RANSE. The entire turbulent flow field is represented using time mean value of the flow. In order to take time average, the momentarily values are decomposed into mean and fluctuating values. Mathematically that can be written as in Equation [1].

$$u = \bar{u} + u', v = \bar{v} + v', w = \bar{w} + w' \qquad \ldots 1$$

where $\bar{u}, \bar{v}, \bar{w}$ are mean velocity components and u', v', w' are fluctuating velocity components.

Time average of fluctuating velocity should be zero. It can be written as in Equation [2].

$$\bar{u}' = 0 \ , \bar{v}' = 0 \ , \overline{w' = 0} \ \bar{p}' = 0 \qquad \ldots 2$$

Substituting these equations in the NS equation and taking time average will result in Equation [3].

$$\left(\overline{\frac{\partial(\bar{u}+u')}{\partial t} + \frac{\partial(\bar{u}+u')^2}{\partial x} + \frac{\partial(\bar{u}+u')(\bar{v}+v')}{\partial y} + \frac{\partial(\bar{u}+u')(\bar{w}+w')}{\partial z}} \right)$$
$$= F - \frac{\partial(\bar{p}+p')}{\partial x} + \mu \left(\frac{\partial^2(\bar{u}+u')}{\partial x^2} + \frac{\partial^2(\bar{u}+u')}{\partial y^2} + \frac{\partial^2(\bar{u}+u\rho')}{\partial z^2} \right) \qquad \ldots 3$$

By simplifying the above equation and final time averaged Navier Stokes equations (RANS) for all three directions (x,y,z) can be expressed in the following tensor form as given in Equation [4].

$$\frac{D\bar{u}_l}{Dt} = F_i - \frac{\partial \bar{p}}{\partial x_i} + \mu \Delta \bar{u}_l - \rho \left(\frac{\partial \overline{u'_l u'_j}}{\partial x_i} \right) \qquad \ldots 4$$

After this simplification, the additional term in the NS equation is called Reynolds stress, which arises due to the fluctuating velocity component. To resolve this term, eddy viscosity stress models are used.

2.1 Turbulence Models

There are many turbulence models based on the Boussinesq's hypothesis in Equation [5] that is used in CFD software ANSYS fluent.

$$-\rho u'v' = \mu_t \frac{\partial u}{\partial y} \qquad \text{...5}$$

Where μ_t is called eddy viscosity term.

The Reynolds number in the fluid domain was considered and it was measured at a value of 39.5×10^6 and with results it could be observed that the buffer region is more than 5 m for this case, hence k-ε model was used in this study as the turbulence model for RANSE simulation. Furthermore, the literature shows more reliable results for wind flows with a high Reynolds number, once the k-ε model is used [8].

This can be obtained from two transport equations for turbulent kinetic energy (k) and dissipation rate of turbulence kinetic energy (ε). This model is represented by Equations [6] and [7].

$$\frac{\partial(\rho k)}{\partial t} + \nabla.(\rho U k) = \nabla.\left[(\mu + \frac{\mu_t}{\sigma_k})\nabla k\right] + P_k \qquad \text{...6}$$
$$+ P_b - \rho\epsilon + S_k$$
$$\frac{\partial(\rho \epsilon)}{\partial t} + \nabla.(\rho U \epsilon) = \nabla.\left[(\mu + \frac{\mu_t}{\sigma_\epsilon})\nabla\epsilon\right]$$
$$+ C_1 \frac{\epsilon}{k}(P_k + C_3 P_b)$$
$$- C_2 \rho \frac{\epsilon^2}{k} + S_\epsilon \qquad \text{...7}$$

where the C_1, C_2, C_3 are model coefficients, those who vary between models.

3. Large Eddy Simulation

In large eddy simulation, the kinetic energy is resolved by an energy cascade diagram and the cell sizes should be determined prior to the formulation of the CFD grid. A good LES model should resolve more than 80% of the turbulent kinetic energy density at a certain point.

To resolve this amount of turbulent kinetic energy, the minimum grid length of a cell should be at least 1/5th of the integral length scale. But some numbers of eddies are still not modelled with this initial mesh. Therefore, some mesh refinement is required to capture those eddies and improve the accuracy of the results.

