Proceedings Paper:
de Boer, G, Delbosc, N, Johns, A et al. (2016) Three computational methods for analysing thermal airflow distributions in the cooling of data centers. In: Massarotti, N, Nithiarasu, P and Joshi, Y, (eds.) 4th International Conference on Computational Methods for Thermal Problems (ThermaComp 2016). ThermaComp 2016, 06-08 Jul 2016, Atlanta, USA. Giannini Editore. ISBN: 978-88-7431-828-5. ISSN: 2305-5995. EISSN: 2305-6924.

Fourth International Conference on Computational Methods for Thermal Problems
THERMACOMP2016, July 6-8, 2016, Georgia Tech, Atlanta, USA
N. Massarotti, P. Nithiarasu and Y. Joshi (Eds.)

Three computational methods for analysing thermal airflow distributions in the cooling of data centers.

Greg de Boer
Department of Aeronautics, Imperial College London, London, SW7 2AZ, UK,
g.de-boer@imperial.ac.uk

Nicolas Delbosc, Adam Johns, Daniel Burdett, Jon Summers
School of Mechanical Engineering, University of Leeds, LS2 9JT, UK,
{mnnd;A.Johns;mn09d3b;J.L.Summers}@leeds.ac.uk

Morgan Tatchell-Evans
School of Chemical and Process Engineering, University of Leeds, LS2 9JT, UK,
pm08mrte@leeds.ac.uk

ABSTRACT

This paper develops computational models to study thermal airflow distribution when cooling a data center. The thermal airflow distribution study employs three computational approaches, namely finite element, finite volume and lattice Boltzmann methods, which are respectively implemented via commercial Multiphysics software, opensource CFD code and home grown GPU based code. The results focus on comparison of the three methods, all of which include models for turbulence, when applied to 2 rows of racks with cool air supplied by a plenum. Advantages and disadvantages of the three computational methods are addressed in terms of application to thermal management of data centers.

Key Words: *Data center cooling, Lattice Boltzmann, Finite Elements, Finite Volume.*

1. INTRODUCTION

A data center hosts a large number of datacom systems, usually laid out in 2m tall racks. The expansive layout of datacom systems in a data center results in a distributed and dynamic generation of heat that needs to be transported by the air and rejected to the outside environment using heat exchangers and chillers. The global growth in the number and size of data centers means that they account for more than 1.5% of global electricity consumption and approximately 45% of this energy is used to remove the heat from the datacom systems [1]. Improvements in energy efficiency are now becoming important to reduce costs and environmental impact based on good practice [2].

Managing air distributions in data centers is one way to help reduce inefficiencies, and this can be aided by the application of Computational Fluid Dynamics (CFD) models, in which it is possible to predict the presence of hot spots for both new and upgraded data center layouts [3]. This in turn enables the air distribution to be optimised to minimise energy consumption whilst ensuring a suitable thermal environment is provided. CFD has been used successfully to investigate the impact of floor grilles [4], optimise placement of IT equipment [5] and study the effect of aisle containment [6]. The major challenges in producing accurate models are the multiple length-scales [3], from the chip to the room level, and the various modes of thermal transport and flow regimes [7].

CFD is an increasingly being applied to analyse air distributions in data centers as it offers a much greater resolution of data compared to most experimental approaches. However, there exist a range of modelling methods and software packages available for this task, each with individual advantages and disadvantages. This paper analyses the air flow through an simplified datacentre for several different modelling strategies to identify the trade-offs between methodologies. The methods applied in this work are; i) a developed lattice Boltzmann method (LBM) GPU based program, ii) an opensource CFD finite volume method (FVM) based package OpenFOAM[1], and the commercial CFD finite element method (FEM) based software Comsol[2]. Previous studies have demonstrated through experimental validation that CFD models can accurately predict data centre air flows and thermal environments [4-7]; this work aims to demonstrate the range of numerical methods available which produce similar results to tried and tested methods.

2. MAIN BODY

The geometry of the data center is designed to allow for a simple implementation in each of the numerical methods considered while keeping most of the characteristics of a typical data center.

The data center floor space has a surface area of 28.8m^2, comprised of ten racks that are organised in two rows. The datacom units in the racks have the fronts facing each other making up the cold aisle in the center. Each rack is separated into two datacom units along the height, so that the digital workload between the top half and bottom half of the rack can differ. This is a simplification as a rack would normally be composed of many more datacom systems. A computer room air conditioner (CRAC) unit is placed in line with the cold aisle at one end of the data center, and feeds cool air into the plenum under the floor as depicted in Figure 1.

