Numerical simulation of wind flow and heat transfer over a cluster of buildings

K Veena$^{1,2,*}$, K M Parammasivam$^2$ and T N Venkatesh$^1$

$^1$ CTFD Division, CSIR-National Aerospace Laboratories, Bengaluru-560017,
$^2$ Madras Institute Of Technology, Chennai-600044.
*e-mail:aarya.vjs@gmail.com

Abstract

There is sudden and abnormal variation in temperature within the cities in a localised manner due to the phenomenon called Urban Heat Islands. Estimation of increase in temperature and its dependence on environmental and anthropogenic factors help us to find the remedies to reduce the intensity of these hotspots. In this paper the influence of heat on wind flow structures are analysed with the help of Computational Fluid Dynamics and a part of real city is modelled for studying the cooling effect within the cluster of buildings using heat coefficients derived from governing equations.

1 Introduction

Increasing temperature within the city is a serious issue facing by the people in the metropolitan cities. Areas wherein there is an accumulation of such heat are termed as Urban Heat Islands [1, 2, 3]. It is the phenomenon of having consistently higher temperature in the urban area when compared to the surrounding rural area and is measured in terms of Urban Heat Island Intensity factor, which is the difference in temperature in the central city to that of its surrounding rural location. This localized increase in temperature is due to the increased anthropogenic heat (especially from vehicles, buildings etc), increased longwave radiation from the earth, reduced sky view factor, rise in number of concrete building structures placed adjacently. There are two categories of Heat Islands: they are Surface Heat Islands and Atmospheric Heat Islands. Surface Heat Islands means the heat islands formed on the ground or land surface and the near surface of it. It can be estimated with the extracted Land Surface Temperature from Remote Sensing/satellite Data. The Atmospheric Heat Islands formed due to the increase in Air Temperature can be measured using instruments like Temperature Probes.

Another way of identifying the hotspots are by using simulations. The ability in predicting the future changes in hotspots according to different scenarios and experimenting the impact of various configurations of buildings for a newer city makes Numerical Simulations helpful and challenging in the Urban Microclimatic Study. This
paper evaluates the stages of achieving hotspot identification in a small part of city and UHII measurement using microscale CFD models and mesoscale climate models. Previous studies are related to wind flow over single and multiple buildings [4], the complex structures formed behind the buildings and the influence of it on the heat transfer [5]. In the paper [5] instead of solving complete temperature equation the presence of heating or cooling within the cluster of buildings is predicted with the help of heat transfer coefficients. It was the initial step in the complete thermal modelling of flows for addressing UHI problem. In this paper thermal modelling part has been included for analyzing the two way influence of wind flow and heat transfer to each other for a model of World Trade Center, Bangalore (as shown in Figure 1) and its adjacent buildings.

2 Assumptions and Methodology

In order to analyze the influence of temperature on wind flow patterns and vice versa, initially a single idealized building case is simulated by considering only the wind flow and then with thermal modelling included. A hybrid RANS/LES model called Detached Eddy Simulation (DES [6]) model is used for simulating the wind flow alone over the buildings. The model acts as a RANS model in attached boundary region and in the separated region it performs LES operations. In the case of flows accompanied by heat transfer, since the density variation is not large the solver which includes the Boussinesq approximation is preferred. So the solver available in OpenFOAM called buoyantBoussinesqSimpleFoam is used which is a steady state solver for incompressible, turbulent flows. An unsteady solver called buoyantBoussinesqPimpleFoam is also used for further studies. The mass and momentum conservation equations employed in the buoyantBoussinesqSimpleFoam solver are:

\[ \nabla \cdot \mathbf{u} = 0 \] (1)

\[ \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \mathbf{u}) = -\nabla p_{rh} - (g.r)\nabla (\frac{\rho}{\rho_0}) + \nabla \cdot (2 \nu_{eff} D(u)) \] (2)

where \( D(u) \) is the rate of strain tensor and defined as

\[ D(u) = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \] (3)

