NUMERICAL STUDY OF THE BUILDING EXTERNAL CONVECTIVE HEAT TRANSFERS: EFFECTS OF BUILDING TOPOLOGY AND URBAN CONTEXT

Lucie Merlier, Christian Obrecht, Frédéric Kuznik, Gilles Rusauën, Julien Hans

1 CETHIL CNRS UMR5008, INSA-Lyon, F-69621, Villeurbanne, France
2 CSTB, Direction Energie Environnement, F-38400, Saint Martin d'Hères, France.

ABSTRACT
Usual building physics practices lack accuracy concerning the estimates of convective heat transfers coefficients (h_{cw}). Usual h_{cw} correlations overlook or insufficiently account for the construction shape and environment. This paper reports a numerical study developed to evaluate the distribution of h_{cw} on building facade depending on the construction shape and urban context. The computational fluid dynamics (CFD) and thermal models are validated by preliminary comparison with detailed experimental and computational data. The analysis of the actual h_{cw} distributions on the outer walls of a cube, courtyard building and a cube array stresses the dependence of h_{cw} to the building topology and surroundings.

INTRODUCTION
A wide diversity of urban fabrics exists in the world. Open, compact, and attached forms were built depending on the local geography, climate and culture. These urban patterns involve different very local micro-climates and air flows, which depend on the incident wind features and morphological properties of the built structure. They induce diverse distributions and intensities of convective heat transfer at building outer walls. Therefore, the resulting convective heat transfer coefficients (h_{cw}) at building facade depend on the construction shape and local urban configuration as well as the nature of convection and surface properties.

The knowledge of h_{cw} is especially useful when dealing with building energy performance issues. Convective heat transfers impact on the building energy loads, the convective cooling of building facades and integrated active systems, as well as the turbulent urban heat flux. Focusing on building energy problems, h_{cw} correlations used in common building thermal engineering often relate h_{cw} to a reference wind speed taken 10m high from the ground (U_{10}). Nevertheless, the free stream (U_{∞}) or the local (U_{loc}) velocities can also be considered, although U_{loc} is generally undefined. As shown in the review provided by Mirsadeghi et al. (2013), h_{cw} models used in building energy simulation tools are mainly based on experimental results. Corresponding models are often derived from experimental studies undertaken in laboratory and involving simplified configurations such as flat plates (e.g. (Jürges et al., 1924)). Such models substantially differ from real building configurations. Therefore, correlations derived from full-scale surveys carried out on real buildings (e.g. (Ito et al., 1972)) appear more appropriate. Nonetheless, they are case specific, might involve strong assumptions or may lack completeness about the experimental setup. This casts doubts upon the relevance of using common h_{cw}-U correlations in building energy simulation tools, and may induce substantially uncertain simulation results, especially for low insulated buildings. To overcome the above mentioned limitations and according to the review provided by Defraeye et al. (2011b) focusing on h_{cw} correlations for external building facades, detailed and dedicated numerical studies can now provide estimations of h_{cw} that are suited to building physics issues.

Extending this approach which focuses on an isolated cubical building, this paper reports a numerical study developed to evaluate the distribution of h_{cw} on building facades depending on the construction shape and urban environment.

The first part of this paper briefly reviews recent studies performed using computational fluid dynamics (CFD) to evaluate h_{cw} distributions on building facades and synthesizes the aerodynamic part of the study. The second part synthesizes the case studies developed to examine the convective heat transfers at building outer walls. It summarizes the computational modeling and discusses the simulation results. h_{cw} distributions are especially analyzed on the outer walls of a cube, a courtyard building and a cube array. Finally, the conclusion synthesizes the main points of the study and puts the results into perspective.

