COMPUTATIONAL MODELING OF TERRAIN AND BUILDING AERODYNAMICS FOR ENHANCING ARCHITECTURAL DESIGNS

V. Dwivedi\textsuperscript{1} and M. Damodaran\textsuperscript{2}

\textsuperscript{1}Graduate Student, Mechanical Engineering, IIT Gandhinagar, GJ, India, Email - vinay.dwivedi@iitgn.ac.in
\textsuperscript{2}Professor, Mechanical Engineering, IIT Gandhinagar, GJ, India, Email - murali@iitgn.ac.in

ABSTRACT

An attempt at using computational fluid dynamics (CFD) to assess and enhance the architectural design for the proposed new IIT Gandhinagar Campus buildings with flow models integrated with climatic data and wind patterns corresponding to average annual weather conditions in the neighborhood to assess the resulting terrain and building aerodynamic flow patterns and thermal fields is addressed in this work. Computed temperature fields, pressure fields, wakes, vorticity structures for a specific free stream field acquired via multiple meteorological resources applied as Atmospheric Boundary Layer are shown along with the assessment of variation of humidity and its effects on the flow fields and temperature field to show the implications for enhancing architectural designs and serving as a basis for the assessment of natural cooling systems in indoor ventilation.

Keywords: - CFD, Terrain and Building Aerodynamics, Architectural Designs, Humidity Transport

1. Introduction

Aerodynamic and thermal profiling of terrains and buildings using modern high fidelity computational engineering tools over the past few decades have added value to architectural and structural design of buildings by providing an effective tool to assess, predict and implement ideas in a faster and accurate way. It has allowed architects and engineers working in the field of design to experiment with innovative building designs while keeping a check on the impact of the design on the environment. Initial attempts pertaining to high fidelity computational analysis of terrain aerodynamic and thermal flowfields over the new IIT Gandhinagar campus (considered as a case study) have been performed and reported in this work. The initial conditions and parameters chosen have been taken from various meteorological resources providing summarized data over a cycle of time. Taking cue from the works of Golubev (2010) and Zheng et.al (2012) and the initial architectural drawings provided by the architects designing the new IIT Gandhinagar campus which comprises of a series of low rise buildings to be built along the banks of the Sabarmati River at Palaj, Gandhinagar, Gujarat, India and shown in Figure 1, a computational study was carried out using an incompressible flow model embedded in the commercial flow solver \textit{STAR-CCM+}(2012) to predict the airflow patterns and thermal fields in the vicinity of the various buildings to assess pedestrian comfort, to identify critical regions which might require structural or shape modification to mitigate these regions.
First the aerodynamic flow field past a collection of bluff bodies of arbitrary shapes (campus buildings) over the region have been modeled and presented under dry conditions and with uniform flow with respect to altitude. Subsequently, the Atmospheric Boundary Layer (ABL) inflow condition has been implemented and modeled to account for the effects of flow variation with respect to altitude from the ground level. Finally, the humidity conditions relative to the chosen scenarios have been applied and modeled so as to study the variation in temperature and observe accumulations. Computed results have been presented and compared accordingly with parameters used for simulating the conditions being chosen as the annual average from different weather statistical resources applicable over the campus site.

2. Computational Modelling

The model used for this study is the set of conservation laws for mass, momentum, and energy describing unsteady incompressible flow in finite volume formulation as follows:

\[
\frac{\partial}{\partial t} \iiint_{\Omega} w dV + \iiint_{\partial \Omega} F_c dS = \iiint_{\Omega} F_{visc} dS + \iiint_{\Omega} B dV \quad 1(a)
\]

\[
\rho C_f \left( \iiint_{\Omega} \frac{\partial T}{\partial t} dV + \iiint_{\partial \Omega} \overline{T} dS \right) = \iiint_{\Omega} \left( k \nabla (T) - (c_{\omega} T + L_{\omega}) \overline{g} \right) \quad 1(b)
\]

