Building and Environment 46 (2011) 2130–2141

Contents lists available at ScienceDirect

Building and Environment

journal homepage: www.elsevier.com/locate/buildenv

# An adjusted temperature wall function for turbulent forced convective heat transfer for bluff bodies in the atmospheric boundary layer

Thijs Defraeye [a,b,*], Bert Blocken [c], Jan Carmeliet [d,e]

[a] Division of Mechatronics, Biostatistics and Sensors, Department of Biosystems, Katholieke Universiteit Leuven, Willem de Croylaan 42, 3001 Heverlee, Belgium
[b] Laboratory of Building Physics, Department of Civil Engineering, Katholieke Universiteit Leuven, Kasteelpark Arenberg 40, 3001 Heverlee, Belgium
[c] Building Physics and Systems, Eindhoven University of Technology, P.O. Box 513, 5600 Eindhoven, The Netherlands
[d] Chair of Building Physics, Swiss Federal Institute of Technology Zurich (ETHZ), Wolfgang-Pauli-Strasse 15, 8093 Zürich, Switzerland
[e] Laboratory for Building Science and Technology, Swiss Federal Laboratories for Materials Testing and Research (Empa), Überlandstrasse 129, 8600 Dübendorf, Switzerland

ARTICLE INFO

*Article history:*
Received 27 February 2011
Received in revised form
11 April 2011
Accepted 11 April 2011

*Keywords:*
Wall function
Computational fluid dynamics
RANS
Convective heat transfer coefficient
Cube
Urban heat transfer

ABSTRACT

Accurate convective heat transfer predictions are required in building engineering and environmental studies on urban heat islands, building energy performance, building-envelope durability or conservation and (natural) ventilation of buildings. When applying computational fluid dynamics (CFD) for these computationally-expensive studies at high-Reynolds numbers, wall functions are mostly used to model the boundary-layer region. In this study, an adjustment to the standard temperature wall function is proposed for forced convective heat transfer at surfaces of typical wall-mounted bluff bodies in turbulent boundary layers, such as the atmospheric boundary layer, at moderate to high Reynolds numbers. The methodology to determine this customised temperature wall function (CWF) from validated numerical data of CFD simulations using low-Reynolds number modelling (LRNM) is explained, where a logarithmic-law behaviour is found. The performance of this CWF is evaluated for several bluff-body configurations. Standard wall functions (SWFs) yield deviations of about 40% for the convective heat transfer coefficient, compared to LRNM. With the CWF however, these deviations are reduced to about 10% or lower. The CWF therefore combines increased (wall-function) accuracy for convective heat transfer predictions with the typical advantage of wall functions compared to LRNM, being a lower grid resolution in the near-wall region, which increases computational economy and facilitates grid generation. Furthermore, this CWF can be easily implemented in existing CFD codes, and is implemented in the commercial CFD code Fluent in this study.



## 1. Introduction

Forced convective heat transfer at surfaces of wall-mounted bluff bodies in turbulent boundary layers at moderate to high Reynolds numbers ($Re = 10^4$–$10^7$) is of interest in many engineering applications, such as the evaluation of wind-induced convective heat losses from building surfaces or building components (e.g. solar collectors) in the atmospheric boundary layer (ABL). In building and urban engineering, convective heat transfer predictions are especially relevant for ABL flow applications on urban heat islands [1,2], building (component) energy performance [3,4], building-envelope durability or conservation [5,6] and (natural) ventilation of buildings [7]. Furthermore, they can be used to estimate the convective moisture transfer from building surfaces, by using the heat and mass transfer analogy [8]. Convective moisture transfer is especially of interest for hygrothermal analysis of building envelopes and for urban applications involving evaporation of water from ponds, roof ponds, green roofs, green walls or surfaces which are wetted by (wind-driven) rain [9].

Convective heat transfer research for this type of flow problem is mainly performed by wind-tunnel experiments [10–14] and by computational fluid dynamics (CFD) studies [15–17] for both isolated bodies as well as arrays of bluff bodies. Compared to wind-tunnel experiments, CFD simulations have the advantage that usually a higher spatial resolution is obtained. With CFD also no restrictions are imposed regarding scaling and accessibility of certain surfaces, in contrast to wind-tunnel infrared thermography measurements [11] for example, which is especially important for more complex configurations. On the other hand, the applied numerical modelling approaches determine to a large extent the accuracy of CFD simulations.

* Corresponding author. Division of Mechatronics, Biostatistics and Sensors, Department of Biosystems, Katholieke Universiteit Leuven, Willem de Croylaan 42, 3001 Heverlee, Belgium. Tel.: +32 (0) 16 321618; fax: +32 (0) 16 322966.
*E-mail address:* thijs.defraeye@biw.kuleuven.be (T. Defraeye).

0360-1323/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.buildenv.2011.04.013