CFD analysis for siting of wind turbines on high-rise buildings

K Veena, V Asha, C Arshad Shameem, T N Venkatesh
Aerospace Weather Research Group, CTFD Division, CSIR National Aerospace Laboratories, Bengaluru, 560017
E-mail: aarya.vjs@gmail.com, tnv@nal.res.in

Abstract. Extracting energy from wind by placing wind turbines over tall buildings is an emerging idea for "greener" cities. This paper discusses the best location for placing wind turbines based on numerical simulations. Different building geometries are considered by varying the dimension and number of buildings. Here real atmospheric boundary layer profiles are created with the help of the Weather Research and Forecasting model and also compared with the theoretical profiles from power law equation. Outputs from the simulations show the location where wind velocity is maximum and the possibility of placing wind turbines on the top as well as in the gap between the buildings. The paper also describes the significance of using realistic profiles instead of theoretical profiles.

1. Introduction
Construction of tall buildings with more than ten-twenty floors has become a part of urbanization all over the world. Nowadays attempts for optimal use of energy from renewable energy resources is gaining ground and wind energy is one of the sources which can be tapped in urban environments. An emerging idea is to place wind turbines over the high-rise buildings to extract wind energy in the form of electricity. Prominent examples include, the Bahrain World Trade Center, which has large scale wind turbines (See Figure 1).

The location where it has to be placed for maximum utilization of wind in the most efficient way is still an area of research. For that computational simulations can help to find out how the wind flows over building/buildings, its direction of flow, turbulence nature and the interference effects if more than one building is present. Among the previous works on energy harvesting using updrafts of wind over buildings using Computational Fluid Dynamics (CFD) are [1–4]. Analysis by [3] shows the increase in wind velocity at the building with different wind direction and size and shape of the building. Dependence of wind direction and size and shape of the building in exploiting the wind energy are analyzed with a series of experiments by [3, 4].

Our approach is to use realistic atmospheric boundary layer (ABL) profiles of wind to ascertain the suitable location for wind-turbines over high-rise buildings. Realistic ABL profiles are derived from simulations of the WRF [5, 6] public domain weather prediction model. Building simulations are done using the OpenFOAM CFD package.
2. Computational simulation model

2.1. Inlet profiles and boundary conditions

The wind which acts on a building can be recreated in software by introducing a velocity profile at the front of the building. It can be achieved in two ways. One is by using equations and the other with real atmospheric boundary conditions obtained from meteorological sources. Here both the types of profiles are used in the CFD simulation. Theoretical velocity profile derived from power law equation and the atmospheric boundary layer profile from WRF model. The power law equation used here is

\[ \frac{U}{U_{\text{max}}} = \left( \frac{y}{y_{\text{max}}} \right)^{\alpha} \]  

where \( U \) is the flow velocity at height \( y \) and \( y_{\text{max}} \) is the maximum flow velocity at the maximum height. Here all sets of profiles are normalized to a maximum height of 500m and maximum velocity of 4.5m/s with average velocity in the range of 3.6-3.9m/s. To have average velocities of the same order the power law profile exponent \( \alpha \) is taken as 0.2 which is similar to the case done by [2] in his works.

The terrain locations selected for creating atmospheric boundary layer profiles were the three Indian cities with lat-lon 13N-77E (Bangalore), 13.07N-80.27E (Chennai) and 9.97N-76.28E (Cochin). These realistic velocity profiles are named as Profile A, B and C respectively for Bangalore, Chennai and Cochin.

In an earlier study [7], the effect of these profiles on Coefficient of Drag for two dimensional and three dimensional buildings was compared. We found that there was a significant difference in drag coefficient between theoretical and realistic profiles and also found that the variations within the realistic profiles are comparatively small. Among them Profile B (Profile of Chennai) is considered here for all simulation. As mentioned above theoretical and realistic inlet velocity profiles are normalized to same height and average velocity and plots are as shown in Figure 2. Average velocity is integrated for further calculations so that we make sure the comparison is meaningful for all cases.
Reynolds number 10,000 was kept constant for all simulation cases and $\nu$ value is calculated in accordance with the Reynolds number. Since the fluid medium is air the density was set as 1.225 kg/m$^3$ itself. The other boundary conditions such as inlet and exit pressure values were taken as 2 bar and 0 bar respectively.

2.2. Geometry and grid
There are three different geometries included in this paper according to the three sets of analysis works. The ultimate aim of the work is to propose the best locations where wind turbines can be placed on a high-rise building. According to the number of buildings the geometries can be divided into two sets.

(i) Single building (Building A and Building B)
Two different dimensional sets of single building are tabulated in Table 1.

| Geometry   | Building dimension | Domain size     | Number of points |
|------------|--------------------|-----------------|------------------|
| Building A | 100 × 100 × 100    | 1200 × 400 × 700| 351421           |
| Building B | 20 × 100 × 20      | 1260 × 500 × 620| 3510015          |

Here the building considered is in rectangular block shape with a constant height (h) of 100m. Building A is much larger in area and looks like cubic in shape. The Building B is more or less similar to the Public Utility Building of Bangalore. Geometries were done using Gambit software and the blockMesh utility of OpenFOAM software. The domain exit was made at least at a distance of 8h from the building.

