



Validation of computational fluid dynamics simulations for atria geometries

C.A. Rundle^a, M.F. Lightstone^{a,*}, P. Oosthuizen^b, P. Karava^c, E. Mouriki^d

^a Dept. of Mechanical Engineering, McMaster University, 1280 Main St. W, Hamilton, ON, Canada L8S 4L7

^b Dept. of Mechanical and Materials Engineering, Queen's University, 130 Stuart Street, Kingston, ON, Canada K7L 3N6

^c School of Civil Engineering and Division of Construction Engineering and Management, Purdue University, 550 Stadium Mall Dr., CIVL 1227, West Lafayette, IN 47907-2051, USA

^d Dept. of Building, Civil and Environmental Engineering, Concordia University, 1455 de Maisonneuve Blvd. West, Montreal, QC, Canada H3G 1M8

ARTICLE INFO

Article history:

Received 21 September 2010

Received in revised form

9 December 2010

Accepted 27 December 2010

Keywords:

CFD

Atria

Validation

Building simulation

Solar energy

ABSTRACT

Atria are becoming an increasingly common feature of new buildings. They are often included for their aesthetic appeal; however, their effect on building indoor environment can be significant. Building simulation tools have the potential to assist designers in enhancing energy efficiency by providing information on the temperature and velocity fields inside the atrium for specified geometries and ambient conditions. The unique nature of the physical phenomena that govern the complex flows in atria, however, are not usually considered in traditional building energy simulation programs. These physical phenomena include turbulent natural convection, radiative heat transfer and conjugate heat transfer. Computational fluid dynamics (CFD) has the potential for modeling fluid flow and heat transfer resulting from the phenomena; however, careful validation is required in order to establish the accuracy of predictions. This paper provides a systematic validation of a commercial CFD code against experimental measurements of the underlying physical phenomena. The validation culminates in the simulation of an existing atrium. This work indicates that CFD can be used to successfully simulate the heat transfer and fluid flow in atria geometries and provides recommendations regarding turbulence and radiative heat transfer modeling.

© 2011 Elsevier Ltd. All rights reserved.

1. Introduction

Atria are large enclosed spaces attached to a building with at least one transparent façade which typically has significant height. As such, the fluid flow and heat transfer within an atrium are governed by physical processes that include turbulent natural convection, radiation and conjugate heat transfer. These processes often result in thermal stratification within the atrium.

Properly designed atria have the potential to significantly reduce building energy consumption. In contrast, a poorly designed atrium can result in uncomfortable daytime temperatures and additional air conditioning loads [1]. By allowing daylight into the building, an atrium can reduce the amount of electricity used, however, if excessive daylight is permitted then glare can be a significant issue. Finally, natural ventilation in an atrium allows for removal of excess heat and can replace or supplement mechanical systems. Simulation tools provide designers with an opportunity to incorporate energy efficiency into the design of atria [2]. This paper provides an assessment of the use of computational fluid dynamics (CFD) for this purpose.

1.1. Traditional simulation tools

There are two types of simulation tools that are typically used to model heat transfer in atria and buildings: energy calculation method and building energy simulation (BES). Energy calculations are perhaps the simplest tool available. Empirical correlations predict energy usage and the general temperature in a room [3]. Energy calculations are typically unsuitable for use in atria geometries because their performance depends on factors those traditional empirical equations do not consider. There has, however, been some work to create empirical models that do take these factors into consideration, particularly stratification [4].

Building energy simulation programs such as EnergyPlus, TRNSYS and ESP-r have been shown to be both versatile and reliable [5]. Typically a number of zones are used to simulate the building; the TRNSYS 'Multi-Zone Building' component allows for heat transfer between multiple zones, each with a single node [6]. Building energy simulations differ from energy calculations in that they are typically more complex and allow interaction between different zones and more complicated interaction with other components such as HVAC systems [6]. A BES does not typically solve for the fluid flow within a building directly. The flow is usually determined by either experimental measurement or an additional program to generate estimates of flows between zones. COMIS is an

* Corresponding author. Tel.: +1 905 525 9140x27726; fax: +1 905 572 7944.
E-mail address: lightsm@mcmaster.ca (M.F. Lightstone).