Building and Environment 75 (2014) 30–39

Contents lists available at ScienceDirect

Building and Environment

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

Coupling building energy simulation and computational fluid dynamics: Application to a two-storey house in a temperate climate

Mathieu Barbason*, Sigrid Reiter [1]

LEMA, Université de Liège, Chemin des Chevreuils, 1, 4000 Liège, Belgium

ARTICLE INFO

Article history:
Received 25 November 2013
Received in revised form
17 January 2014
Accepted 21 January 2014

Keywords:
Computational fluid dynamics
Multizone model
Building physics simulations
Validation
Overheating risks

ABSTRACT

This article reports the coupling of a building energy simulation (BES) made with TRNSYS with a computational fluid dynamics (CFD) simulation made with ANSYS FLUENT and its application to a typical Belgian two-storey house. The coupling scheme developed in this study aims to improve the overheating prediction for buildings. This phenomenon is becoming increasingly frequent in Northern Europe due to increased insulation and a lack of sun protection and natural cooling strategies. Complementary contributions of the two numerical approaches are underlined and used to obtain accurate results in an acceptable computing time, even in a thermally stratified room. The space and time coupling is discussed to obtain an optimised tool in which BES is in charge of the primary portion of the effort, while CFD intervenes punctually on one room of interest. The numerical results are compared both qualitatively and quantitatively to the experimental results, and the improved accuracy of the coupled tool compared with a standalone BES is underlined.

1. Introduction

The reduction of a building's energy consumption has become one of the most challenging goals worldwide. This problem is especially challenging in Europe, where 40% of the total energy is dedicated to the heating and cooling of buildings [1]. Therefore, building designers are urged to use new strategies to develop near-zero energy buildings. In fact, European regulations will impose the building of zero-energy buildings as soon as 2020.

The scientific community has developed several approaches for building energy simulations (BES) to help building designers, such as multizone dynamic simulations. In parallel, aeronautical studies have led to the development of computational fluid dynamics (CFD) models that could be applied to building cases. These approaches aim to optimise building design and retrofitting. Nevertheless, they have a number of limitations and drawbacks.

BES is widely used due to its ease and speed. Chen [2] shows that multizone models have been the main tools for predicting ventilation performance in an entire building over the past years. The multizone model allows the prediction of overall flow through the building and the prediction of mean temperature in small rooms, but it cannot predict detailed temperature and airflow distributions within each room. Specifically, the multizone approach assumes the perfect mixing of the air in each zone, which generally corresponds to a perfect mixing of air in each room. Thus, this approach suffers from a lack of accuracy for thermally stratified rooms. However, recent architectural designs have developed massively glazed buildings and atrium configurations. This type of configuration significantly increases the risk for overheating and thermal stratification in large or high rooms. Thus, multizone models are not reliable for this type of building. Some studies have already extensively discussed the multiple assumptions and drawbacks of the multizone model [3–5].

Conversely, computational fluid dynamics software has already proven its ability to accurately model all types of aero-thermal phenomena in buildings and their surroundings, such as mechanical ventilation [6–8], natural ventilation [9,10], contaminant dispersion [11,12], airflow around buildings [13], heat islands [14], etc. Chen [2] shows that CFD models have been mainly applied to study indoor air quality, natural ventilation and stratified ventilation because these phenomena were difficult to predict via other models.

As the most sophisticated airflow model, CFD simulations can provide detailed spatial distributions of air velocity, temperature and contaminants in each room. Unfortunately, they suffer from long calculation times, especially for highly partitioned buildings, which limit their adoption by practitioners. Moreover, Li and Nielsen [15] argue that large efforts must be devoted to raising

* Corresponding author. Tel.: +32 4 366 9360; fax: +32 4 366 2909.
E-mail addresses: mbarbason@ulg.ac.be (M. Barbason), sigrid.reiter@ulg.ac.be (S. Reiter).
[1] Tel.: +32 4 366 9482; fax: +32 4 366 2909.

0360-1323/$ – see front matter © 2014 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.buildenv.2014.01.012

awareness on the reliability of these techniques, defining good practice rules and helping new operators to suitably use these techniques. Several recent studies have attempted to address this issue, especially in terms of the selection of turbulence models [6,16–18].

To meet the needs of building designers, the scientific community has studied the complementarity of these two tools: the ease and speed of the multizone approach and the accuracy of CFD. These efforts yielded simulations that couple BES and CFD. The coupling allows operators to exploit the qualities of each approach. However, this requires overriding three major discontinuities between the two techniques [19]:

1. Spatial discontinuity meshes are completely different;
2. The temporal discontinuity: the multizone approach can easily be used to conduct studies over an extended period of time (several months to a year calculated per time step of an hour), while the CFD solves simulations over relatively short periods (a few hours, calculated per time step of a second);
3. Discontinuity of computation time between the two approaches.