Subgrid-scale models in FLUENT also use the Boussinesq's hypothesis to compute the subgrid-scale turbulent stresses using an eddy viscosity approach in this model. An additional stress term is applied to dissipate the eddies just larger than mesh size as shown in Equations [8] and [9].

$$\frac{\partial p}{\partial t} + \frac{\partial(\rho U_j)}{\partial x_j} = 0 \qquad \text{...8}$$

$$\frac{\partial(\rho U_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j)$$
$$= -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{sgs})$$
$$\text{...9}$$

To calculate τ_{sgs}, an eddy viscosity approach is used as given in Equation [10].

$$\tau_{sgs} = 2\rho v_{sgs} S_{ij}^* - \frac{2}{3} k_{sgs} \delta_{ij} \qquad \text{...10}$$

where,

$$S_{ij}^* = \frac{1}{2}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_j} - \frac{\partial U_k}{3\partial x_k}\delta_{ij}\right) \qquad \text{...11}$$

Now to calculate v_{sgs}, Equation [12] can be used.

$$v_{sgs} = l_0^2 * \sqrt{S_{ij}S_{ij}} \qquad \text{...12}$$

where l_0 is called the sub grid length scale which is similar to the mixing length used in the RANS equations. The sub grid length scale can be determined using the following Equation [13].

$$l_0 = C_s \Delta \qquad \text{...13}$$

where *Cs* is the Smagorinsky coefficient. The value of *Cs* is between 0.1-0.2. In ANSYS FLUENT it is taken as 0.1.

4. Model Development

4.1 Computational Domain and Mesh Arrangement

The dimensions of the building model used in the computational domain is the full-scale rigid model of the Commonwealth Advisory Aeronautical Council (CAARC)standard building as shown in Figure 1. The CAARC standard tall building model is a rectangular prismatic body with flat surfaces, without parapets, and all surfaces contain no geometric disturbances as defined in Melbourne1980 [6]. The building has a height of 180 m, a length of 30 m and a width of 45 m.

The computational domain should be selected as same as in wind tunnel that has been created to eliminate the flow obstacle effect discussed in Murakami [9]. It is the practice in wind tunnel tests to limit the blockage effects to facilitate the boundary layer development on wind tunnel walls without creating any turbulences at the test subject level.
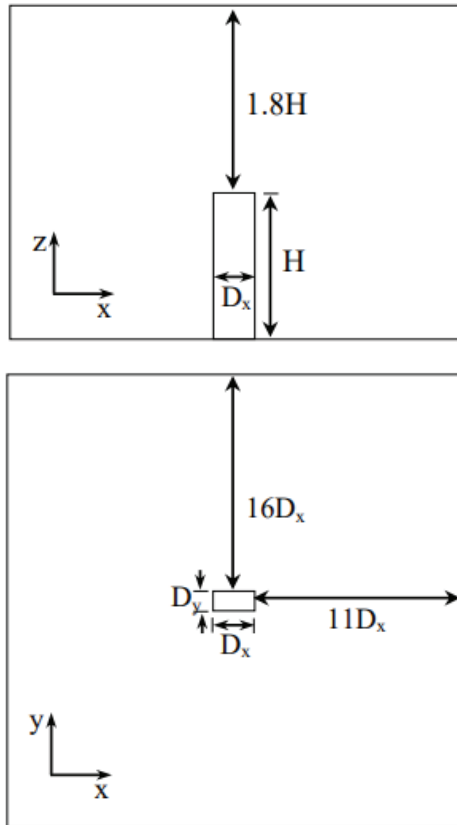

Figure 1 - Domain used for the Simulation

Choi et al. [10] explain the importance of selecting the inlet size to limit the blockage effect less than 5% in a wind tunnel. Also, the guidelines of Huanget al. [11] were considered in selecting the computation domain so that in the domain, the upstream length is more than 6 times the width of the building, while the space on either side of the building is more than 8 times the width of the building. Furthermore, the domain height is more than 2 times the height of the building. These guidelines have been followed by most of the CFD simulations that have been done in the recent past to come to a better agreement in validating the results from numerical simulation with wind tunnel testing.