Boussinesq approximation is valid when the density induced temperature changes are small. The density in the gravitational term is expressed as

\[ \rho \approx \rho_0[1 - \beta(T - T_0)] \] (4)

\( \beta \) is the volumetric thermal expansion coefficient and its definition is

\[ \beta \equiv -\frac{1}{\rho} \frac{\partial \rho}{\partial T} \approx -\frac{1}{\rho_0} \frac{\rho - \rho_0}{T - T_0} \] (5)

The building models are as shown in Figure 2. The single building has a height of 100m and for the cluster of building, a simplified model of the World Trade Center, Bangalore and some of its surrounding buildings are used. The wind flow profile and temperature profile is extracted from Weather Research and Forecasting Model (WRF) for a single day in April 2019. It is the mean value profile for the whole day up to a height of 600m from the ground. The modelling part is done in Catia software, then meshing and analysis in OpenFOAM software for all the test cases. The maximum number of grid points simulated is 35 million for the test cases.

For the thermal modeling part, in the initial runs the specific temperature was set for the ground, concrete building and the air. But the surface energy balance equation shown below has to be considered for a detailed realistic study. As per [7],

\[ Q^* + Q_F = Q_H + Q_E + \Delta Q_S + \Delta Q_A \] (6)

where

\( Q^* \) = net all wave radiation flux  
\( Q_F \) = anthropogenic heat flux
Figure 2: Mesh view of a single building case and the cluster of buildings

\( Q_H \) = turbulent sensible heat flux
\( Q_R \) = turbulent latent heat flux
\( \Delta Q_S \) = net release or uptake of energy in the urban ground canopy air volume
\( \Delta Q_A \) = net horizontal advective heat flux

In [8], the net surface energy budget was balanced as on the left hand side sum total of Incoming solar radiation and Longwave downwelling radiation which is equivalent to the sum of Longwave upwelling radiation, Sensible Heat Flux, Latent Heat Flux and Ground Heat Flux on the right hand side. In numerical simulations it is difficult to add all the parameters that influence the increase in atmospheric heat, so the effect of limited parameters only can be considered in the initial works. Upwelling longwave radiation \((Q_{Lu})\) can be calculated from the equation

\[
Q_{Lu} = \epsilon_g \sigma T_g^4
\]  

(7)

where \( \epsilon_g \) is the emissivity of the ground and its values ranges from 0.9 to 0.99([8]). \( \sigma \) is the Stefan Boltzmann constant and \( T_g \) is the ground temperature. In paper [9], it is given that emissivity factor of concrete is 0.85. According to the emissivity factor of ground and building, the relation between the temperature of the building with respect to ground is calculated and tabulated as shown in the Table 1 below.

<table>
<thead>
<tr>
<th>Emissivity of ground ( \epsilon_g )</th>
<th>Emissivity of Concrete ( \epsilon_c )</th>
<th>% increase in temperature for concrete with respect to ground</th>
<th>Ground Temperature (K)</th>
<th>Building Temperature (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>0.85</td>
<td>1.442</td>
<td>300</td>
<td>304.326</td>
</tr>
<tr>
<td>0.91</td>
<td>0.85</td>
<td>1.719</td>
<td>300</td>
<td>305.157</td>
</tr>
<tr>
<td>0.93</td>
<td>0.85</td>
<td>2.275</td>
<td>300</td>
<td>306.825</td>
</tr>
<tr>
<td>0.99</td>
<td>0.85</td>
<td>3.890</td>
<td>300</td>
<td>311.67</td>
</tr>
<tr>
<td>0.93</td>
<td>0.85</td>
<td>2.275</td>
<td>302</td>
<td>308.871</td>
</tr>
</tbody>
</table>

3 Model Validation

The validation study is conducted for wind flow alone over the building as well as for the wind flow accompanied with thermal effects.
3.1 Validation Case 1: For wind flow alone

For validating the DES model for wind flow analysis, two buildings of height 18m separated by a passage of 4m which is having experimental as well as numerical results [10] is considered here. DDES model is used for present study and in [10] the RSM turbulence model of Fluent is used to compare with Wiren wind tunnel test results [11]. The Figure 3 below shows the test case model as well as the analysis results.

