



## Automatic extraction of Reduced order models from CFD simulations for building energy modeling

Mullen, D. T., Keane, M. M., Geron, M., & Monaghan, R. F. D. (2014). Automatic extraction of Reduced order models from CFD simulations for building energy modeling. Paper presented at Roomvent 2014, Sao Paulo, Brazil.

### Queen's University Belfast - Research Portal:

[Link to publication record in Queen's University Belfast Research Portal](#)

### Publisher rights

Copyright 2014 the authors.

### General rights

Copyright for the publications made accessible via the Queen's University Belfast Research Portal is retained by the author(s) and / or other copyright owners and it is a condition of accessing these publications that users recognise and abide by the legal requirements associated with these rights.

### Take down policy

The Research Portal is Queen's institutional repository that provides access to Queen's research output. Every effort has been made to ensure that content in the Research Portal does not infringe any person's rights, or applicable UK laws. If you discover content in the Research Portal that you believe breaches copyright or violates any law, please contact [openaccess@qub.ac.uk](mailto:openaccess@qub.ac.uk).

# Automatic extraction of Reduced-order models from CFD simulations for building energy modelling

D.T. Mullen<sup>1</sup>, M.M. Keane<sup>1,2</sup>, M. Geron<sup>3</sup>, R.F.D. Monaghan<sup>1,2</sup>

<sup>1</sup> National University of Ireland Galway – College of Engineering and Informatics – Ryan Institute

<sup>2</sup> National University of Ireland Galway – Informatics Research Unit for Sustainable Engineering (IRUSE)

<sup>3</sup> Queen's University Belfast – School of Mechanical and Aerospace Engineering

## Abstract

Thermal comfort is defined as “*that condition of mind which expresses satisfaction with the thermal environment*” [1] [2]. Field studies have been completed in order to establish the governing conditions for thermal comfort [3]. These studies showed that the internal climate of a room was the strongest factor in establishing thermal comfort. Direct manipulation of the internal climate is necessary to retain an acceptable level of thermal comfort. In order for Building Energy Management Systems (BEMS) strategies to be efficiently utilised it is necessary to have the ability to predict the effect that activating a heating/cooling source (radiators, windows and doors) will have on the room. The numerical modelling of the domain can be challenging due to necessity to capture temperature stratification and/or different heat sources (radiators, computers and human beings). Computational Fluid Dynamic (CFD) models are usually utilised for this function because they provide the level of details required. Although they provide the necessary level of accuracy these models tend to be highly computationally expensive especially when transient behaviour needs to be analysed. Consequently they cannot be integrated in BEMS.

This paper presents and describes validation of a CFD-ROM method for real-time simulations of building thermal performance. The CFD-ROM method involves the automatic extraction and solution of reduced order models (ROMs) from validated CFD simulations. The test case used in this work is a room of the Environmental Research Institute (ERI) Building at the University College Cork (UCC). ROMs have shown that they are sufficiently accurate with a total error of less than 1% and successfully retain a satisfactory representation of the phenomena modelled. The number of zones in a ROM defines the size and complexity of that ROM. It has been observed that ROMs with a higher number of zones produce more accurate results. As each ROM has a time to solution of less than 20 seconds they can be integrated into the BEMS of a building which opens the potential to real time physics based building energy modelling.

*Keywords: Computational modelling, reduced order models, natural ventilation, energy efficient building.*

## Introduction

One of the most pressing issues facing society today is the increasingly expensive, polluting and insecure sources of energy available. Energy efficiency will play an ever-increasing role in ensuring sustainable energy use. Up to 40% of the world's energy is consumed in buildings with 30-40% of that energy used in the maintenance of the thermal comfort in buildings [5]. The implementation of a well-designed BEMS can drastically improve the efficiency of building energy usage while maintaining a sufficient level of thermal comfort.

Validated CFD simulations [6] [7] have been successfully created and utilised to analyse thermal behaviour and user comfort of office space [8] [9] [10] in combination with wireless sensor networks (WSN) [11]. Although successful these CFD simulations are computationally expensive, especially if unsteady simulations are required or if they are to be used in conjunction with Building Energy Management Systems (BEMS) for operational strategies. Reduced order models (ROMs) have shown to have the potential for near real-time analysis [12] and can be a valid alternative for CFD models [13]. The main issue with the development of these models is that the decision on how to divide the domain into zones is left to the experience of the researcher. The researcher must also utilise empirical data or correlations to evaluate the parameters which characterise zone-to-zone energy or fluid exchange.