PRELIMINARY CONSIDERATIONS
Estimating h_{cw} using CFD

CFD models can theoretically account for any real or future geometry of buildings and environments. They also allow the control of the problem boundary conditions and are free of similarity requirement as models can be full-scale. Thus, this technique presents significant advantages over wind-tunnel and field studies. However, CFD studies have to be
carefully validated and verified to be relevant (Blocken, 2014). CFD models are often based on the Navier-Stokes equations and the control volume method. There are two ways of modeling the near wall flow and thermal behaviors. The low Reynolds number modeling (LRNM) is the most detailed approach. It solves the flow down to the wall, including the viscous sub-layer. This approach requires very fine cells next to walls to solve the boundary layer accurately. It is generally not possible to use such a fine mesh when addressing building physics problems. Therefore, studies commonly use wall-functions (WFs), which bridge the viscous and buffer sub-layers and model the flow and thermal behavior close to the wall using logarithmic correlations. The value of the dimensionless wall unit ($z^+$) determines the use of the first or the second method. For $z^+ < 1$, the LRNM applies, while for $30 < z^+ < 10^3$, the use of WFs is relevant. However, note that standard WFs (SWFs) usually fail to predict accurately convective heat transfers at building outer walls because the main temperature gradient occur in the viscous and buffer sub-layers (Blocken, 2009).

Based on detailed CFD modelings, Blocken et al. (2009) and Defraeye et al. (2011) suggested CFD LRNM-based $h_{cw} - U_{0}$ correlations, which address both the local distribution of $h_{cw}$ on the windward and leeward facades of isolated cuboid buildings or surface averaged values. Using large eddy simulation (LES), Liu et al. (2014) designed $h_{cw} - U_{0}$ correlations for the windward, leeward and top faces of non-isolated cuboid buildings that also depend on the packing density. In comparison with usual estimates of $h_{cw}$, the above-mentioned studies provide correlations for different faces of sharp-edged buildings that integrate the effects of the local aerodynamic conditions. As a result, their use in building physics is more relevant than commonplace $h_{cw} - U_{0}$ correlations.

In addition, when addressing specifically convective heat transfers at a given building outer walls, Defraeye et al. (2011a) highlighted the possibility of modifying usual WFs to better match LRNM results. On this basis, Defraeye et al. (2011a, 2012) and Allegretti et al. (2012) designed enhanced temperature WFs for k-ε models that fit LRNM predictions. Considered case studies are cuboid buildings or street canyons. These enhanced temperature WFs may apply in case of forced or mixed convection. As they have to be included into CFD models, these improved temperature WFs may provide accurate and case specific estimates of $h_{cw}$.

In order to analyze further the distribution and intensity of $h_{cw}$ and their relations with local flow structures, the following develops the study performed to evaluate the effects of the topological properties of constructions and of the urban environment on the building external convective heat transfers, by considering three case studies: a cube, a courtyard building and a cube array.

**Validation of the aerodynamic model and analysis of the main flow structures**

Prior to the study of convective heat transfers, a steady RANS aerodynamic model was validated by comparing simulation results to detailed experimental and computational data. This validation exercise is necessary to ensure the validity of the computational approach and to highlight its limits. Reference experiments are wind-tunnel tests, for which the CEDVAL database provides high resolution and high quality velocity measurements of the flow (CEDVAL, 2013). According to the best practice guidelines suggested in (Tominaga et al., 2008), the case studies considered are an isolated rectangular block and a multi-obstacle configuration with canyons (array of rectangular blocks). Note that in the latter case, the experimental documentation reports strong flow intermittency with no clear stable recirculation areas, which involved very long averaging periods. Reference numerical data were obtained using the lattice Boltzmann method (LBM) LES approach for these case studies (Obrecht et al., 2014). Due to its more detailed and transient formulation as well as a very fine spatial resolution compared to the steady RANS model, this numerical approach highlights in detail the dynamic of the flow and complete experimental data.

The accuracy of the steady RANS realizable k-ε turbulence model (Rk-ε) and a Reynolds stress model (RSM) together with SWFs are evaluated. Simulations were performed using Ansys Fluent 14.5 (Fluent, 2013). The computational domain reproduced the wind-tunnel test section. A preliminary simulation provided the actual approach flow profile, which matches the experimental mean velocity profile. The iterative convergence of the solution was assessed by monitoring several velocity profiles along the domain. Mesh sensitivity analyses were carried out to verify the minimal mesh resolution required to simulate accurately the aerodynamic flows.

