Numerical simulation of convective heat transfer at the surfaces of a cube

Citation for published version (APA):

Document status and date:
Published: 01/01/2013

Document Version:
Accepted manuscript including changes made at the peer-review stage

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication
Numerical simulation of convective heat transfer at the surfaces of a cube

H. Montazeri, B. Blocken, J.L.M. Hensen

Building Physics and Services, Eindhoven University of Technology, P.O. box 513, 5600 MB Eindhoven, the Netherlands

h.montazeri@tue.nl

Abstract. Knowledge of the convective heat transfer coefficient (CHTC) on building walls is important for research on building energy and building component durability. In building aerodynamics, steady RANS is frequently used to model air flow, rather than unsteady RANS (URANS) or Large-Eddy Simulations (LES). To gain insight into the performance of LES compared to steady RANS, this paper presents LES and RANS CFD simulations of the temperature distributions at the surfaces of a reduced-scale cubic model measured in turbulent channel flow. The evaluation is based on a grid-sensitivity analysis. The results show that LES can accurately predict the surface temperature distributions of the cube walls. Steady RANS, however, indicates a satisfactory agreement with the experiments only for the windward surface.

1. Introduction

Knowledge of the convective heat transfer coefficient (CHTC) at surfaces of bluff bodies immersed in turbulent flows is essential for many engineering applications. For example research on building energy and building component durability is dependent on detailed information of the local and mean CHTC. Using inappropriate models to calculate CHTC can lead to considerable uncertainties in the results of the Building Energy Simulation (BES) programs [1].

Convective heat transfer studies on bluff bodies are mainly performed by full-scale measurements (e.g. [2-3]), wind-tunnel experiments (e.g. [4-5]) and by Computational Fluid Dynamics (CFD) (e.g. [6-7]). Experimental results can be used for CFD validation studies. Some previous studies on high-resolution CFD simulations of CHTC used the steady RANS approach (e.g. [7]). However, steady RANS is incapable of capturing the inherently transient behaviour of separation, reattachment and recirculation downstream of the windward facade and of von Karman vortex shedding in the wake [8]. LES on the other hand is known to provide accurate descriptions of the mean and instantaneous flow field around bluff bodies (e.g. [8-9]). Therefore more accurate CHTC simulations should be pursued using transient simulations with LES.

To gain insight into the performance of LES compared to steady RANS, this paper presents LES and RANS CFD simulations of the temperature distributions at the surfaces of a reduced-scale cubic model immersed in turbulent channel flow. The evaluation is based on a grid-sensitivity analysis and on validation with measurements performed by Meinders et al [5].
2. Description of wind tunnel experiments

In the present paper, the experiments by Meinders et al [5] are used for validation purposes. In these experiments the distribution of local convective heat transfer at the surfaces of a cube placed in turbulent channel flow was investigated. The channel had a height of 0.05 m and a width of 0.6 m (Fig. 1a). Two heat exchangers were used to maintain the approach air flow at a constant temperature of 21°C with an uncertainty of ±0.5 ºC.

The cube had a height (H) of 0.015 m resulting in a blockage ratio of 0.75%. The cube itself had a copper core (12 × 12 × 12 mm³) around which an epoxy layer of 0.0015 m was applied on all surfaces (Fig. 1b). The copper core was heated at a constant temperature of 75°C by a dissipating source (resistance wire) that was placed inside the core. The temperature of the copper core was measured by a thermocouple with the uncertainty of ±0.1 ºC. Due to the high thermal conductivity of the copper, a uniformly distributed temperature at the interior of the epoxy layer was obtained in absence of wind.

The surface temperature distribution of the cube when exposed to the wind flow was measured with infrared thermography. The uncertainty of the measured surface temperature by using this method was within ±0.4 ºC.

The epoxy layer, applied on all surfaces of the cube, was used to be able to measure the surface temperature distribution and evaluate the surface heat flux and convective heat transfer coefficients. Meinders et al [5] used the Finite Volume Method to solve the equation for the three-dimensional heat conduction problem for the epoxy layer, when the uniform copper temperature (i.e. 75°C) and the surface temperature distribution (from infrared thermography measurements) were known. A heat balance between the heat conduction (from the epoxy layer to the surface of the cube) and the heat convection (from the cube surfaces to the air) yielded the local convective heat transfer coefficient.