The CAARC building models have been simulated in the wind tunnels by the City University, University of Bristol, Monash University, National Aeronautical Establishment (NAE), and National Physics Laboratory (NPL). These tests have been done

for different scale models with different power law exponents, turbulent intensities, and incident angles (β). For comparison, zero incident angle ($\beta=0$) has been used since it has highest values in the dataset compared to the other incident angles. The wind tunnel test provides the normalized pressure at 2/3rd height of the building by observations made in 20 tapping points around the perimeter of the building. These test results were used for the comparison of numerical results.
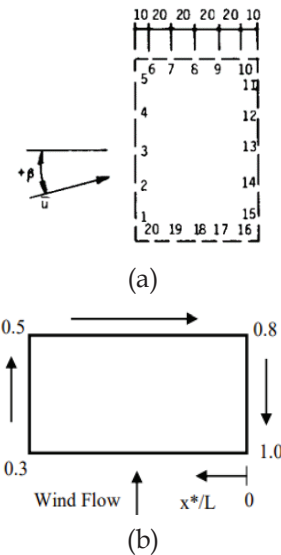

Figure 2 - Defined Pressure Tapings and incident Angle (a), and Definition of X/L (b) in Melbourne

The domain was created by ANSYS workbench space claim as a structured mesh consisting of 4 layers. Structured mesh arrangements give the opportunity to assign regions and desired element sizes and, shapes at places of vortices, flow separation, and boundary layer development hence this will facilitate modelling more turbulent kinetic energy. These layers were arranged in a way that more volume of the layers lies in rear side of the building. This favours predicting more accurate behaviour in the rear of the building where the turbulences are very high after interacting with the building. The layers are then meshed separately by defining smaller elements in layers at the proximity of the building and larger elements in layers away from the building progressively, and combined by the cut cell method which will allow the interconnection of regions with different sizes using transition layer.

The mesh arrangement is shown in Figure 3. The most inner layer was 0.5 m while 0.9 m, 2.5 m, and 10 m size hexahedral elements were placed thereafter respectively, in rest of the layers to

ensure the continuity of the domain contact regions were defined and coupled for energy transition. Inflation layers and sharp angle edges were defined for further smooth interactions between solid walls and wind. This method facilitates more manual interactions, so the programmer can manually define the regions that it wishes to do mesh refinement hence a large computational power can be saved compared to automatic meshing which assigns more elements throughout the entire domain unnecessarily. Number of elements generated in this mesh was 4.13×10^6.
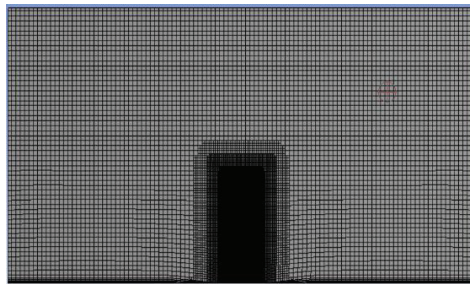
(a)
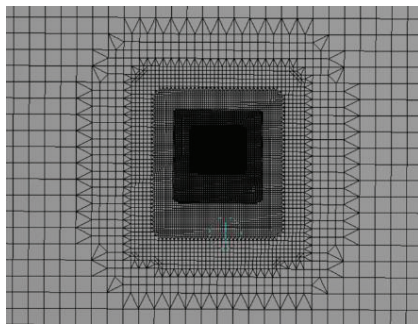
(b)

Figure 3 - (a) Sectional View of the Structured Mesh used (b) the Cut Cell Assembly of Layers

4.2   Boundary Conditions
Same boundary conditions used in the wind tunnel test were used in numerical models to obtain a better agreement in the final comparison of results. The power law exponent wind profile of NAE A, NAE B, and western with exponent 0.28 have been used in the simulation. The velocity profile was defined at the inlet face by a user-defined function in ANSYS fluent to create a wind profile for x direction varying along the y-axis. Velocity components in the y and z directions are given zero at the inlet to create the power law wind profile in the Equation [14].

$$\frac{U}{U_H} = \left(\frac{Z}{Z_H}\right)^{\alpha}$$

...14

where U is the velocity in the x-direction at height Y, U_H is the maximum velocity of the domain 15 m/s, Y_H is the height of the domain and a being the power law exponent 0.28. Incompressible fluid domain density was taken as 1.225 kg/m³ and viscosity was taken as 1.79 x 10-5 kg/m.s at 25 °C. Zero pressure was given at the outlet face preventing any reverse flow. The ground and faces of the building areno-slip boundaries while the other boundaries of the domain were defined to be slip boundaries so that there will be no interactions of wind in wind tunnel faces.