Simulation results show that the Rk-ε predicts flow recirculation phenomena in the vicinity of the isolated obstacle more accurately than the RSM does. Conversely, the RSM better reproduces the near wake recirculation downstream the isolated obstacle. Nonetheless, the RSM still over-predicts the experimental cavity zone length. In addition, no clear stabilization of the mean velocity profiles occurs inside the canyon when using the RSM, which might be linked to the flow unsteadiness reported by the experimental documentation. The Rk-ε yields a better iterative convergence but this model does not reproduce the vortical structure in the canyon. Moreover, RSM predictions generally better agree with LBM LES data. Hence, being able to reproduce the main flow structures that develop around a sharpe-
edged obstacle and in more complex configurations, yet imperfectly, the RSM was used for subsequent simulations.

Based on the results of the CFD model validation, aerodynamic simulations were performed for a cube and courtyard building as well as a cube array. Each construction is 10m high. The two isolated buildings are archetypes of the most common building shapes (e.g. isolated housing or courtyard buildings). They are negative images of each other and show totally external or internal enclosed outdoor spaces. Accounting for additional constructions, the cube array represents an idealized urban environment. As such, these case studies highlight the effects of the topological and dimensional properties of constructions and the built environment on the aerodynamic field next to buildings.

A preliminary simulation of a long and empty tunnel provided the actual approach flow profile. The wind direction is perpendicular to the obstacles front faces, with $U_{10}=4.3\text{m}/\text{s}$. Recommendations given in (Blocken, 2014 and Tominaga et al., 2008) are respected in terms of domain size, mesh resolution and boundary conditions.

CFD simulation results highlighted the development of the basic flow structures that generally characterize flow fields around sharp-edged obstacles. More specifically, flow impingement, upwind standing vortices, flow separation on the obstacle leading edges and arch vortices in the cavity zone generally develop in the surroundings of the isolated cube and courtyard building as well as the cube array and privileged flow paths develop in alleys parallel to the approach flow direction. Internal vortical flow structures also develop inside the court of the courtyard building and between the cubes of the cube array.

**VALIDATION OF THE THERMAL MODEL**

**Part 1: LRNM approach**

When the flow and the obstacles walls show different temperatures, the flow structures developing around obstacles would induce different convective heat transfers on the surface of obstacles (Meinders et al., 1999). To study their distribution, heat transfers were coupled with the initial CFD model. The thermal model was preliminary validated by comparing simulation results to the experimental results of Meinders et al. (1999) for a 1.5m high cube. This cube is composed of copper core heated at 75°C, surrounded by a 1.5mm thick layer of epoxy and which is placed in a developing boundary layer. Such a model does not correspond to usual building physics configurations but there are only few published data on $h_{\text{c, w}}$ around sharp-edged obstacles. Furthermore, other building physics studies also refer to this reference work, including (Defraeye et al., 2010). As a consequence, some reference numerical data (mainly based on steady RANS Rk-$\varepsilon$ simulations) also exist.

The same computational strategy as presented in (Defraeye et al., 2010) but using the RSM was implemented. Rk-$\varepsilon$ simulations were performed as well to verify the accuracy of the model implementation with respect to the results of Defraeye et al. (2010). The epoxy layer was explicitly modeled while the copper core was simplified by setting a uniform wall temperature as this temperature was uniformly set in the experiment with low uncertainty. The computational model involves more than $4.8\times10^5$ cells in the epoxy layer and more than $3.9\times10^6$ cells in the fluid. Because of the small dimensions involved by the physical model, $z^*=1$ next to the outer faces of the cube. As a consequence, the enhanced wall treatment (EWT) was used instead of WF, which involves a LRNM approach.