The focus of this research is to develop a CFD-ROM method that will facilitate the automatic extraction of ROMs from validated CFD models. As the data used to create the ROM will be extracted directly from a CFD file the need for the use of empirical data and the reliance on user decisions greatly reduced. As this method maintains the computational efficiency of ROMs it can be used in the optimization and operation of a BEMS. This paper will present the CFD-ROM method that has been developed over the course of the on-going research. The following sections will describe the CFD model used in the validation of the CFD-ROM method, provide a description how the CFD-ROM method functions and present the results generated to date.

## **Experimental data and CFD model**

### **Experimental setup**

The location for the study is a north facing office in the Environmental Research Institute (ERI) building at the University College Cork (UCC). It is shown in Figure 1 and has dimensions 5.2 m x 5.6 m x 2.9 m. It is heated by two Dimplex *SmartRad* fan convectors. Two electronically controlled windows are present and have a maximum aperture of 20°. Furniture comprises of five desks and chairs, and shelf units. No additional heating or momentum sources are present, for example computers and people. A set of 38 temperature sensors (TelosB) and 4 anemometers (HOBO) are used for experimental measurements. The anemometers are positioned in front of the opened windows to evaluate the air velocity component normal to the opening and on top of the fan assisted convectors. The temperature sensors are deployed at three different levels: 10 at ceiling level (2.65 m above floor), 20 at user level (1 m above floor) and 5 at floor level (0.2 m above floor) as shown in Figure 1. Three different boundary conditions were analysed and deemed as the most influential: the convectors, the windows and the door. The convectors are modelled at full capacity, and windows and door are open at angles of 20° and 90°, respectively.

### **CFD Model**

CFD models have been developed using the modelling software Phoenics to generate a database of test cases [14]. All simulations model turbulence using the steady state Reynolds Average Navier-Stokes (RANS), coupled with the Re-Normalisation Group (RNG)  $k-\epsilon$  approach, with air modelled as an incompressible ideal gas. The domain is discretized using a structured grid to give 1,572,165 cells (115x147x93). Constant temperature boundary conditions have been utilised for the ceiling and the floor, with all other objects considered adiabatic. The glass panels of the window have an overall heat transfer coefficient of  $U = 1 \text{ W/m}^2\text{K}$ . The door, when opened is modelled as a zone of constant pressure. The two convectors have been modelled using two surfaces: an inlet surface at the top of the convectors, where velocity and temperature were imposed and an outlet surface at the bottom of the convectors where an extraction mass flow rate was imposed. Opened windows are also modelled as inlet surfaces.