\[
\rho \left( \iiint_{\Omega} \omega dV + \iiint_{\partial \Omega} \overline{\omega} dS \right) = \iiint_{\Omega} \left( \rho (D \nabla (\omega)) \right) dS \quad 1(c)
\]

where \( w = [0 \ U_x \ U_y \ U_z \ E_{\text{total}}] \) represent the flow variables namely the total velocity components \( U_x, U_y, U_z \) and the total energy \( E_{\text{total}} \), \( F_c \) represents the convective terms, \( F_{visc} \) the viscous terms, \( B \) represents the body forces, \( \rho \) is the air density, \( C_f \) the air thermal capacity, \( k \) the thermal conductivity, \( c_{\omega} \) the thermal capacity of the water vapor, \( L_{\omega} \) the latent heat of vaporization, \( g \) the water vapor diffusion flux, \( D \) the diffusion coefficient of water vapor in air, \( T \) is the temperature and \( \omega \) is the mass fraction of water vapor \( \omega \) in the air. Eqn 1(a)-(b) is the generalized form of Navier Stokes Equations embodying continuity, momentum and energy transport. The energy equation (Eqn. 1(b)) is coupled with the species transport
equation for relative humidity (RH) (Eqn. 1(c)). Humidity is defined in terms of the mole fraction governing variations in pressure and temperature fields. The convective fluxes in the momentum equations are discretized using second order upwind scheme as outlined in Steeman et al. (2009). Airflow turbulence modeling is based on the Spalart-Allmaras turbulence model by solving a single transport equation for estimating the turbulence eddy viscosity coefficient $\mu_T$ which is used in the transport equations.

Finite volume discretization of these equations are implemented on a computational mesh shown from various viewpoints in Figure 2 which show the structure of the polyhedral meshes which are used to tessellate the domain in the passages between the buildings, gaps and also up to a certain altitude from the ground level. Prismatic layers of structured meshes are used on the wall surfaces for capturing boundary layers and flow separation. Visualization planes to plot contours of flow variables are also defined in this figure. More than 4 million cells have been used in this study and about 5 prism layers are used in the vicinity of the ground and building walls and roofs such that the $y^+$ values for ground and building surfaces are different. While the mesh structures have been kept rather fine near the ground in close proximity of the buildings, the mesh size is gradually increased or coarsened with increasing altitude where flow gradients are expected to decrease.

Figure 2: Meshing up the region of interest

For the initial study, the buildings are treated as boxes according the floor shapes defined in the architectural drawings and the terrain is assumed flat for establishing the model. Future investigations will incorporate specific geometrical details of the buildings and the terrain elevation into the computational mesh. The general areas of concern expected in such studies are the existence of multiple flow separation and reattachment points considering the
presence of a number of bluff bodies of myriad shapes in close proximity in the the area of interest, low temperature variations which may not be very significant, large recirculation regions and the presence of horse shoe vortices. Ambient wall temperatures on building surfaces have implications for the thermal comfort inside the buildings. In order to simulate with parameters closer to the real life atmospheric conditions, the Atmospheric Boundary layer (ABL), the formulation, implementation and treatment of which is defined in Crasto (2007) has been used in this study. The variation of wind speed from ground level upwards is defined as:

\[ \frac{U}{U_{\text{ref}}} = \left( \frac{Z}{Z_{\text{ref}}} \right)^{\alpha} \]