(a) Schematic representation of the two building model
(b) Velocity plot over xz-plane

Figure 3: Validation Case 1: Wind flow over two buildings with a passage

In Figure 4, the velocity of the wind to the free stream velocity of the wind $U/U_0$ is plotted at different locations in between the gap of the buildings. The OpenFOAM results shows a similar result pattern in comparison with wind tunnel and Fluent simulation results. Towards the downstream portion the OpenFOAM results are slightly overpredicting.

3.2 Validation Case 2: For wind flow with thermal effects

A 2D street canyon model which is having wind tunnel results are used for the validation of the current simulation model for thermal effects. The model created is of six street canyons of Aspect Ratio 1 and the height of the domain is about three times in height with respect to the street canyon height from the top of the building as shown in Figure 5(a). The boundary conditions are set according to the studies of [12], [13] and [14]. The Bulk Richardson Number, $R_b = -0.27$ and temperature difference between street canyon $T_s$ and ambient air $T_a$, $\Delta T_{sa} = 2K$ is considered for the validation [13, 14]. Boussinesq approximation is used in the model and the turbulence model used here is the $k - \epsilon$ one. Figure 5 (b) shows the mean temperature plotted over the 2D case model and the results are compared with the wind
tunnel results of [12] and the numerical simulation results of [13] and [14]. In reference to the method of [14],
the normalized temperature \(\frac{T - T_a}{T_s - T_a}\) is plotted over the vertical sections from the floor to a height
of \(y/H = 2\) as shown in Figure 6. The 'OpenFOAM' indication in the graph shows the current validation case
outputs. From the results seen in the plot the numerical model used in the study is considerable for further
wind and heat transfer simulations.

Figure 6: Validation Case II: Vertical profile of Normalized Temperature from the present results in Open-
FOAM and from other published studies of Uehara et al., 2000 [12] (wind tunnel results), Kim and Baik 2001
[13](standard \(k - \epsilon\) model) and Xie et al., 2006 [14](realizable \(k - \epsilon\) model)

4 Results

The results obtained can be classified in to two sets. One set for evaluation of changes in wind flow structures in
the presence of temperature for single building and another set the same for cluster of buildings. The sections
below explains the effect of temperature on the wind flow features for single and cluster of buildings.
4.1 Analysis of results for single building cases

The single high-rise building of height 100 m is analyzed with only wind flow and wind flow along with thermal effect. Flow equations without the inclusion of temperature part is indicated as Case A and the wind flow along with temperature equations resolved are indicated as Case B hereafter. The temperature of the ground and air was set as 300K and the building temperature was varied in steps for Case B. Thus there are 4 different runs made for the Case B category where the building temperature was kept as 0.5, 1.0, 2.0 and 4.0 K higher than the ground temperature and another case with zero change in temperature.

![Case A](image1.png)

![Case B with \( \Delta T = 0 \)](image2.png)

![Case B with \( \Delta T = 0.5 \)](image3.png)

![Case B with \( \Delta T = 1.0 \)](image4.png)

![Case B with \( \Delta T = 2.0 \)](image5.png)

![Case B with \( \Delta T = 4.0 \)](image6.png)

Figure 7: Velocity contours over z-normal plane for Case A and Case B runs

The Figure 7 below shows the velocity contours over z-normal plane for Case A and Case B runs. The temperature has impact on the wind flow structures formed behind the buildings. From Figure 7 and 8, it is clear that as temperature of the building increases there is difference in the re-circulation zone created behind the buildings and the flow re-attachment to the ground. In front of the buildings contours are the same for all sets of cases, the slight difference is only due to the unsteadiness. These results are with a steady state solver and
building temperature was set theoretically to identify how the variation influences the wind flow pattern as well as heating around the building. As $\Delta T$ increases the flow re-attachment length gets reduced and the values are as listed in the Table 2 given below.