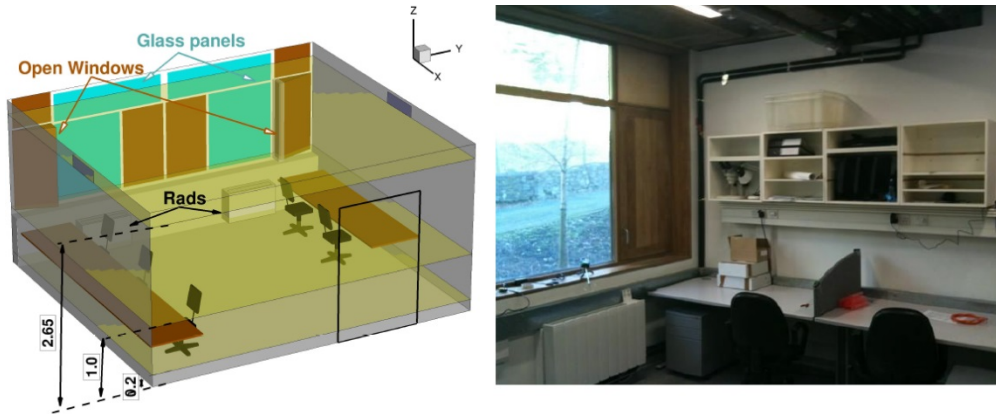


Figure 1: Numerical and physical domain

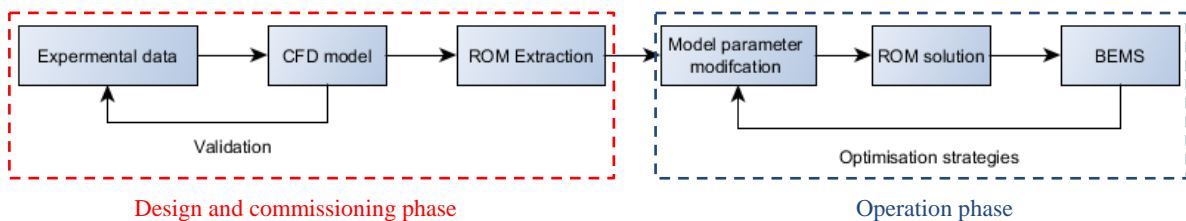


Figure 2: Integration of automatically extracted ROMs into the BEMS

## Method

As can be seen from Figure 2 the CFD-ROM method will function in conjunction with validated CFD models and allow the BEMS to rapidly model a variety of different model parameters for a given ROM. This rapid modelling ability will allow the BEMS to predict the thermal response of an NV action within the range of functionality of the ROM being used.

The method under development consists of a python [15] script that will automatically extract a ROM from a CFD solution file that can be used in conjunction with the ROM solver SINDA/FLUINT [16]. SINDA/FLUINT is a comprehensive finite-difference, lumped parameter tool for heat transfer design and fluid flow modelling. This python script consists of four sections of code shown in Figure 3 which are the pre-processor, zoning, zone interactions and boundary conditions and generating the input file for SINDA/FLUINT. These sections each complete a different function in the generation of a ROM. They are listed below and are described in the following sections.

1. Pre-processor – Any functions completed in the pre-processor only require to be completed once for a given CFD file.
2. Zoning – This section of the code uses user defined criteria to coarsen the CFD domain into a number of zones. These zones will be generated based on the range of the physical property of interest, connectivity between elements and minimum zone volume.
3. Zone interactions and boundary conditions – After the zoning section the code calculates the required mass and heat transfer interactions between connected zones. This section of code also calculates the boundary conditions to be applied to each zone.
4. Input file for SINDA/FLUINT - The last section in this method uses the information acquired in the previous steps to generate an input file that can be used with the ROM solver.