where \( \alpha = \ln \left( \frac{Z_{\text{ref}}}{Z_{o}} \right) \), \( U \) is the velocity at altitude \( Z \), \( U_{\text{ref}} \) is the reference velocity at reference altitude \( Z_{\text{ref}} \), and \( Z_{o} \) is the roughness length. For this study \( U_{\text{ref}}, Z_{\text{ref}} \), and \( Z_{o} \) are assumed to be 10 m/s, 25 m and 0.02 respectively. Corresponding to this the reference pressure and temperature are assumed to be 101325 Pa and 304 K respectively, a turbulent intensity of 3\% and a relative humidity of 60\% assumed as initial conditions, At this stage a real time model may be overarching as well as unnecessary and hence only extreme weather patterns in a typical year such as summer and winter months are considered for the flow simulation. The ABL profile shown in Figure 3 and used in the present study is based on the averaging of wind data provided by various internet sources [Numerical Weather Atlas of India (2005) and Wind Statistics at Ahmedabad (2013)] providing real time and summarized details of wind and temperature conditions in a desired geographical region of India. The ABL variation of wind speed from ground level to altitude is defined at the inflow boundaries. At the outflow the gauge pressure is set to null. Further work on the incorporation of meteorological data as outlined in Pielke and Nicholls (1997) will be explored for realistic model of airflow in the region. Relative humidity (RH) expressed in terms of mass fraction of water vapor and air and being the ratio of vapor pressure \( P_{v} \) defined as:

\[ P_{v} \approx 6.107 \times 10^{\alpha T} \]  

and saturated pressure \( P_{s} \) defined as

\[ P_{s} \approx 6.107 \times 10^{\alpha T} \]

is used to calculate the mass fraction of water vapor, \( X_{H_{2}O} \) using the Magnus formulation outlined in Steeman et al. (2009) as

\[ X_{H_{2}O} = \frac{M_{H_{2}O}}{M_{M}} \left( \frac{P_{v}}{P - RH \cdot P_{s}} \right) \]  

where \( a = 7.5 \) and \( b = 235.7 \) for \( T > 0 \), \( X_{H_{2}O} \) is the mass fraction of water vapor, \( M_{H_{2}O} \) is the molar weight of water vapor, \( M_{M} \) is the molar weight of the mixture.

3. Results and Discussions

First a converged steady flow field corresponding to uniform wind speed in a prescribed direction based on average meteorological conditions over the terrain and bluff bodies representing the buildings is computed. Using this computed flow field as initial conditions, the ABL conditions are imposed at the inflow boundaries and the flow field computed. The contours of various flow field variables and flow parameters of interest corresponding to the steady flow field that has been computed using the ABL velocity profile at the inflow boundaries are shown in Figures 4(a)-(c).
The contours of the computed velocity flow field at an altitude of 4m from the ground are shown in Figure 4(a) and that at an altitude of 5m from the ground in Figure 4(c). The stagnation points are clearly visible from Figure 4(a)-(c) and so are bow shaped regions in front of the structures representing low speed zones with rapid circulations. This has been explored further using vector plots as shown in Figure 5.
It can be clearly seen from Figures 4 and 5 that where the flow impinges on the surfaces of the buildings, regions of circulation build up and are propagated downstream with the flow direction and which results in wakes emanating from various building structures to produce re-circulating regions. These wakes are helical in nature and are dominant near the ground due to the atmospheric boundary layers which induce low velocities near ground which gradually goes to a complete standstill on the ground. These wake regions and flow patterns are monitored using flow visualization tools based on velocity contours in Figure 6(a) which show the velocity vectors showing 3 dimensional circulations between structures and Fig 6(b) which represents the velocity vectors in line integral convolution form giving vertical regions and these figures together gives a view of the flow patterns being generated by the propagated wake interactions with each other. Figure 6(c) shows a closer view of the multiple circulations generated within structures.

Fig 6: (a) Velocity Contours (vector form) (b) Velocity Contours showing circulation within structures (c) Velocity Contours showing multiple circulation regions (Visualisation based on the Line Integral Convolution form)

Along with computed flow circulations on horizontal planes, vertical circulation patterns were also observed. These patterns are inferred emerging due to the interactions of flow with structures accompanied by the atmospheric boundary layer inducing lower flow
velocities in the proximity of ground which leads to helical flow structures. Due to the ABL, flow on upper layers is forced downwards, adding to the circulations generated by the disturbed flow. These variances have been shown in Figure 7(a) in terms of velocity line integral convolution plots on the vertical sectional plane depicted in the panned image. Figure 7(b) shows similar variances but in terms of the computed vorticity contours on the considered sectional plane. A clear variation of flow in accordance with ABL can be seen in the velocity plots, with minimal velocities on ground and maximum at farfield. The rotation of flow aft of the structures is clearly visible from both the cases.

