Building and Environment 58 (2012) 23–36

Contents lists available at SciVerse ScienceDirect

Building and Environment

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

Sensitivity of night cooling performance to room/system design: Surrogate models based on CFD

Kim Goethals [a,*], Ivo Couckuyt [b], Tom Dhaene [b], Arnold Janssens [a]

[a] *Research Group of Building Physics, Construction and Services, Department of Architecture and Urban Planning, Faculty of Engineering and Architecture, Ghent University, J. Plateaustraat 22, B-9000 Ghent, Belgium*

[b] *Internet Based Communication Networks and Services Research Group, INTEC, IBBT-Ghent University, Gaston Crommenlaan 8 Box 201, B-9050 Ghent, Belgium*

ARTICLE INFO

Article history:
Received 14 March 2012
Received in revised form
14 June 2012
Accepted 22 June 2012

Keywords:
Night cooling
Convective heat transfer
Design
Computational fluid dynamics
Surrogate modelling

ABSTRACT

Night cooling, especially in offices, attracts growing interest. Unfortunately, building designers face considerable problems with the case-specific convective heat transfer by night. The multizone building energy simulation programs they use actually need extra input, from either costly experiments or computational fluid dynamics (CFD) simulations. Alternatively, up-front research on how to engineer best night cooled spaces can thrust the application of night cooling. The authors of underlying paper set up a global surrogate-based optimization procedure to find room/system design solutions which induce a high convective heat flux during night cooling in a generic open plan office. This fully-automated configuration of data sampling, geometry/grid generation, CFD solving and surrogate modelling generated several surrogate models. These surrogate models indicated how the convective heat flux in the night cooled open plan office related to the ventilation concept, the thermal mass distribution, the geometry and the driving force for convective heat transfer. Actually these surrogate models merely guided the data sampling towards the global optimum. However, they also provided additional rough-hewn insights into the global behaviour. The results indicated that cases with thermal mass at the floor produce convective heat fluxes which are significantly higher than the ones with thermal mass at the ceiling. Among these cases, the performance of cross ventilation surpasses the ones of both single sided ventilation and under floor ventilation. Among the cases with a thermally massive ceiling, single sided ventilation seems superior. However, while cross ventilation is generally a robust ventilation concept, single sided ventilation is particularly sensitive to the geometry.

1. Introduction

Growing interest lies in passive cooling techniques, especially night cooling. After all, night cooling improves the summer comfort and minimizes the need for mechanical cooling. At night, natural or mechanical ventilation cools down the building fabric. The following day, the thermal mass absorbs the heat, by which peak temperatures are reduced and delayed [1]. For optimal performance three basic elements are necessary: the supply of cold air, the ability to store heat and the related heat transfer. Especially the convective heat transfer during nighttime plays a key role. Unfortunately, today's customary design tools, i.e. building energy simulation (BES) programs, cannot grasp this case-specific convective heat transfer, at least not without extra input [2,3]. Setting up new costly experiments to derive convective heat transfer correlations is not feasible for designers. Also conflating computational fluid dynamics (CFD) with BES is hardly realistic in rapid building design because of the large computation time. Without doubt, researchers and software developers work hard on ways out. However, they will probably not succeed overnight. In the meantime, deploying CFD to investigate to the convective heat transfer in specific case problems can provide new insights and might inspire other studies.

Yet, it is still necessary to limit the number of simulations. One popular way is to apply approximation methods to produce a model which to some extent comes close to some part of the (unknown) reference model (i.e. local modelling) or is accurate over the complete design space (i.e. global modelling). In particular the so-called data-driven approximation methods are prevalent. They disregard the dynamics of the deterministic simulation model (or better, simulator) and focus on the input–output relationship. The drawback is that they lack traceability. Data-driven modelling is also often referred to as surrogate modelling and can be subdivided into forward and inverse surrogate modelling. Forward surrogate

* Corresponding author. Tel.: +32 9 264 78 61; fax: +32 9 264 41 85.
E-mail address: kim.goethals@ugent.be (K. Goethals).

0360-1323/$ – see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.buildenv.2012.06.015

modelling approximates the response of the simulator to a set of design parameters while inverse surrogate modelling starts from the target performance and tries to find the corresponding input values. Another distinction of surrogate modelling relies on whether the surrogate model itself is the goal (global) or it is used to drive an optimization (local). Yet, also intermediate forms exist. The workflow of surrogate modelling is always pretty much the same, but interpretation of each step challenges even experts in the field, let alone laymen like building engineers. Last-mentioned group of people merely want a globally/locally accurate surrogate model as fast as possible, with minimal overhead. Guidance on selecting and setting up such techniques or perhaps even a ready-made computer program is no luxury for them. One such convenient computer tool connecting the two worlds is the Matlab SUrrogate MOdelling (SUMO) toolbox [4]. Successful applications of this toolbox are plentiful: e.g. optimization of microwave filters and identification of electrical properties of textile antennas [5] and blood flow data modelling [6].