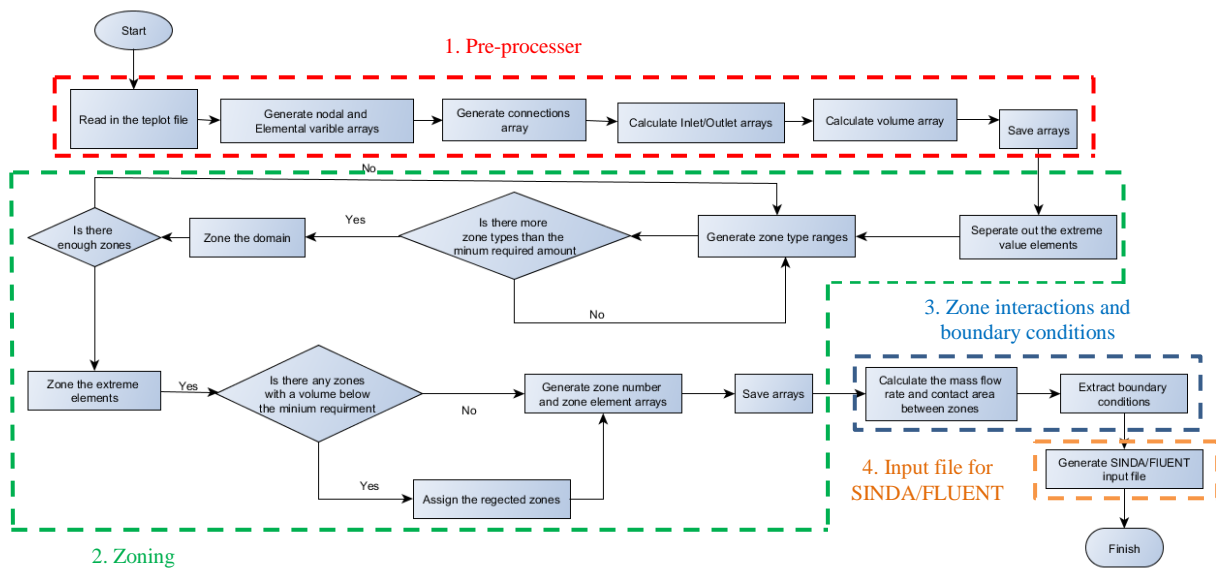


Figure 3: Flow chart of the ROM extraction method

## Pre-processor

The pre-processing section of the code is used to perform any operations that are not ROM specific and thus only have to be performed once per CFD file. Many of the functions performed in the pre-processor are complex operations which require substantial processing power, performing these operations in the pre-processor greatly improves the computational efficiency of the ROM extraction. The main functions performed in this section are creating the element connections array, defining inlet and outlet elements and calculating the volume of each element.

## Zoning

During this section the fluid domain is separated into the required number of zones as defined by the user. The maximum number of zones generated is user defined to represent the level of accuracy required for the application. A zone is created by initially taking 1 element that has not already assigned to a zone which also falls within the current zone-type range. A check is then performed to determine which of its connected elements are also within the zone-type range and the applicable elements are added to the zone. The process is then repeated for the newly added elements until no further elements that meet the required criteria can be found. Once this occurs the zone has been fully be defined.

An important factor to note with the method is that the zone generation is an iterative process. The zoning process starts by generating the number of zones associated with the minimum number of zone (two). The program then checks if the minimum number of zones has been met. If the number of zones produced is less than the minimum requirement the code repeats the zoning process for the next number of zone types (four) and repeats the check, if during this iteration the minimum number of zones has been met the code returns the defined zones and continues to the next section otherwise it will continue to the next number of zone-types.

## Zone interactions and boundary conditions

Once the zones have been defined the next section in the ROM extraction method is to calculate how the zones interact with each other and to define the boundary conditions applied to the model. The sole method for the transfer of energy between the zones is convection. Therefore the mass flow rates between zones need to be calculated. During the zoning section two arrays were generated, the zone element array and the zone number array.

The entries in each array list details of the elements that are connected to elements in different zones. These arrays are used in the calculation of the mass flow rates as they represent the elements that are on the boundary between zones.

In order to calculate the mass flow rates between zones the program iterates over each element in a zone-zone interface. Each iteration calculates the area of contact between each pair of elements, the normal component of the velocity to the area of contact and the average density. The mass flow rate is found using Equation 1 to Equation 3 [17].

$$V = [V_x, V_y, V_z] \quad \text{Equation 1}$$

$$V_n = V \cdot \vec{n} \quad \text{Equation 2}$$

$$\dot{m} = \rho A V_n \quad \text{Equation 3}$$

In these equations,  $V_x$ ,  $V_y$ , and  $V_z$  are the X, Y and Z components of velocity,  $\vec{n}$  is the normal vector to the plane,  $\rho$  is the density of the fluid,  $A$  is the area of the interface and  $V_n$  is the component of velocity normal to the plane. Once this operation is completed for each element in a zone-zone interface the sum of the mass flow rates was taken to be the total for that interface.

The boundary conditions that are required for the ROM solver include domain wall temperatures ( $T$ ), convection coefficient ( $h$ ) at the domain walls, inlet and outlet mass flow rates ( $\dot{m}$ ) and temperatures. Whether an element was on the boundary of the domain or not has been previously established geometrically in the pre-processor. The average wall temperature was found by averaging the temperature of all elements in a boundary present in each zone. In order to calculate the convection coefficient over the wall flat plate forced convection correlations were used; see Equation 4 to Equation 6.

$$Re_l = \frac{u_\infty L}{\nu} \quad \text{Equation 4}$$

$$Nu_L = 0.664(Re_L^{1/2})(Pr^{1/3}) \quad \text{Equation 5}$$

$$h = \frac{Nu_l k}{L} \quad \text{Equation 6}$$

In these equations,  $u_\infty$  is the fluid velocity,  $L$  is the length of an element,  $\nu$  is the kinematic viscosity,  $Pr$  is the Prandtl number ( $Pr = 0.71$ ) and  $k$  is the thermal conductivity of the fluid. In order to find the location of the inlets and outlets of the model a mass flow rate balance was completed on each element during the pre-processor. This data was used to calculate the mass flow rate entering or leaving each of the zones and the temperature associated with the flow.