Note that in this experiment the accuracy of the surface temperature measurements greatly depended on the resistance to conduction in the epoxy layer relative to the resistance to convection from its outer surfaces. The ratio of the two mentioned resistances is defined by a dimensionless number called Biot number, \( \text{Bi} = \frac{h L}{k} \), where \( h \) is convective heat transfer coefficient (W/m²K), \( L \) characteristic length (m) and \( k \) thermal conductivity of the solid (W/mK). A relatively small value of the Bi number for the epoxy layer could lead to uniform surface temperature. On the other hand, large values of the Bi number (e.g. rather thick layer) caused a too low surface temperature which was not suitable for accurate measurements by using infrared thermography as mentioned by Meinders et al [5].

The thermal conductivity of the epoxy layer was determined experimentally with hot-wire transient method (\( k = 0.237 \) W/mK) with an accuracy of ±5%. The cube was mounted on a base plate with a thermal conductivity of 0.33 W/mK to prevent excessive conductive heat losses from the cube to the floor.

![Figure 1. Experimental setup of Meinders et al. [5]: (a) Perspective view of cube model in turbulent channel flow (figure not to scale). (b) Detail of the heated cube. All dimensions are in meter.](image)
As the accuracy of the heat transfer coefficient measurement was sensitive to the epoxy layer thickness, special care was taken in the experiments to reduce the experimental uncertainties [5]. The layer was machined accurately with an uncertainty of 0.01 mm. Thermal expansion of the epoxy layer was also determined experimentally (6.8×10⁻⁵ °C⁻¹). To increase the accuracy of the infrared thermography measurements, the cube was painted with a black paint layer (0.06×10⁻³ m) with a high thermal conductivity to ensure the temperature decay across the paint-layer is negligible.

To generate the turbulent boundary layer flow, tripping strips were used which were located 5H upstream of the cube. The resulting vertical profile of mean wind speed for the turbulent boundary layer at the location of the building is represented by a log law with aerodynamic roughness length \( z_0 = 7.6 \times 10^{-6} \) m and a friction velocity \( u^* = 0.25 \) m/s. Laser Doppler Anemometry (LDA) was used to measure and document the flow field characteristics. The overall experimental uncertainties for the mean velocities and the Reynolds stresses were estimated to be 5% and 10%, respectively.

Experiments were performed under perpendicular approach flow and for several Reynolds numbers (based on the cube height \( H \)) ranging from \( 2 \times 10^3 \) to \( 5 \times 10^3 \). For the validation study, only the Reynolds number of \( 4.44 \times 10^3 \) was considered since also flow field data were available for this case. The average mass flow rate was 0.262 kg/s per unit area yielding an average bulk velocity of 4.47 m/s.

3. CFD simulations

3.1. Computational geometry and grid
A computational model is made of the cube including the epoxy layer used in the turbulent channel measurements. The size of the three-dimensional computational domain is based on the guidelines by Franke et al. [10] and Tominaga et al. [11]. The upstream and downstream domain lengths are \( 4H = 0.06 \) m and \( 10H = 0.15 \) m, respectively. The domain height was chosen equal to that of the channel in the experiments (= 3.3H) (Fig.1a). The vicinity of the channel ceiling to the cube might cause an artificial acceleration of the flow over the top surface and a suppression of turbulence. These phenomena can affect heat transfer process in the flow especially at the top of the cube [5]. The lateral extension of the domain is determined based on the required blockage ratio of the domain. The distance between the cube walls and the lateral planes of the domain is \( 10H = 0.15 \) m, resulting in a blockage ratio of 1.4%. This ratio is below the maximum blockage ratio of 3% recommended by the aforementioned guidelines [10-11].

The computational grid was generated using the surface-grid extrusion technique [12]. The procedure was executed with the aid of the pre-processor Gambit 2.4.6, resulting in a structured grid with 4,445,010 hexahedral cells (Fig. 2b). A total number of 50 cells with a uniform grid spacing (i.e. stretching ratio = 1) is applied along the cube surfaces in x-, y- and z-directions (Fig. 2b). These cubical cells are also used for the epoxy layer and extended to a distance of \( H/3 \) from the cube surfaces. Away from this distance a low stretching factor of 1.05 is applied. The grid resolution resulted from a grid-sensitivity analysis (not reported in this paper). The distance from the centre point of the wall adjacent cell to the wall for the cube planes and ground plane is \( 3 \times 10^{-4} \) m, corresponding to \( y^* \) values less than 6.9.

![Figure 2](image.png)

Figure 2. Perspective view of (a) of computational domain and (b) high-resolution grid at cube surfaces and part of the ground surface (total number of cells: 4,445,010).
3.2. Boundary conditions
Planes 1 and 3 in Fig. 2a are the inlet and outlet planes, while planes 2 and 4 are the side planes. In the simulations the inlet boundary conditions (mean velocity U, turbulent kinetic energy k and turbulence dissipation rate $\varepsilon$) were based on the measured incident vertical profiles of mean wind velocity U and longitudinal turbulence intensity $I_u$. To impose a time-dependent velocity profile at the inlet, the spectral synthesizer [13] is used. Zero static pressure is applied at the outlet plane. Symmetry conditions, i.e. zero normal velocity and zero normal gradients of all variables, are applied at the lateral sides of the domain. The ground and top of the domain as well as the inner and outer surfaces of the cube are defined as no-slip walls.