![Image](Image.png)

**Fig. 1: Comparison of the simulated $h_{\text{c, w}}$ profiles with reference experimental (Meinders et al., 1999) and numerical (Defraeye et al., 2010) data on a 15mm high cube ($U_{\text{hub}}=4.47\text{m}/\text{s}$).**

Fig. 1 compares the simulated $h_{\text{c, w}}$ profiles on the streamwise mid-plane of the cube with the experimental data of Meinders et al. (1999) as well as the simulation results of Defraeye et al. (2010) obtained using the Rk-$\varepsilon$ and SST k-$\omega$. Both the numerical and experimental profiles match fairly well on the front and rear faces of the cube. However, $h_{\text{c, w}}$ profiles significantly deviate on the top face of the cube, where the experiment reports an increasing $h_{\text{c, w}}$ profile streamwise with a steep gradient. This face is located directly downstream the flow separation at the leading edge of the obstacle, and the prediction of the top separated bubble and the related turbulent processes is a well-known problem of steady RANS models, while being critical in determining heat transfers at this location. In the current case, the Rk-$\varepsilon$ predicts opposite variations but
a similar mean value. On the contrary, RSM-based $h_{c,w}$ predictions increase streamwise but with a slight slope and generally under-predict experimental data.

Comparing $h_{c,w}$ profiles on the horizontal mid-plane of the cube highlights similar trends (not shown). Experimental and numerical profile match well on the front and rear faces of the cube. They substantially deviate on the side face, where complex flow separation and reattachment processes physically occur. In addition, RSM-based profiles do not stabilize as functions of the number of iterations on the side face, where the experiment stresses strong flow unsteadiness. Simulation results generally under-predict experimental data. Nonetheless, the RSM yields rather realistic, but unstabilized, profile variation.

Hence, the RSM/LRNM approach provides satisfactory results on the front and rear faces of the cube. Results are less accurate on the top and side faces of the cube where complex flow separation processes occur.

**Part 2: Adapted temperature wall-function**

As it is generally not possible to use the LRNM when dealing with urban physics problems, additional steady RANS simulations were performed to evaluate the accuracy of WFs in predicting $h_{c,w}$. The considered case study is a 10m high cubic building for which Rk-$\varepsilon$/LRNM data are provided in (Defraeye et al., 2010, 2011) for $U_{10}$=0.5m/s$^3$. In addition, these studies performed LRNM simulations for other wind speeds up to 7.5m/s$^3$.

To evaluate the accuracy of the steady RANS RSM/SWFs, almost the same modeling strategy as presented in (Defraeye et al., 2011) was implemented. In particular, the design of the domain and the inflow properties were set as recommended in (COST, 2007 and Tominaga et al., 2008). $2 \times 10^6$ cells compose the mesh, which involves $x^+ = 4.5 \times 10^2$ on average on the windward face of the building. Additional simulations were performed using the Rk-$\varepsilon$ in order to compare the results with reference ones.

SWFs-based results substantially over-estimate $h_{c,w}$ on the different faces of the building compared to LRNM data, except on the top face. These results confirm the observations reported in (Defraeye et al., 2011). To improve the accuracy of WFs-based $h_{c,w}$ predictions with respect to LRNM results, the latter study proposed a customized temperature wall-function (CTWF). This CTWF addresses forced convection cases, applies for $50<\tau^+<500$ at least and works with the Rk-$\varepsilon$. The CTWF can be easily implemented into Ansys Fluent models by modifying the Pr$_{\kappa}$ value that is used in the SWF from 0.85 by default to 1.95. As the RSM is also a steady RANS model and the formulation of WFs is similar for the RSM and k-$\varepsilon$, it is assumed that a modification of the Pr$_{\kappa}$ value would also improve the accuracy of $h_{c,w}$ predictions.