### SINDA/FLUINT input file

The last section of code in the ROM extraction method is to create the input file for the ROM solver. This involves taking the information generated from the previous steps and writing it in the form required by SINDA/FLUINT.

### Results and Discussion

The following results were generated using the ROM extraction method described above. The data used in the extraction was taken from the previously validated CFD model [13] of the office space described above. All the data used to obtain the results was automatically extracted with the only input from the user being the minimum number of zones. The total error ( $E_t$ ) of the domain for each ROM is calculated using Equation 7 and Equation 8.

$$E_t = \sum_{i=1}^{N_z} \frac{V_i |E_i|}{V_t} \quad \text{Equation 7}$$

$$E_i = \frac{T_{CFD} - T_{ROM}}{T_{CFD} - T_{ref}} \quad \text{Equation 8}$$

In these equations,  $T_{CFD}$  is the average temperature of the CFD elements that make up a zone,  $T_{ROM}$  is the zone temperature predicted by SINDA/FLUINT,  $T_{ref}$  is a reference temperature set at 0 °C,  $V_i$  is the zone volume and  $V_t$  is the total domain volume.  $T_{ref}$  was set at 0 °C as it is generally the lowest outside temperature encountered in Ireland.

The contour plots shown in Figure 4 are generated using the visualisation software Tecplot [18]. Each contour plot shows and X,Y and Z planes of the domain with each plane originating from the centre of the domain. As can be seen from Figure 4 the ROM extraction method has been developed to the point where a ROM can be fully extracted and solved from a CFD file. When visually compared with the CFD contour plot in Figure 4 the ROMs retain a satisfactory representation of the thermal conditions in the office space. The pronounced stratification of temperature seen on the CFD result is adequately captured by a ROM consisting of 33 zones.

It can also be seen that as the number of zones in a ROM increases the accuracy of the solution. Figure 5 shows the impact of ROM size on total error ( $E_T$ ) and time to solution. Total error is seen to decrease with ROM size from 2.5% for 3 zones to 0.6% for 33 zones. This range of error was deemed an acceptable level for the required application. ROM size independence is achieved in the range 10-30 zones. Figure 5 also shows that time to solution increases with ROM size as is to be expected with the increased complexity of a ROM. The maximum ROM time to solution for this work is 20 seconds. When this is compared to the typical runtime for the CFD model from which the ROMs are extracted (5-8 hours), it is clear that the CFD-ROM method has potential for real-time application with BEMS.

## Conclusion and Future work

A method to automatically extract ROMs from a CFD solution has been presented and validated. The CFD-ROM method has been shown to have the ability to automatically extract ROMs from a converged and previously-validated CFD solution. The method retains a high level of accuracy, with a total error of 0.6% for a size-independent ROM consisting of 33 zones. Furthermore, it is highly computationally efficient, with a maximum time to solution of 20 seconds, compared to 5-8 hours for comparable CFD simulations. This time to solution enables ROMs to be used in the development and maintenance of a BEMS with the possibility for real-time building energy modelling. At present the work is focused on further developing the method described to increase its functionality and versatility. The next stage of method development will evaluate the ability of ROM to model conditions that are different from those from which they are generated. Validation of all results will be performed with comparable CFD solutions. Future development of this method will aim to incorporate and predict the response of the domain when users are present in the office space.

## Acknowledgments

This project is funded by the NUI Galway Millennium Fund for research.