4.3   Turbulent Solver Control
For the comparison of RANSE and LES, both simulations were carried out at 500-time steps each of 0.5s. Hence the total duration of analysis is 250s, which is enough for the flow to attain its steady state. Each time step contains 30 iterations, enough for converging results to an accuracy of 10-7 at the end of each time step for pressure, velocity components, and energy. For RANSE, numerical simulations were carried out using k-ε stress modal and for the LES Smagorinsky-liliy model was used with the higher order term relaxations which helps more quick convergence of results.

5.   Results

After the simulations, non-dimensional pressure distribution along the perimeter at 2/3H byRANSE and LES, taken from the structured mesh, was compared with the wind tunnel test results of Melbourne1980 [6] and results that were obtained in past studies using unstructured meshes [12,13]. Figures 4 and 5 show the pressure and velocity contours and streamlines for RANS and LES, respectively.

5.1   Comparison of Normalized Pressure
The numerical simulation produces pressure and velocity values at the nodes of the elements. The pressure values around the perimeter of the building at 2/3rd height can be used to derive the non-dimensional pressure (Normalized pressure) to compare with the wind tunnel test. The non-dimensional pressure is calculated by Equation [15] and can be plotted against the X/L as defined in Figure 2. For comparison purposes, RANSE and LES were carried out in the same mesh at the same input parameters as explained above. Figure 6 shows the distribution of pressure coefficients around the perimeter at 2/3rd height.
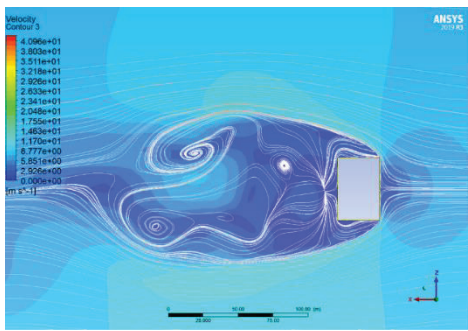
$$\text{Normalized pressure } (P') = \frac{p}{\frac{1}{2}\rho U_H^2} \qquad \dots 15$$

Comparing the results, RANSE was able to overcome the large deviation of results that were observed at the points of flow separation and sharp edge [12, 13] as shown in Figure 7. The result of RANSE is now more accurate on the windward side, rear side and side of the building. This is mainly because of the structured mesh arrangement that was used in this simulation is in a way to model the shear layers effectively.
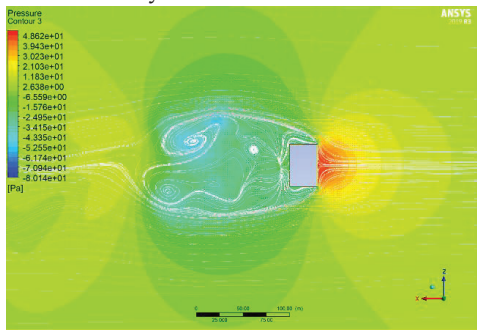
(a)

(b)