<table>
<thead>
<tr>
<th>Sl No.</th>
<th>Cases</th>
<th>Re-attachment length from building foot ($l_R$)</th>
<th>Re-attachment length in terms of building height ($l_R/h$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$\Delta T = 0$</td>
<td>180m</td>
<td>1.8</td>
</tr>
<tr>
<td>2</td>
<td>$\Delta T = 0.5$</td>
<td>130m</td>
<td>1.3</td>
</tr>
<tr>
<td>3</td>
<td>$\Delta T = 1.0$</td>
<td>100m</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>$\Delta T = 2.0$</td>
<td>80m</td>
<td>0.8</td>
</tr>
</tbody>
</table>

In Figure 8, variation in temperature over the plane horizontally cut over the building is shown. Due to buoyancy the spreading of temperature varies according to the increase of $\Delta T$. The spreading of high temperature near to the building reduces as $\Delta T$ increases.

Another case was simulated by calculating the building temperature according to the emissivity factor of concrete and ground. From the calculation, for an emissivity factor of ground and building as 0.93 and 0.85 respectively, the temperature of the building was obtained as 308.871K for 302K ground temperature. The Table 1 shows the emissivity calculation for different ground temperature. The velocity and temperature plots are as given in Figure 9. Due to the high difference in temperature between ground and building the reattachment length reduced to the minimum value.
4.2 Analysis of results for cluster of buildings

The preliminary results showing wind flow over the cluster of buildings are as given in Figure 10. Here the building structures are complex and the wind flow patterns too. The case with only wind flow over the buildings and streamlines plots are given here. As explained in the before section similar conditions where tried for cluster of buildings as well. The Figure 10 shows the different views of streamlines plotted over the cluster of buildings. Here, the unsteady solver in DES is used without solving temperature equations. The wind direction is from North East and the directions of buildings are as marked in the Figure 1 (b).
From these plots, assessment of the suitable locations for siting roof-top wind-turbines can be made. From K-theory, it was understood that one of the heat transfer coefficients which is prominent in the temperature equation is the eddy diffusivity of heat $K_H$ ([5]). The presence and values of heat transfer coefficient gives insight to the heating/cooling near the buildings. The isosurface of $K_H$ coloured by velocity magnitude is as shown in Figure 11.

![Figure 11: (a) and (b) shows the isosurface plots of $K_H$ over cluster of buildings coloured by velocity magnitude, (c) shows the $K_H$ over a plane at a height of 3m from the ground and (d) shows the $K_H$ value plotted over xz-plane](image)

The $K_H$ value near the ground at a height of 3m from the ground is shown over a plane in Figure 11 (d). This indicates the presence of cooling effect in between the buildings near to the ground.

5 Conclusion

CFD simulations of flow past a cluster of buildings has been carried out using the DES model of OpenFOAM, with realistic wind profiles imposed from an atmospheric model. The Boussinesq approximation has been used to model the temperature and density effects. A complex cluster of buildings has been modelled and the initial results of the simulations have helped in understanding how the wind flow structures get modified due to the influence of temperature. This work has many potential applications such as improving building layout for better ventilation, assessing wind loads and also for finding suitable locations for wind-turbines.
Acknowledgments

Availability of OpenFOAM software as an open source greatly helped for simulating the cases. All the computations were carried out using CSIR’s HPC facility at CSIR-4PI. First author is supported by CSIR through Direct SRF Scheme. The support from Mr. Arshad Shameem and other members of CTFD Division, CSIR NAL is greatly acknowledged.

References