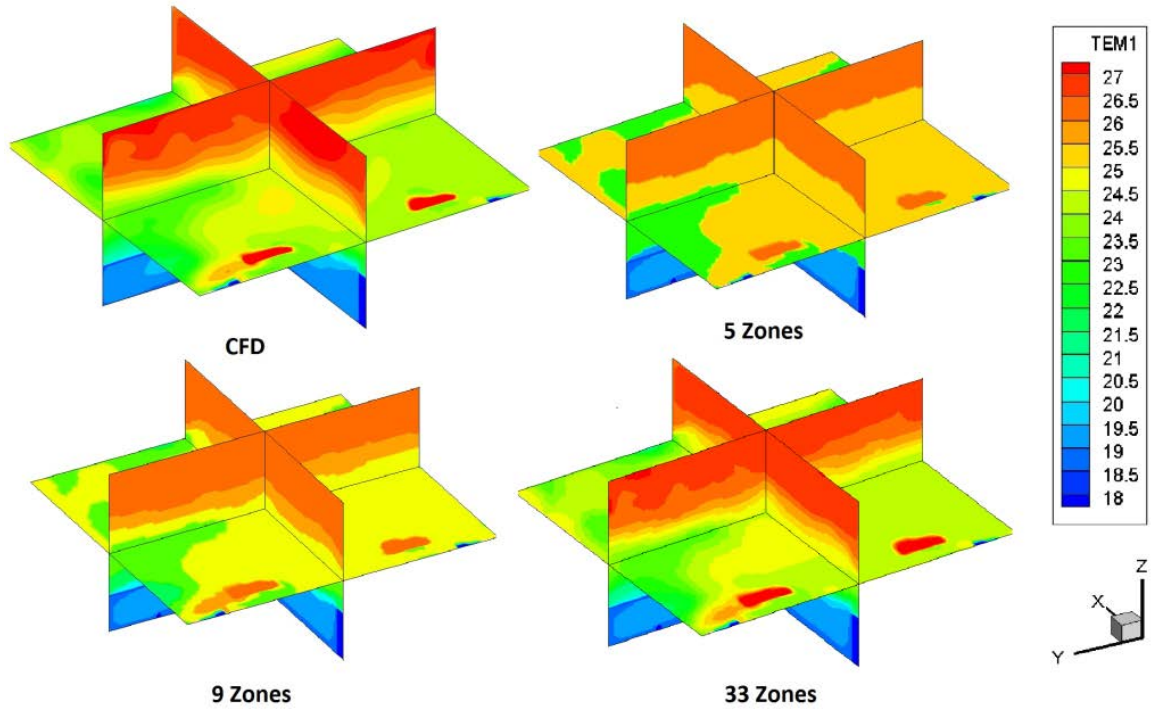


Figure 4: Temperature contour plots for ROMs of different sizes and the original CFD output

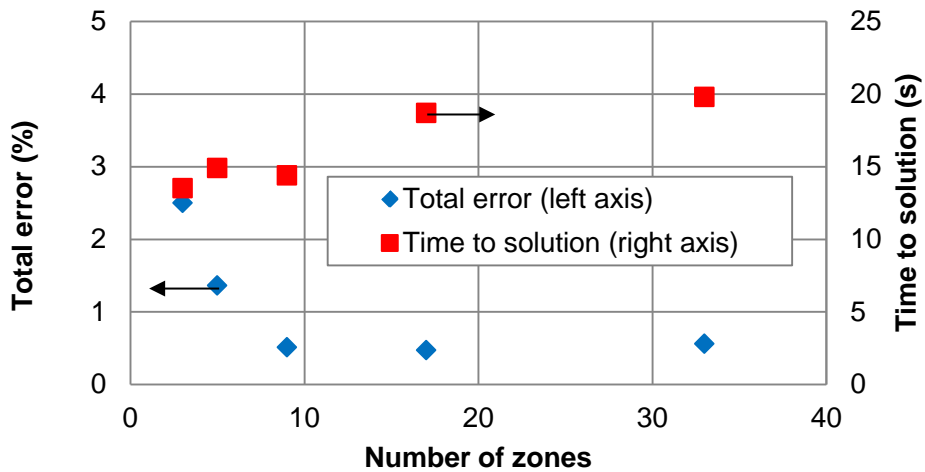


Figure 5: Impact of ROM size on total ROM error ( $E_T$ ) and time to solution

## Glossary

$E_i$	Weighted zone error (%)	$E_t$	Total error (%)
$L$	Characteristic length (m)	$\dot{m}$	Mass flow rate (kg/s)
$\vec{n}$	Normal vector to a plane	$Nu_L$	Nusselt number
$Pr$	Prandtl number	$Re_l$	Reynolds number
$T_{CFD}$	Average temperature of CFD elements that make up a zone ( $^{\circ}C$ or K)	$T_{REF}$	Reference temperature ( $^{\circ}C$ or K)
$T_{ROM}$	Zone temperature predicted by SINDA/FLUINT ( $^{\circ}C$ or K)	$u_{\infty}$	Fluid velocity (m/s)
$V_n$	Normal component of velocity (m/s)	$V_x, V_y, V_z$	X, Y and Z components of velocity respectively (m/s)
$V_i$	Zone volume ( $m^3$ )	$V_t$	Domain volume ( $m^3$ )
$\rho$	Density ( $kg/m^3$ )	$\nu$	Kinematic viscosity ( $m^2/s$ )