![Fig. 2: Comparison of the simulated and reference LRNM $h_{c,w}$ profiles at 1m and 3m from the external edge of the cube front face.](image)

![Fig. 3: Comparison of the simulated and reference LRNM $h_{c,w}$ profiles at 1m and 3m from the external edge of the cube rear face.](image)

As expected, Fig. 2 shows that using the Rk-$\varepsilon$ with the modified value of Pr$_{\kappa}$, as suggested by Defraeye et al. (2011), yields $h_{c,w}$ predictions that match LRNM results. However, when using the RSM, this modification yields $h_{c,w}$ predictions that substantially under-estimate LRNM data (not shown).

Fig. 2 shows a satisfactory match when using the RSM together with a revised Pr$_{\kappa}$ =1.55 (rCTWF). More specifically, results show a very good match between the different CTWF-based and LRNM profiles on the cube front face. On the other hand, Fig. 3 shows that CTWF-based $h_{c,w}$ profiles slightly deviate on the rear face, which may be explained by differences in the prediction of the flow properties in the cavity zone. The resulting $h_{c,w}$ profiles differ in distribution and intensity from LRNM data, which may be explained by the effects of WFs as well as by the differences in the meshes used in the current or the reference study. Nevertheless, with respect to the estimation of the surface-averaged $h_{c,w}$ value based on the $h_{c,w}$-$U_{10}$ correlation of Defraeye et al. (2010) derived from the LRNM data, RSM/rCTWF results deviate by less than 10%. The deviations observed when using SWFs are about 30% when used with the Rk-$\varepsilon$, and 35% with the RSM.
As aerodynamic studies were performed accounting for \( U_{10} = 4.3 \text{ m/s} \) which involves high \( z^* \) at building outer walls, the accuracy of the rCTWF was also examined in case of \( U_{10} = 5 \text{ m/s} \). Following the same modeling strategy as previously, RSM-based \( h_{w} \) predictions on the front and rear faces of the cubic building were compared to the LRNM data of Defraeye (2009).

Results (not shown) highlight no significant loss of accuracy of the \( h_{w} \) predictions compared to LRNM data on the front face of the cube. However, RSM/rCTWF \( h_{w} \) profiles often over-predict RK, \\

\( \varepsilon \) LRNM data on the rear face by about 30\%. Nevertheless, the RSM/rCTWF model substantially improves the prediction accuracy compared to standard WFs. In comparison with the surface averaged \( h_{w} \) value estimated using the LRNM-based \( h_{w} = U_{10} \) correlation of Defraeye et al. (2010), predictions deviate by 13\% when based on the RSM/rCTWF model, while the use of SWF involves a 58\% deviation.

CASE STUDIES AND SIMULATION RESULTS

Case studies

Coupled thermal and aerodynamic simulations were performed for the cube, courtyard building and cube array in order to examine the relations between the flow structures that develop around constructions and \( h_{w} \) distributions. Simulations were performed using the same computational configuration as in the aerodynamic study in terms of geometry, mesh, approach flow profile and aerodynamic boundary conditions. The mesh resolution at building outer walls involves cell size of 15cm to 1m. Note that to keep \( z^* \) as close as possible to the initial range for applicability of the CTWF, the finer meshes addressed in the aerodynamic study, which correspond to the mesh sensitivity study, were considered. This involves \( 4.7 \times 10^6 \), \( 5.2 \times 10^6 \) and \( 16.6 \times 10^6 \) cells for the cube, courtyard building and cube array models respectively. The rCTWF is used to model near wall heat transfers. No radiative heat transfer is accounted for. Computations were initialized using the simulation results of the aerodynamic study. Such an initialization also avoids some numerical stability problems.

As shown in Fig. 4, the inflow temperature is \( T_{\text{in}}=15^\circ \text{C} \), the ground is adiabatic and the building model is simplified assuming an empty envelope with:

- an interior temperature: \( T_{\text{in}}=20^\circ \text{C} \);
- zero thickness outer walls: \( e_w=0 \text{m} \);
- and an equivalent and homogeneous interior \( h_{w} \) standing for the wall transmittivity : \( h_{w}=1 \text{W m}^{-2} \text{K}^{-1} \).