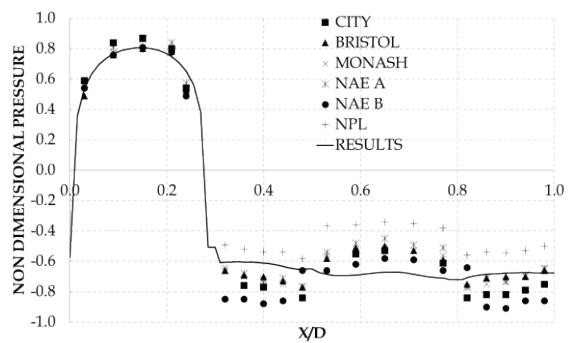**Figure 4 - (a) Pressure contours, (b) Velocity Contours and Streamlines of RANSE**
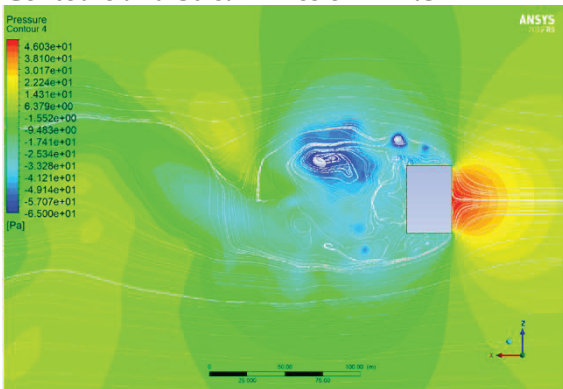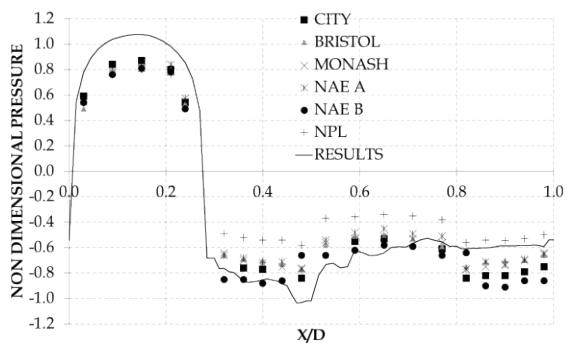

(a)
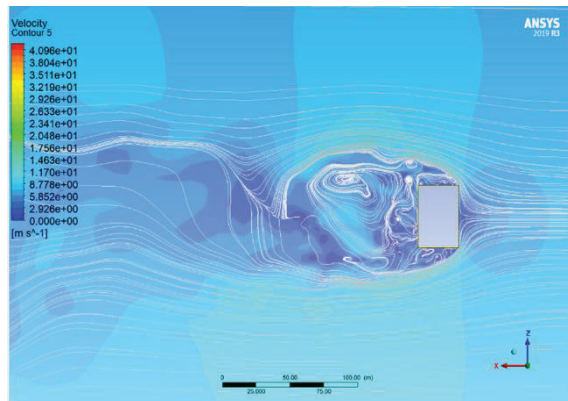
(b)

Figure 5 - (a) Pressure Contours, (b) Velocity Contours and Streamlines of LES

(a)

(b)

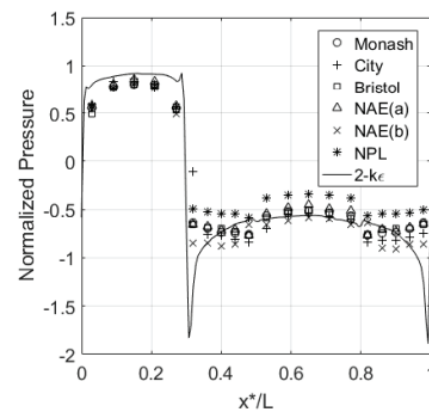**Figure 6 - Normalized Pressure at 120 m (a)RANSE (b) LES**

Figure 7 - Deviation of Results seen in Past Studies using RANSE k-ε Model [13]

The structured mesh could provide more mesh refinement at the edges of the building so that the effects of shearing and flow separation as in Figure 8, can be modelled very accurately by this mesh arrangement than it was modelled using auto mesh arrangements in previous research. Furthermore, it can be observed that, although the RANSE is now free of major deviations, results on the leeward side and the side walls are now much more conservative compared with the wind tunnel test results. The first layer was selected with 5m thickens (1:6 compared with the building width) to satisfy the buffer zone effectively for the k-ε model and a total of 30m was covered by the first two layers (1:1 compared with the building width) so more turbulent kinetic energy is solved in the immediate critical region.

Even though LES is producing a slight overprediction on the windward side, on both sides and leeward side of the building, it gives a good variation, observed similarly to the wind tunnel data. This variation is almost similar to the variation of the NAE dataset. Moreover, LES clearly gives a pressure difference at the side walls of the building which cannot be observed in the RANSE simulation.