(ii) Twin buildings
Two buildings are constructed with same dimension and height kept the same as for single building. Breadth and width of both the buildings are of 20m in length and the gap between the buildings is varied. The domain size is 1260 × 500 × 620 and Figure 3 represents the model.
The incompressible Navier-Stokes equation used in the simulation is

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u}$$

(2)

where $\mathbf{u}$ is the flow velocity, $\rho$ density of the flow, $p$ pressure and $\nu$ the kinematic viscosity of the fluid. Incompressible, unsteady solver called icoFoam of OpenFoam was used in the test cases. OpenFOAM uses pressure-implicit splitting operator (PISO) algorithm to solve the Navier Stokes equations iteratively and the method of parallel computing used by OpenFOAM is known as domain decomposition, in which the geometry and associated fields are broken into pieces and allocated to separate processors for solution. The parallel version uses the public domain openMPI implementation of the standard message passing interface (MPI). The current simulations were done using the HPC systems at the CSIR Fourth Paradigm Institute and up to 64 nodes were used.

3. Computational results

3.1. Evaluation of wind velocity around the Building A and Building B

The flow gets separated at the front corner of the building and this flow stagnation creates a reversed flow at the base of the building. The vortices appear near to the corner of the building and the accumulation of these vortices results in a re-circulation zone behind the building. This region of turbulence has less intensity of wind and is not suitable for placing wind turbines. From the Figure 4 it can be inferred that just above this wake region there is a region of maximum wind speed created. The magnitude of this maximum wind velocity is much higher to that of mean wind speed. The area covering maximum wind speed depends on the cross sectional area of the building.

3.2. Comparison between theoretical wind profile with realistic profiles

As there is a drag coefficient variation due to the use of realistic boundary layer profiles [7], it is likely that it has an influences on the maximum velocity at the building roof top. Interestingly , the trend of variation is similar to that of drag coefficient change. When compared to theoretical power law profile, by the use of realistic profile an increase of 20% in maximum velocity near the building can be achieved from simulations. Both Building A and Building B are tested and
the results are as tabulated in Table 2. From the results, Building A and Building B showed an increase in velocity of 30% and 22% respectively.

The velocity contours obtained for this quantitative analysis is depicted in Figure 4.

![Figure 4. Comparison of different inlet profiles](image)

**Figure 4.** Comparison of different inlet profiles

### Table 2. Comparison of theoretical and realistic wind profiles

| Geometry   | Profile    | Maximum velocity ($U_{max}$ in m/s) |
|------------|------------|-------------------------------------|
| Building A | Theoretical| 4.36                                 |
|            | Realistic  | 5.68                                 |
| Building B | Theoretical| 4.53                                 |
|            | Realistic  | 5.56                                 |

#### 3.3. Feasible location for placing wind turbines

From the plots of velocity contours (Figure 4) it can be seen that, the area covering maximum wind velocity is at a distance of 15m from the building top. Placing wind turbines at this height will lead to maximum power generation. Even at a lesser height (upto 10m) there is sufficient wind for power generation. The other regions to place the turbines are at the sideways of the building below to the rooftop, where velocity is still high in magnitude.

When more than one building is present, the interference effect also has to be considered. Figure 5 shows the simulation over twin buildings and a schematic representation in which $s$ is the distance of separation between the buildings and $U_{max}$ is the maximum wind speed in between the buildings. The Table 3 indicates the influence of separation distance on the wind speed.

**Table 3.** Influence of separation distance between the buildings on wind velocity

| Twin building cases | $s/h$ | $U_{max}/U_\infty$ |
|---------------------|-------|---------------------|
| Case I              | 50/100| 0.973               |
| Case II             | 20/100| 0.351               |
4. Conclusion
We have carried out a study on the optimal siting of wind turbines over high rise buildings using computational simulations using OpenFOAM. A feature of this study is the use of realistic wind profiles derived from a numerical weather prediction model. While it is true that results depend on the exact geometry of the building, the general conclusions we can draw are as follows. We find that for a single tall building, there is a significant fraction over which winds are sufficient to be used for wind power generation.

For twin buildings, wind turbines can be placed not only at the roof top but also in between the buildings. A velocity of 2m/s can be obtained for a separation distance of fraction 0.2 and 2-2.8m/s for 0.5.

5. Acknowledgments
This work was carried out as part of the ARIEES project, under the twelfth Five year plan of the Government of India. The support is gratefully acknowledged. The computational facilities at CSIR-4PI were used for the simulations.

References
[1] Park J, Jung H J, 2 S W L and Park J 2015 *energies* 8 11846
[2] Mohamed A, White C and Watkins S 2012 *15th Australasian Wind Engineering Society Workshop*
[3] Cai Y, Zhao F and Liu S 2015 *International Conference on Information Sciences, Machinery, Materials and Energy* 591
[4] Toja-Silva F, Peralta C, Lopez-Garcia O, Navarro J and Cruz I 2015 *computation* 3 299
[5] Michalakes J, Dudhia J, Gill D, Henderson T, Klemp J B, Skamarock W C and Wang W 2005 *Proceedings of the Eleventh ECMWF Workshop on the Use of High Performance Computing in Meteorology*
[6] Klemp J B, Skamarock W C and Dudhia J 2007 *Monthly Weather Review* 135 2897
[7] Veena K, Asha V, Shameem C A and Venkatesh T N 2015 *17th CFD Annual Symposium CFD Division - Aeronautical Society of India* 11-12 August, Bangalore 164