Given \( \Delta T_{\text{wall}}=5^\circ \text{C} \), \( U_{10}=4.3 \text{ m/s} \) and \( H=10 \text{m} \), \( R_i<1 \). This means that this is a case of predominant forced convection and buoyancy effects can be neglected. Therefore, the use of the constant density air model and the rCTWF is relevant.

Fig. 5 shows the distribution and surface averaged values of \( h_{w} \) on the different faces of the three case studies. Simulation results were post-processed using ParaView, based on the \( h_{w} \) values directly exported from the CFD modeling. Relatively similar \( h_{w} \) distributions occur on the cube and the external faces of the courtyard building. The highest \( h_{w} \) intensity occurs on the top of the obstacle front faces, where high flow velocities develop without recirculation. The lowest \( h_{w} \) reciprocally occurs in sheltered zones, and especially on the rear faces of the obstacles. The side and top faces of the theoretical buildings show clear recirculation effects, which are characterized by low \( h_{w} \) values.

Slightly lower \( h_{w} \) intensities are observed on the external faces of the courtyard building than on the cube. About 5\% lower surface averaged values are found on the front and sides faces of the courtyard building than on the cube. The decrease is of 10 to 23\% for the rear and top faces. These differences may mainly reflect dimensional effects. The larger the building is, the lower the surface averaged \( h_{w} \) on the front and rear faces of obstacles are. External walls of the cube array show \( h_{w} \) distributions and intensities that are almost comparable to those of isolated buildings, although being relatively smaller. On average, surface averaged \( h_{w} \) are reduced by 2, 11, and 40\% compared to those of the isolated cube on the front, side and rear faces of the cube array respectively. This decrease is also certainly explained by the increased size of the array compared to that of the isolated buildings.

The development of internal recirculation phenomena generally induces low \( h_{w} \). It is especially the case for the walls surrounding the court of the courtyard building. On average, \( h_{w} \) intensities are comparable to those occurring on the rear face of the building. \( h_{w} \) occurring within the court is well correlated to the recirculation that develops inside the courtyard. Maximum \( h_{w} \) occurs on the upper part of the windward face, leading to a higher surface averaged
Fig. 5: \( h_{CG} \) distribution on the cube, courtyard building, cube array for a wind incidence perpendicular to the front faces of the obstacles with \( U_{10} = 4.3 \text{ m/s} \).
The effects of recirculation phenomena and more generally of different flow regimes on $h_{cw}$ also explain the distribution of $h_{cw}$ observed within the cube array. On average, $h_{cw}$ on the central cubes of the first line of the array is similar to that of the isolated cube. Slightly lower $h_{cw}$ are nonetheless observed on the front and rear faces of these cubes, but higher surface averaged $h_{cw}$ occur on the top and especially on the sides of these cubes compared to that of the isolated cube. This may be related to the intensification of the corner streams in alleys. The upstream line of the array shows substantial differences between the surface averaged $h_{cw}$ on the different faces of each cube because of the incident flow impingement on the front faces and the recirculation phenomena that develop downstream.

$h_{cw}$ intensities are lower downstream in the array, $h_{cw} = 11.6$ W$m^{-2}K^{-1}$ on average on the cubes located in the first line on the array, whereas $h_{cw} = 7.8$ W$m^{-2}K^{-1}$ in the fourth line. Surface averaged $h_{cw}$ intensities are also rather homogeneous on buildings as faces are sheltered from winds by the surrounding obstacles and recirculation phenomena develop. The air may be heated up by surrounding obstacles as well, so that wind and thermal conditions are more homogeneous inside the array. However, windward faces still show higher $h_{cw}$ than the others, and are similar to those occurring on the corresponding roof. The rear face of obstacles generally show the lowest $h_{cw}$.