Comparing the streamlines in RANSE, it creates two large vortices at the rear of the building and observing the animation generated by ANSYS FLUENT frame-by-frame analysis for each time step, the size of these vortices does not undergo a considerable change after the flow reaches its steady state.
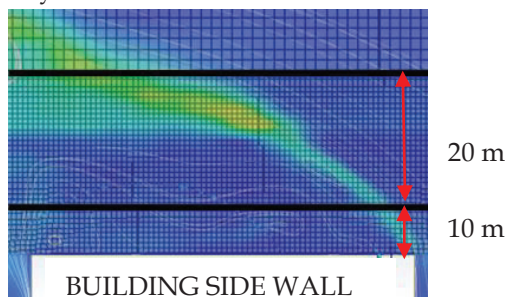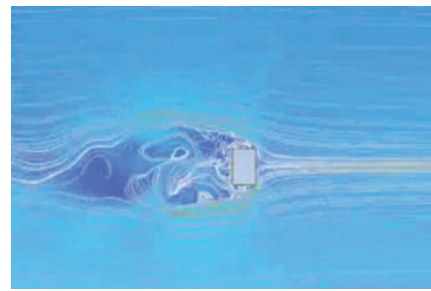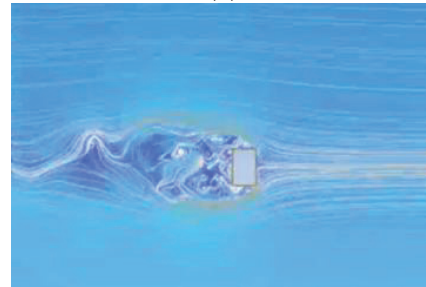

BUILDING SIDE WALL

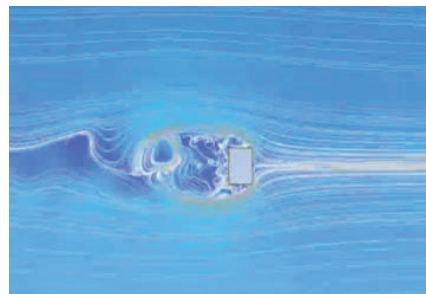**Figure 8 – Placement of Structured Mesh Layers**

(a)

(b)

(c)

(d)

Figure 9 - Key Frame Analysis (a) 60s (b) 120s (c) 150s (d) 250s after the Start of Wind Flow. In (c) and (d) the Size of Vortices Changes Alternatively

Hence, the crosswind influence is comparatively negligible in results generated by RANSE. It can be seen that by frame-by-frame analysis of LES, vortices change their sizes alternatively while they are being at the same location relative to the building as shown in Figure 9 (c) and Figure 9 (d), and small variations of the wind are recreated by LES models than it was observed in RANSE. Hence this justifies the pressure variation across the building that is observed in results of LES. Since the scope of this study has been limited to rigid structures, this pressure gradient does not affect the behaviour of the building significantly and it is thereby true for relatively heavy structures with high inertia which has insignificant deformation under wind loadings. Mainly these buildings are reinforced concrete buildings. But for slender structures which have lower lateral stiffness, this pressure gradient may induce crosswind responses that have not been effectively captured in RANSE simulation. Slender structures such as buildings with 1:10 base width to height ratio and super

slender structures with heights beyond 200m may undergo such cross-wind responses creating fluttering, vortex shedding and even tortional responses. For such purposes LES can be used as per these findings.

5.2   Computational Effort

Both RANSE and LES in this experiment had to perform 12500 calculations in total to produce the results on a computer with 16 cores with RAM of 256 GB. Considering the time taken to obtain results, it can be seen that RANSE had taken only 22 hours for the complete generation of results.

At the same time, LES had taken more than 30 hours which is more time compared with RANSE. This shows that LES consumes more time than RANSE due to the complexity of its algorithms but produces much clear and more realistic wind structural interactions compared to RANSE. Since LES requires more computational power it is very much important to decide the number of iterations per time step and the size of time step for the required duration of analysis before a full-scale simulation is carried out.

