ELSEVIER

Journal of Wind Engineering
and Industrial Aerodynamics 94 (2006) 699–723

JOURNAL OF wind engineering AND industrial aerodynamics

www.elsevier.com/locate/jweia

Numerical modeling of the flow conditions in a closed-circuit low-speed wind tunnel

Peter Moonen[a,*], Bert Blocken[a], Staf Roels[a], Jan Carmeliet[a,b]

[a]*Laboratory of Building Physics, Department of Civil Engineering, Katholieke Universiteit Leuven, Kasteelpark Arenberg 40, 3001 Leuven, Belgium.*

[b]*Building Physics and Systems, Faculty of Building and Architecture, Technical University Eindhoven, P.O. Box 513, 5600 MB Eindhoven, The Netherlands.*

Received 27 March 2005; received in revised form 15 November 2005; accepted 13 February 2006
Available online 29 March 2006

Abstract

A methodology for numerically simulating the flow conditions in closed-circuit wind tunnels is developed as a contribution to the general philosophy of incorporating Computational Fluid Dynamics (CFD) in wind tunnel design and testing and to CFD validation studies. The methodology is applied to the full-scale Jules Verne climatic wind tunnel in which experimental data have been obtained. Due to the specific features of this closed-circuit wind tunnel, the conventional CFD modeling approach, in which only the flow in the wind tunnel test section is modeled, is inadequate. To obtain accurate results the entire wind tunnel has to be modeled. In the numerical closed-circuit wind tunnel, the conventional flow inlet and outlet are replaced by a single ''fan boundary condition''. Special attention is given to the theoretical background and the practical implementation of this type of boundary condition in the CFD model. The numerical model is validated for the case of an empty wind tunnel and for the case in which a block-type building is placed in the test section. It will be shown that this methodology can generally reproduce the wind tunnel measurements of mean velocities with an error equal to or less than 10% despite the occurrence of multiple flow separations upstream of the test section. This provides perspectives for the future use of this methodology as a tool for wind tunnel design and testing and for CFD validation purposes.

Keywords: Computational Fluid Dynamics (CFD); Numerical simulation; Fan boundary condition; Wind tunnel design; Testing; Validation; Wind flow; Air flow; Building model; Test section flow quality

*Corresponding author. Tel.: +32 16 321346; fax: +32 16 321980
E-mail address: peter.moonen@bwk.kuleuven.be (P. Moonen).

0167-6105/$ - see front matter
doi:10.1016/j.jweia.2006.02.001

Nomenclature

a_0	first polynomial coefficient of fan performance curve, Pa
a_1	second polynomial coefficient of fan performance curve, Pa. s/m
A_{fan}	cross-sectional area of the wind tunnel at the fan position, m^2
C_{KS}	roughness constant, –
G_a	volumetric flow rate, m^3/s
I_U, I_V, I_W	turbulence intensity in streamwise, vertical and lateral direction, %
k	turbulent kinetic energy, m^2/s^2
K_S	physical roughness height, m
ℓ	number indicating wind tunnel section, -
L	length, m
L_ℓ	length of wind tunnel section ℓ, m
P	pressure, Pa
ΔP_{tot}	total pressure loss, Pa
R_ℓ	frictional losses per unit length, Pa/m
u, v, w	instantaneous velocity components in streamwise, vertical and lateral direction, m/s
U, V, W	average velocity components in streamwise, vertical and lateral direction, m/s
w, d, h	width, depth and height of the building model, m
W, H	width and height of a cross-section in the wind tunnel, m
X, Y, Z	streamwise, vertical and lateral cartesian co-ordinates, m
y^+	dimensionless wall unit, –

Greek letters

ε	turbulence dissipation rate, m^2/s^3
Φ	diameter, m
ρ	air density, kg/m^3
ξ	loss coefficient, $(Pa.\,s^2)/m^6$

1. Introduction

Wind tunnel testing is a well-established discipline in Wind Engineering [1–6] and it is applied for a wide range of Wind Engineering studies [7–13]. In spite of the vast increase of computing performance in the past decades and contrary to what has been suggested in the past, CFD has not succeeded in replacing the wind tunnel, as adequately indicated by Meroney et al. [14] and by Stathopoulos [15]. In spite of not living up to earlier expectations, it can be stated that CFD has reached a stage in which its direct combination with the wind tunnel approach can yield a synergistic effect. On the one hand, wind tunnel measurements are being used to validate CFD calculations performed with different turbulence models, different near-wall modeling approaches, etc. On the other hand, CFD can be used as a tool to support wind tunnel design, wind tunnel testing and the