This article aims develop a coupled BES–CFD approach that is optimised for overheating predictions in complete buildings. This tool is based on two of the most widespread software programs: TRNSYS [20] for the BES part and FLUENT (ANSYS Inc. [21]) for the CFD part.

An application of this tool to a Belgian residential building (two floors and 11 rooms) on a sunny summer day will allow its numerical results to be compared with experimental measurements in the studied house, as well as with the numerical results of a standalone BES simulation. This article is structured as follows: Introduction, State of the art of BES–CFD coupling, Case study description, Developed coupling tool description, Simulation parameters of the case study, BES standalone results, Coupled results and Conclusions.

2. BES–CFD coupling – state of the art

The idea of coupling BES and CFD was first developed by Negrão [22], who focused on the necessity for the two models to exchange appropriate boundary conditions. Indeed, CFD requires surface temperature to accurately describe the flow condition, but these values are unknown at the design stage. Conversely, BES requires convective heat flux coefficients for each wall and thermal gradients description within rooms with high vertical development. This study developed 3 models. In the first one, the two approaches run in parallel without direct interaction. In the second one, BES provides the surface temperature to CFD, while CFD supplies convection heat transfer fluxes. Finally, this study attempted to add airflow exchanges between small rooms and the open space to the coupling approach. Unfortunately, this last technique was not a success.

Zhai et al. [19] reviewed several coupling approaches and classified them based on their coupling iterative process and the number of data exchanges between the two tools. In a static coupling, the study focuses on one particular moment of interest for which BES generally provides wall temperatures to CFD. A reverse data exchange, from CFD to BES, may also help to improve accuracy. In a dynamic coupling, BES and CFD exchange useful data at several time steps to capture the transient phenomena. Zhai et al. [19] considered 4 exchange protocols that will be explained in detail hereafter. Finally, they applied these different schemes to two single room cases. They underlined that a coupling approach can significantly improve the cooling/heating loads and comfort predictions.

Zhai and Chen [23] verified the existence, uniqueness and convergence of a solution obtained by a coupling approach based solely on thermal aspects. They tested several exchanges parameters (such as convective heat transfer coefficients or heat fluxes from CFD to BES). Finally, they claimed that the most stable approach was to transfer the surface temperature from BES to CFD and convective heat transfer coefficients from CFD to BES.

Wang and Chen [24] have expanded on the study of Zhai and Chen [23] to include airflow exchanges parameters. They claimed that the stable approach consisted of exchanging the pressure boundary conditions between BES and CFD and vice versa. Wang [25] pursued the development of this approach by studying contaminant dispersion in a four-room case. He demonstrated that a coupling approach yields more realistic results than standalone BES software. In parallel, Djunaedy et al. [26] noted that internal coupling (in which BES and CFD are assembled in one single tool) has limitations that can be overcome by the use of an external coupling (in which BES and CFD work sequentially). This approach drastically decreases the computing time and improves the accuracy of results.

Pappas and Zhai [27] studied the performance of a double skin cavity with a coupled BES–CFD programme: the model was validated using measured data, and errors were calculated for airflow rate prediction (9%) and for temperature stratification (15%).

Fan and Ito [28] compared BES standalone simulations and coupling approach simulation for 3 different types of ventilation devices placed in typical offices in Japan. They underlined the ability of a coupling approach to obtain accurate results but also the need to continue the validation of such approach. Gowreesunker et al. [29] also used a similar approach to study the airflow inside an airport equipped with a displacement ventilation. The coupling process was slightly different: the thermal regulation of the room was driven by the BES simulation while CFD predicted the temperature distribution in the airport. This proves the numerous application of BES–CFD coupling. Finally, Gijón-Rivera et al. [30] applied a coupling approach with a two-equation turbulence model to a building office in Mexico. Three different configurations of glazed area were investigated with a BES standalone a coupled BES–CFD approach. Results showed that the coupled approach is always the most accurate.

In this frame, our study aims to demonstrate that a coupling approach can easily be applied to a complete building (two floors and 11 rooms), including an open space with high air temperature stratification. This approach accurately predicts the overheating phenomenon. This problem is becoming increasingly prevalent, and building designers do not have efficient tools to predict overheating. The developed tool is based on an external and dynamic coupling that addresses thermal and airflow aspects.

3. Case study description

The chosen case study is a typical Belgian house from the 1990s composed of two storeys with an entrance hall with a stairwell, a living room, a kitchen, a laundry and a professional office at the ground floor and a personal office open on the entrance hall and stairwell, four bedrooms and two bathrooms at the first floor. The two storeys are linked by the entrance hall, which also fulfils the role of personal office on the upper level (Fig. 1). This room is often the only office of this type of house. Therefore, ensuring good thermal conditions in the upper part of this room is important. For simplicity, we refer to this space as “the open space” in the rest of the article.