Corresponding to the computed velocity patterns of airflow, the computed local pressure fields on ground, walls and at an altitude of 4m from ground have been shown in Figures 8(a) and 8(b) respectively. From these figures the pressure variations at stagnation regions where
Airflow interacts with structures have been observed to be of higher than elsewhere and these pressure variations are also affected by the continuous and random fluid-structure interactions in the vicinity of buildings leading to gusty regions in the passages between buildings. These inferences can also be derived from Figure 6(c) which shows the zoomed in image of the flow patterns at an altitude of 4 m. A similar trend can be seen in the computed temperature field shown in Figures 8(c) and 8(d) where temperature is being analysed.

![Pressure contours](a)

![Pressure contours](b)

![Temperature contours](c)

![Temperature contours](d)

![Relative Humidity (RH) Contours](e)

![Relative Humidity (RH) Contours](f)

Fig 8: Pressure contours at (a)ground (b) on a plane 4m from ground; Temperature contours at (c) ground (d) on a plane 4m from ground; Relative Humidity (RH) Contours at (e)ground (f) on a plane 4m from ground.
It appears that even for low speed flow conditions the temperature variations are markedly visible and are mainly induced in the areas of high pressure variation i.e. at regions where the flow interactions are high. These variations, though not severe, provide an insight that these can be significant enough in the presence of gusts in the region of interest. Inclusion of the transport of relative humidity (RH) also indicates variations in the concentration of vapor content at different regions in the computational domain as shown in the contours of relative humidity of air shown in Figures 8(d) and 8(e) respectively. From mass fraction in Eqn. 2(c) depends on the saturation and vapor pressures which are calculated in terms of local and dew point temperatures respectively. Mass fraction thus calculated has been used to track relative humidity in the region. As shown in Figure 8(e) and 8(f), high concentrations of RH were observed in stagnation regions and regions with high fluid interactions. This expresses the relative humidity variation as a ratio of the varying computed pressures fields which are tracked via the change in relative humidity showing the accumulation regions where stagnation conditions prevail and in regions with high fluid interaction between building structures and airflow. As the variations are primarily dependent on the computed temperature field, molar concentration variations can be observed at regions showing high temperature and pressure concentrations.

4. Conclusions

The aspects of airflow in an urban environment accompanied by atmospheric effects, as discussed in this work, show the areas of key importance for enhancement of architectural designs. The inclusion of humidity and temperature renders a more physically realistic image for analysis and use. The airflow patterns clearly show the regions of high circulation. This information can be fruitful for use in erection of structures sensitive to airflow, such as efficient installation of low cost wind energy systems. It may also aid in designing open air units as sports arenas or recreational spots. It shall also prove beneficial in pre-assessing circulations in areas where it is desirable to have lower wind activity, so that proper blockades whether natural or artificial may be put in place. While these inferences are of great importance, future aspects of the work may provide an even deeper insight. Further analysis of multiple scenarios representing weather extremities can be done so as to define the upper limits of atmospheric effects over the area of interest. This shall ensure a controlled and effective design. Taking into account the presence of time dependent and directional variations of wind characteristics that shall prevail over the area where the structures are to be installed, may also prove beneficial. Indoor thermal comfort can be assessed as well, considering the calculations indicating accumulations of humidity.

Acknowledgements

The efforts of P. Nyati and N. Nuwal, 3rd year BTech (ME) students of IIT Gandhinagar in this computational work is acknowledged. The winter internship support granted to them and the 1st author of this paper for 3 weeks in December 2012 at CD-Adapco, Bangalore enabled the initial attempt at this problem during that period.

References


http://www.cwet.tn.nic.in/html/departments_numerical_wind_atlas.html
Centre for Wind Energy Technology, Chennai, India