It can be suggested to create an immediate region surrounding the building, with thickness more than the buffer layer as described in Section 2.1, with elements size 1/5th of integral length scale. The time step value can be decided by considering the velocity of air at each layer. The time step size should be a value closer to the time required by an air particle to move across one integral length scale with the undisturbed velocity at the inlet. In this case it was 0.5 sec for a rectangular building with the size of CAARC standard building, the behaviour of wind can be predicted easily, and the mesh can be arranged in layers as in this experiment. But for a building with complex cross sections such as irregular sections, regular but complex shapes, a rough idea of flow behaviour should be there in order to create structured mesh layers.

5.3   Error Comparison

For the error comparison, percentage deviation can be calculated in both RANSE and LES with the NAE data set as explained in Section 4.2.
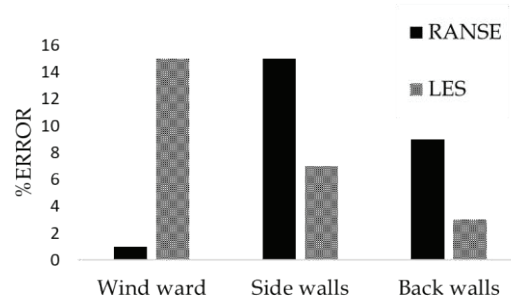

Figure 10 - Comparison of Average Error in RANSE and LES Compared with Wind Tunnel Data NAE

Observing the pressure contours it can be seen that the windward side has a large compression region due to stagnation of air. Although, the turbulent kinetic energy ratio satisfies in this region, the results are overpredicted implying it requires mesh refinement in that area. It can be suggested to define a separate layer in model formation part to improve results in the stagnating regions like this.

With this, it can be said that, even though the LES requires a lot of computational effort, it is very much effective in analysing shapes that cause more complicated flow separation and vortices with a reliable accuracy than RANSE simulation. LES has been able to model the most natural behaviour of the wind in a more reliable manner.

5.4   Response Region in Domain

Referring to the earlier research [12,13], the domain selection has been done for the conditions that were used for wind tunnel testing. The main reason for this domain selection is to reduce the blocking effect and obstacle effect as explained in Murakami [9]and Choi et al. [10]. In real situations, the wind tunnel walls have no slip boundary condition with the wind flow, creating a boundary layer to the flowing wind. Hence the wind tunnel walls should be widened to eliminate any effect from this boundary layer to the building. This makes the scaled down model wider, hence it makes the real domain even wider. Therefore, the domain used in this simulation has been over 1000m in width. But in a numerical simulation it is possible to define slip boundaries for the wind tunnel walls hence this boundary layer development in real scenario can be avoided because this provides the advantage in ANSYS fluent over the wind tunnel testing to reduce the required domain for a proper simulation. Considering the results, this effect can be clearly seen from flow lines and pressure contours that

are only limited to the proximity of the building. Hence all the volume with negligible variations can be removed from the domain. It is important to define the side walls in a way not to produce any reverse flow when particles collide with the walls after interacting with the structure.

Hence, the domain can be reduced up to 670m width (22 times the width of the building) and 400m (2 times the height of the building) height at the inlet. This will help to reduce the number of elements up to 2.6 million which is half the number of elements in this simulation while analysing the half of the volume of the previous domain. Hence same results can be obtained by spending lower computational power. This is true for modal validation of CAARC standard building hence can be applicable for buildings without screening effect of other buildings in proximity of the subject.
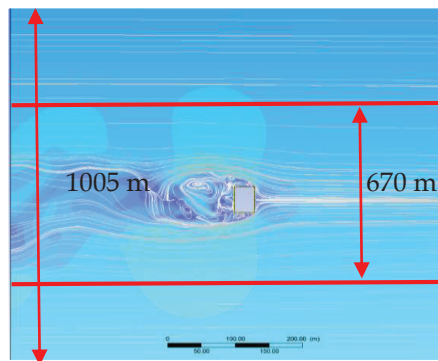
(a)
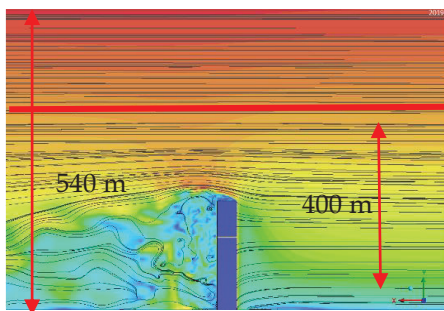
(b)

Figure 10 - Limiting the Computational Domain. In (a) Plan View (b) Sectional View

6.   Conclusions

LES can provide more realistic, natural wind and structural interactions compared to RANSE. But LES overpredicts the windward side that can be improved by introducing fine layers in compression region.