The thermal boundary conditions are an inlet air temperature of 294 K (21 ºC) and a fixed surface temperature of 348 K (75 ºC) for the inner surface of the epoxy layer. For the outer surface of the cube, a coupled boundary condition is used in which heat transfer is calculated directly from the solution in the adjacent cells of the fluid (air) and solid (epoxy layer). The bottom and top planes of the computational domain are adiabatic walls.

3.3. Solver settings
The commercial CFD code Ansys/Fluent 12.1 is used for both the RANS and LES simulations. For the 3D steady RANS simulations the realizable k-$\varepsilon$ model (Rk-$\varepsilon$) [14] was used in combination with the low-Re number Wolfshtein model [15]. The SIMPLE algorithm was used for pressure-velocity coupling, pressure interpolation was second order and second-order discretization schemes were used for both the convection terms and the viscous terms of the governing equations.

The standard Smagorinsky Subgrid-scale model [16] with $C_s = 0.1$ is applied and the bounded central-differencing scheme is used to discretize the convection term in the filtered momentum equation. Pressure-velocity coupling is performed with the fractional step method [17]. Time discretization is second-order implicit and the non-iterative scheme is used for time advancement.

The results of the steady RANS simulations were used as initialization for the LES simulations. Note that the LES and energy equations (including the transient heat conduction equation in the epoxy layer) are solved simultaneously and the same time step is used. In this study, the time step value ($\Delta t$) is $6 \times 10^{-4}$ s corresponding to a maximum Courant number of 8 in the whole domain. A suitable length for the averaging period has been determined by monitoring the net mean heat flux from all surfaces of the cube. After an initialization period $T_{\text{init}} = 1.5$ s corresponding to 26.79 flow-through times ($T_{\text{ft}} = L_x/U_h$, where $L_x$ is the length of the computational domain), the statistics are sampled for $T_{\text{avg}} = 29.03$ s = 519.6 $T_{\text{ft}}$.

4. Results
Figs. 3a and 3c compare the CFD results and the experimental results of surface temperature along the perimeter of a vertical and a horizontal cross-section by a plane cutting midway through the cube. To quantify the agreement between numerical and experimental results, the validation metric of Normalized Mean Square Error (NMSE) [18] is used (shown in Figs. 3b and d). For the windward facade the general agreement is quite good for both LES and steady RANS, though LES provides a lower NMSE. The same conclusion can be made for the leeward facade where steady RANS gives an over-estimation of the surface temperature.

For the top and sides of the cube, where flow separation and reattachment are very complex and intermittent, LES clearly provides much more accurate results than RANS. The main reason for these discrepancies can be incapability of steady RANS in predicting the inherently transient behavior of separation and recirculation.
Figure 3. (a and c) Comparison of simulated and measured surface temperature along lines on the cube surfaces. (b and d) NMSE along the same lines.

Figure 4 compares the simulated and experimental non-dimensional streamwise velocity distribution at the top of the cube along the lines $x/H=0.3$, $x/H=0.5$ and $x/H=0.7$. It can be seen that, in contrast to LES, steady RANS cannot accurately reproduce the reverse flow on the top of the cube. The recirculation zone predicted by RANS is smaller than the one measured experimentally and obtained by the LES simulations. A conclusion is that the surface temperature discrepancies on the top and side walls (Fig.3) can be related to the inaccurate flow field predicted by the RANS turbulence model.

Figure 4. Comparison of simulated (lines) and experimental (symbols) non-dimensional streamwise velocity distribution at the top of the cube along the lines: (a) $x/H=0.3$; (b) $x/H=0.5$; (c) $x/H=0.7$.

5. Conclusions
To evaluate the performance of LES compared to steady RANS, this paper presents LES and RANS CFD simulations of the surface temperature distributions at the surfaces of a reduced-scale cubic model measured in turbulent channel flow. The evaluation is based on a grid-sensitivity analysis and
on validation with the measurements by Meinders et al [5]. The results show that LES can accurately predict the surface temperature distributions of the cube walls. Steady RANS, however, indicates a satisfactory agreement with the experiments only for the windward surface.

References
[2] Ito N, Kimura K and Oka J 1972 A field experiment study on the convective heat transfer coefficient on exterior surface of a building ASHRAE Tran. 78 184–91