Hence, $h_{cw}$ distributions on building outer walls are strongly correlated to the flow structures that develop around constructions and inside courtyards. In particular, flow impingement on surface leads to high $h_{cw}$ intensities and the development of recirculation phenomena yield low $h_{cw}$ intensities. The surface averaged $h_{cw}$ value on the four sides of the courtyard building is 50% lower than the averaged $h_{cw}$ value on the four vertical walls of the cube. The reduction is of 47% when considering the windward and leeward faces only, as the thermal model was only validated for these faces. As a consequence, the building topology is crucial in defining the distribution of convective heat transfers on its envelope in case of an isolated construction.

Similarly, the relative sheltering provided by surrounding constructions is critical in determining the development of convective heat transfers at building facade. As mean wind speeds are generally reduced and recirculation phenomena develop, $h_{cw}$ intensities are lower inside the array than for isolated building configurations. The averaged $h_{cw}$ value on the four vertical walls of a cube located in the theoretical urban environment is 35% lower than the corresponding value observed for the isolated cube. The reduction is of 38% only considering the front and rear faces.

**CONCLUSIONS**

This paper reports a numerical study performed to examine the effects of the topology and built environment of constructions on $h_{cw}$ in case of forced convection. A steady RANS RSM together with the rCTWF are used.

Common $h_{cw}$-U correlations used in building thermal engineering practices are generally not based on such a dedicated approach, and do not or badly account for the effects of the construction shape and environment. However, results highlight substantial effects of the built morphology on the intensity of building convective heat transfers. In particular, the development of recirculation phenomena next to building outer walls substantially decreases $h_{cw}$ intensities on building outer walls, potentially up to 50%. Such differences may significantly affect building energy loads, especially in cases of low insulated buildings. It may also affect the convective cooling of building outer surfaces and of integrated active systems such as building integrated photovoltaic systems. Accurately estimate $h_{cw}$, using e.g. computational studies such as the current one, appears therefore necessary.

However, current results are based on a steady RANS approach. If they provide $h_{cw}$ values that are certainly more accurate than standard values, especially on the front and rear faces of buildings, further studies should consider more wind speeds and directions to provide more applicative results with a wide scope of applicability. Natural convection effects should also be examined, especially for low wind speeds. In addition, results may be significantly improved by the use of more detailed computational methods, such as the LES. This computational method does not suffer from the well-known limitations characterizing steady RANS models, including the RSM though this model accounts for anisotropic effects of turbulence. In particular, the reproduction of the dynamic behavior of the flow and a better modeling of turbulence would substantially improve $h_{cw}$ predictions, as these factors are critical in determining heat transport in turbulent flows around sharp-edged obstacles. Although such studies are
Currently difficult to generalize because of the computational resources they require, recently developed and cost-effective models such as the LBM-LES method by Obrecht et al. (2014) are promising.

**NOMENCLATURE**

- CFD = Computational Fluid Dynamics
- EWT = Enhanced wall treatment
- \( h_{c,w} \) = Convective heat transfer coefficient
- LBM = Lattice Boltzmann Method
- LES = Large eddy simulation
- LRNM = Low Reynolds number modeling
- RANS = Reynolds averaged Navier-Stokes
- \( k-\varepsilon \) = Realizable \( k-\varepsilon \) turbulence model
- RSM = Reynolds stress model
- \( \text{SST} \ k-\omega \) = Shear Stress Transport \( k-\omega \) model
- \( \text{Pr}_{k,w} \) = Wall turbulent Prandtl number
- \( U_{\text{v}} \) = Streamwise mean velocity (at 10 m high)
- \( z^* \) = Dimensionless wall unit
- \( z_0 \) = Roughness length

**ACKNOWLEDGMENT**

Authors thank the CETHIL and the CSTB for funding this work, and T. Defraeye for providing the detailed LRNM results used for the thermal model validation.

**REFERENCES**

- Defraeye, T., 2009. LRNM-based convective heat transfer coefficients on the windward and leeward faces of a 10 m cubical building.
- Jürges, W., 1924. The heat transfer at a flat wall (der Wärmeübergang an einer ebenen wand). Beihette zum Gesundh-Ingr 19.