## References

- [1] Ashrae, "Thermal Environmental Conditions for Human Occupancy," *Ashrae Stand.*, vol. 2004, p. 30, 2004.
- [2] ISO, "ISO 7730: Ergonomics of the thermal environment ♦ Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria," *Management*, vol. 3, pp. 605–615, 2005.
- [3] J. F. Nicol and M. A. Humphreys, "Adaptive thermal comfort and sustainable thermal standards for buildings," *Energy Build.*, vol. 34, no. 6, pp. 563–572, Jul. 2002.
- [4] H. B. Rijal, P. Tuohy, M. A. Humphreys, J. F. Nicol, A. Samuel, and J. Clarke, "Using results from field surveys to predict the effect of open windows on thermal comfort and energy use in buildings," *Energy Build.*, vol. 39, no. 7, pp. 823–836, Jul. 2007.
- [5] "U.S. Energy Information Administration (EIA)." [Online]. Available: <http://www.eia.gov/>. [Accessed: 20-Jun-2014].
- [6] D. Amsallem, J. Cortial, and C. Farhat, "Towards Real-Time CFD-Based Aeroelastic Computations Using a Database of Reduced-Order Information," pp. 1–26.
- [7] H. B. Awbi, "Application of computational fluid dynamics in room ventilation," *Build. Environ.*, vol. 24, no. 1, pp. 73–84, Jan. 1989.
- [8] T. Catalina, J. Virgone, and F. Kuznik, "Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling," *Build. Environ.*, vol. 44, no. 8, pp. 1740–1750, Aug. 2009.
- [9] W.-H. Chiang, C.-Y. Wang, and J.-S. Huang, "Evaluation of cooling ceiling and mechanical ventilation systems on thermal comfort using CFD study in an office for subtropical region," *Build. Environ.*, vol. 48, pp. 113–127, Feb. 2012.
- [10] M. Hajdukiewicz, M. Geron, and M. M. Keane, "Formal calibration methodology for CFD models of naturally ventilated indoor environments," *Build. Environ.*, vol. 59, pp. 290–302, Jan. 2013.
- [11] X. Wang, X. Wang, G. Xing, J. Chen, C.-X. Lin, and Y. Chen, "Towards Optimal Sensor Placement for Hot Server Detection in Data Centers," *2011 31st Int. Conf. Distrib. Comput. Syst.*, pp. 899–908, Jun. 2011.
- [12] C. Farhat and D. Amsallem, "REDUCED-ORDER MODELING , DIFFERENTIAL GEOMETRY AND PHYSICS-BASED NEAR-REAL-TIME PREDICTIONS," vol. 195, no. Eccomas, p. 5742, 2008.
- [13] M. M. Keane, "REDUCED ORDER MODELLING OF THE THERMAL BEHAVIOUR OF AN OFFICE SPACE," no. 1.
- [14] "CHAM Your Gateway to CFD Success." [Online]. Available: <http://www.cham.co.uk/>. [Accessed: 20-Jun-2014].
- [15] "Welcome to Python.org." [Online]. Available: <https://www.python.org/>. [Accessed: 20-Jun-2014].
- [16] "Heat Transfer and Fluid Flow Design and Analysis, SINDA/FLUINT." [Online]. Available: <http://www.crtech.com/sinda.html>. [Accessed: 20-Jun-2014].
- [17] M. J. Moran, H. N. Shapiro, B. R. Munson, D. P. Dewitt, J. Wiley, K. Hepburn, H. Grossman, and L. Fleming, *Introduction to Thermal Systems Engineering : and Heat Transfer* . .
- [18] "Tecplot CFD post processing tools visualize & analyze data results." [Online]. Available: <http://www.tecplot.com/>. [Accessed: 20-Jun-2014].