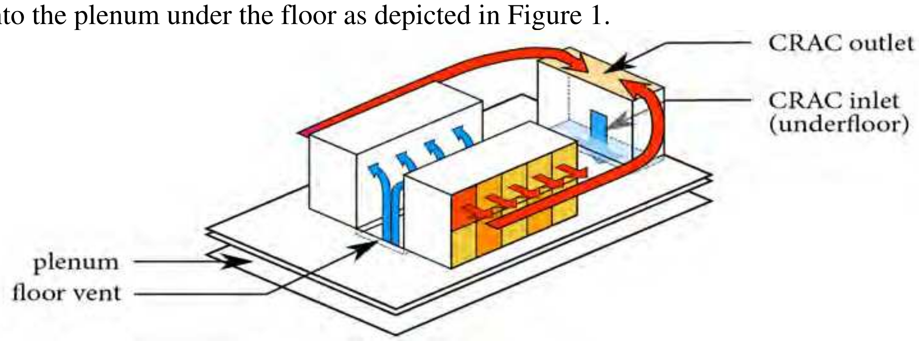


FIGURE 1. Schematic of the data center racks and the air flows.

The cold air is supplied to the room at a constant temperature of 289K and a constant flow rate of 1.134 m^3/s through the bottom of the CRAC unit, which coincides with the top of the plenum as indicated in the schematic of Figure 1. Cold air travels in the plenum and then enters the room through the floor vents, modelled here as open holes and located equidistant between the two rows of racks. This is a simplified model, as in reality the plenum is supported by an array of legs to support the floor of the data center, but also there will be cables and/or pipes that would noticeably affect the distribution of velocities arriving at the floor vent. Floor grilles are used and contain detailed features that straighten the flow and affect its momentum, and they can sometimes be oriented to change the angle of the flow to provide the cool air where it is required.

The air enters each datacom unit through their front-faces and exits from their back-faces after an increase in momentum and a rise in temperature. The volume flow rate through each datacom system, $Q_{datacom}$, as well as the temperature rise, $\Delta T_{datacom}$, are dependent on the datacom unit's

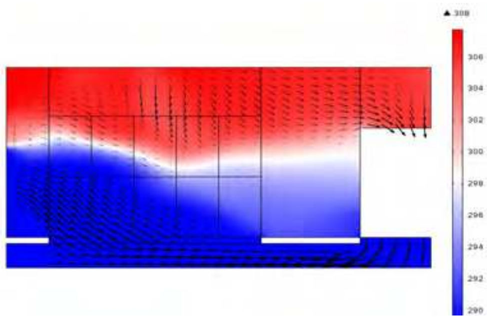
[1] www.openfoam.org

[2] www.comsol.com

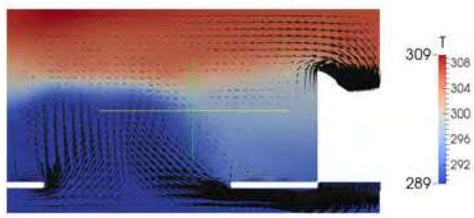
power draw, $P_{datacom}$, since $P_{datacom} = \rho c_p Q_{datacom} \Delta T_{datacom}$. Therefore knowing the datacom unit's power demand and air flowrate yields a value of temperature difference between the front and the back of the unit. The inlet to each datacom system is an outlet of the data center domain and the outlet from each datacom system is an inlet to the data center domain. The boundary conditions on each datacom unit for both air flowrate and temperature are linked as described by Summers et al [8]. All other boundary conditions are set as no-slip for the flow and insulating for the temperature field. The air flowrates are such that a turbulence model is required in the simulations. For the FEM and FVM the flows are solved under steady state conditions and the Reynolds Averaged Navier-Stokes (RANS) is employed. Whereas the LBM is inherently transient and the turbulence is accounted for by the adoption of a Large Eddy Simulation (LES) approach.

3. RESULTS

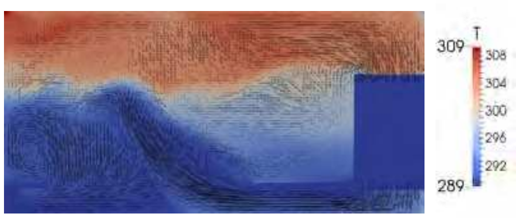
To enable the comparison of the results between the three computational methods, the LBM results are required to be averaged over time, since these results are transient. This is achieved by averaging for 3 minutes (of physical time), starting after 2 minutes from an initial state of zero air velocity and a constant temperature everywhere in the room of 293K. Figure 2 demonstrates the temperature and velocity fields for all three methods: the temperature at the back of the datacom systems shows similar distributions, the cold air coming up through the floor vents rises to approximately 1.6m in height. The full paper will show that the hot air is projected out of the back of the datacom systems with similar angles and strengths, all of which indicate similar treatment of the buoyancy effects in each of the three computational approaches.