Underlying study is just another such application, now in building engineering. It intends to find with the aid of a so-called global surrogate-based optimization (SBO) procedure room/system design solutions which induce a high convective heat flux during night cooling in a generic open plan office (i.e. a large open floor where several persons work). Night cooling is frequently applied in open plan offices, because they are unoccupied at night and require limited additional investments costs [7]. To this end, the authors set up a fully-automated framework of data sampling (SUMO [4]), geometry and grid generation (Gambit [8]), CFD solving (Fluent [8]) and surrogate modelling (SUMO), which generated several surrogate models. These surrogate models indicated how the convective heat flux in the night cooled open plan office related to several room/system design parameters, which were subdivided into ventilation concept, thermal mass distribution, geometry and driving force for convective heat transfer. Strictly speaking, these surrogate models merely guided the data sampling towards the global optimum. However, they also provided additional rough-hewn insights into the global behaviour. In addition, these surrogate models could help to improve BES modelling in two ways. They indicated profitable design solutions for which new convective heat transfer correlations could be derived. Or, derived more globally accurate surrogate models could be coupled with BES.

2. Experiment design

2.1. Simulation experiment setup

2.1.1. Annex 20 2-D case as a starting point

Open plan offices usually have a large longitudinal section compared to the crosscut and often have line-shaped diffusers and band windows. This leads, roughly speaking, to 2-D airflow, indeed influenced by 3-D eddies. So, it is not a bad choice to limit the problem to a 2-D case. This study started from the 2-D Annex 20 case [9] (Fig. 1). This test case was basically a rather long ($L_r/H_r = 3.0$) and wide ($W_r/H_r = 1.0$) ventilated room with a wall-to-wall opening on either side. The supply on the left side was a quite high channel ($h_{sup}/H_r = 0.056$); which obviously differed from practical diffusers. However, this simple description led to a fully-developed flow between two walls, which in simulations did not necessitate an approximating diffuser model and still relaxed the number of grid points near the opening [10]. The height of the exhaust opening on the right h_{exh} was to the height of the room H_r as 0.16 to 1. In the Annex 20 2-D2 test case, the supply temperature T_{sup} equalled 20 °C, the air change rate was 10.2 h^{-1} and the constant heat flux added along the floor was raised in successive

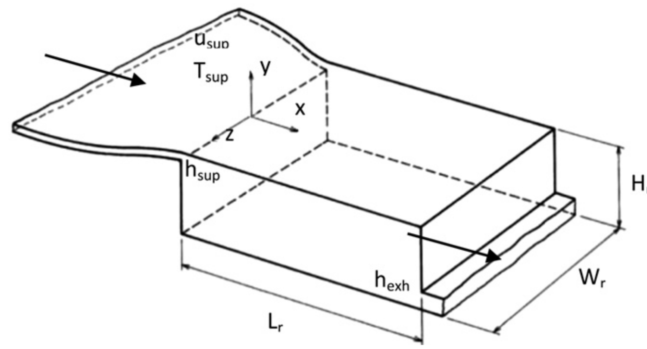

Fig. 1. Blueprint of Annex 20 2-D experiment setup [21].

experiments. The critical factor was the impact of the Richardson number (Equation (1)) on the jet penetration at the midplane.

$$Ri = \frac{\beta \cdot g \cdot h_{sup} \cdot (T_w - T_{sup})}{u_{sup}^2} \tag{1}$$

2.1.2. Parameterization

This simple reference case enabled a straightforward parameterization. Table 1 shows all the considered design parameters with their respective categories/continuous intervals. As previously mentioned, underlying study identified among the design parameters four subsets: ventilation concept, mass distribution, geometry and driving force for convective heat transfer.

The first subset, i.e. ventilation concept, included only a single input parameter, which in addition was categorical (i.e. without numerical meaning). There were three discrete possibilities: cross, single sided and under floor ventilation. Another such subset was the location of the isothermal plane. Here, the choice between floor and ceiling meant that the quoted surface with zero thickness was at a higher temperature than the supply air while the remaining surfaces, also with zero thickness, behaved adiabatically (Fig. 2). Actually, in the simulations with constant boundary conditions the one warm surface represented a thermally heavy finishing while the adiabatic surfaces corresponded to light structures. The idea behind it was that a specific airflow pattern corresponds to one set of boundary conditions, no matter what the previous state was. The subset geometry comprised the length of the room L_r, the distance of the supply to the zero point (0,0) H_{sup}, the distance of the exhaust opening to the zero point (0,0) H_{exh}, the height of the supply/exhaust $h_{sup/exh}$ and the inclination of the supply α (Fig. 2). Here, continuous numerical intervals applied, in contrast with the previous two categorical parameters (Table 1). Note that, for programming simplicity, the bounds of H_{sup} and H_{exh} were corrected for the boundary layer thickness BL. For that same reason, H_{sup} was limited to 4.0 m in case of under floor ventilation. Also mark in Fig. 2 that underlying study constantly made use of a quite

Table 1
Overview of parameters for sensitivity study.

Parameter	Type	Min	Max
–	Ventilation concept	Cross/single sided/under floor	
–	Location isothermal plane	Floor/ceiling	
L_r (m)	Geometry	4.5	9
H_{sup} (m)		0+BL	2.6−BL (4.0)
H_{exh} (m)		0+BL	2.6−BL
$h_{sup/exh}$ (m)		0.1	0.5
α (°)		60	120
$(T_w - T_{sup})$ (°C)	Driving force	1	10
n (h^{-1})		1.5	10