Due to the higher requirement of computational power, it can be recommended to use a structured mesh so that it gives the opportunity for the user to define regions separately and conduct mesh refinement at regions of interest separately. Results obtained in this experiment using RANSE have 1% average deviation in windward side and 9% deviation in leeward side compared with NAE. Hence, using RANSE simulation to model the wake regions and flow separations are accurate with a structured mesh, eliminating the unlikely deviations observed in past studies done using unstructured meshes. But prior knowledge should be there before arranging the mesh. Comparing the time and computational effort, RANSE is still cheaper than LES but produces more conservative results with less variations.

Furthermore, it is not useful when using the theories that are there for wind-tunnels in defining the computational domain for CFD since programs like ANSYS have the ability of modelling wind tunnel walls without developing boundary layers. Hence the width of the domain can be reduced to 20 times the width of the building and height of the domain can be reduced to twice the size of the building, reducing the volume of analysis by half, to obtain reliable results by only using 2.6 million elements for a single building problem like this.

References

1. Jayasundara, D., Koliyabandara, N. &Wijesundara, K., (2018). Wind Loads on Tall Buildings: A Comparative Study of the International Wind Codes and Numerical Simulation. *Engineer: Journal of the Institution of Engineers, Sri Lanka.* 51. 31. 10.4038/engineer. v51i3.7304.

2. Chen, Z., Xu, Y., Huang, H., &Tse, K. T. (2020). Wind Tunnel Measurement Systems for Unsteady Aerodynamic Forces on Bluff Bodies: Review and New Perspective. *Sensors*, 20 (16), 4633.

3. Mendis, P., Ngo, T., Hira, A., Samali, B. & Cheung, J. Wind Loading on Tall Buildings, *Electronic Journal of Structural Engineering, Special Issue: Loading on Structures*, 2007, 42-54.

4. Holmes, J. D., *Wind loading of structures*, 2nd Ed. Taylor & Francis, New York, 2007

5. Frison, G., Marra, A., Bartoli, G., & Scotta, R. (2019). Full-Aeroelastic Model of CAARC Building: Iterative Design Procedure and Wind Tunnel Tests: *IN-VENTO 2018. 10.1007/978-3-030-12815-9_25*.

6. Melbourne, W. H., Comparison of Measurements on the CAARC Standard Tall Building Model In Simulated Model Wind Flows, *Journal of Wind Engineering and Industrial Aerodynamics*, 73-88, 1980.

7. Launder, B. & Spalding, D., 1974. The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, 3(2), pp.269-289.

8. Huang, S., Li, Q., & Xu, S. (2007). Numerical Evaluation of Wind Effects on a Tall Steel Building by cfd. *Journal of Constructional Steel Research*, 63 (5), 612–627.

9. Murakami, S., 1998. Overview of Turbulence Models Applied in CWE–1997. *J. Wind Eng. Ind. Aerodyn*. 74–76, 1–24.

10. Choi, C.-K. and Kwon, D.-K. (1998) "Wind Tunnel Blockage Effects on Aerodynamic Behaviour of Bluff Body," *Wind and Structures*, 1(4), pp. 351–364.

11. Huang, S. H., Li, Q. S.& Wu, J. R., A General Inflow Turbulence Generator for Large Eddy Simulation, *Journal of Wind Engineering and Industrial Aerodynamics*, 98, 600–617, 2010.

12. Jayathilake, S., & Wijesundara, K. (2021). Prediction of Wind Induced Response of Tall Buildings using RANS Simulations. *Journal of the Institution of Engineers, Sri Lanka*.

13. Koliyabandara, N., Jayasundara, D. & Wijesundara, K., (2018). Evaluation of Different Turbulence Models in Determining Wind Loads on Tall Buildings. *Society of Structural Engineers, Sri Lanka, Annual Sessions* 2018.