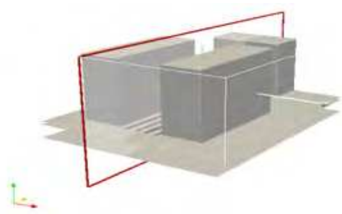
(a) FEM results (Comsol).



(b) FVM results (OpenFOAM)



(b) LBM results



(d) Geometrical position of cross section.

FIGURE 2. Velocity and temperature fields at the cross section depicted in (d) for the three methods.

A trend is confirmed by comparing the average temperature at each of the datacom inlets across the three computational approaches. For some datacom inlets all three methods give the same temperature within 1K, but for inlets of some datacom systems towards the center of the aisle, there is up to a 4K difference between the hottest and coldest prediction. Overall, the predicted temperature at the datacom inlets agree well, but the LBM appears to give consistently a slightly higher temperature (by about 1 degree) than the other methods, this may be a side effect of using an LES turbulence model as opposed to the RANS model used for the FEM and FVM but is likely a result of the choice of the initial temperature. Each computational approach has its own performance constraints, levels of accuracy, stability and convergence conditions. The FEM solution required 403825 elements to meet a convergence criterion of 10^{-5} and took 52814 seconds to compute, the FVM solution required 1303093 cells to meet a convergence criterion of 10^{-5} and took 19996 seconds to compute. The LBM solution was computed on a regular grid composed of 753984 lattice points with a time-step of $5x10^{-3}$s and took 300 seconds to compute (i.e. real-time). While being the fastest method for the given resolution and time-step, the LBM displays spurious thermal fluctuations that appear as a chequerboard pattern and are due to numerical instabilities. This work is also able to highlight the computational performance of both the FVM and FEM as well as identifying further aspects of the modelling approach and assumptions that position these computational techniques as valuable tools for analysing thermal air management of data centers. The full paper discusses the trade-offs of each of the computational methods, time required to develop the model (or technical knowledge required to apply the software) and additional challenges of modelling data center airflows.

4. CONCLUSIONS

The simulation results of thermal air flows in data centers obtained with three computational methods demonstrate good agreement in terms of the overall flow structures and the average temperatures at each datacom inlet. The level of accuracy achieved by the FVM, the FEM and the LBM are similar. There are clear advantages of using the LBM in respect to computational performance and applicability for transient flows, which also offers the potential to inform on the dynamic nature of real thermal air management of data centers.

REFERENCES

[1] Koomey, J., 2011. Growth in data center electricity use 2005 to 2010. *A report by Analytical Press, completed at the request of The New York Times*, p.9.

[2] Liam Newcombe, Mark Acton, John Booth, Sophia Flucker, Paul Latham, Steve Strutt, and Robert Tozer. 2012 Best Practices for the EU Code of Conduct on Data Centres. European Commission, Joint Research Centre, 2012.

[3] Yogendra Joshi and Pramod Kumar. Introduction to Data Center Energy Flow and Thermal Management. In Yogendra Joshi and Pramod Kumar, editors, Energy Efficient Thermal Management of Data Centers, pages 1–38. Springer US, 2012.

[4] Emad Samadiani, Jeffrey Rambo, and Yogendra Joshi. Numerical modeling of perforated tile flow distribution in a raised-floor data center. Journal of Electronic Packaging, 132(2):021002, 2010.

[5] Jayantha Siriwardana, Saman K Halgamuge, Thomas Scherer, and Wolfgang Schott. Minimizing the thermal impact of computing equipment upgrades in data centers. Energy and Buildings, 50:81–92, 2012.

[6] Kyosung Choo, Renan Manozzo Galante, and Michael M. Ohadi. Energy consumption analysis of a medium-size primary data center in an academic campus. Energy and Buildings, 76(0):414 – 421, 2014.

[7] S.K. Shrivastava, B.G. Sammakia, R. Schmidt, M. Iyengar, and J.W. VanGilder. Experimental-Numerical Comparison for a High-Density Data Center: Hot Spot Heat Fluxes in Excess of 500 W/FT2. In Thermal and Thermomechanical Phenomena in Electronics Systems, *2006* ITHERM '*06*. The Tenth Intersociety Conference on, pages 402–411, May 2006.

[8] Summers, J., Kapur, N. and Thompson, H., 2013, March. Design of data centre rack arrangements using open source software. In *Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM), 2013 29th Annual IEEE* (pp. 45-51). IEEE.