Numerical Simulations of Different Duct Shape for Centralized Air-Cooling System

Raj Patel[1], Dr. D. S. Rawat[2]
[1]Research Scholar, Department of Mechanical Engineering
[2]Assistant Professor, Department of Mechanical Engineering
[1,2]Jabalpur Engineering College Jabalpur (M P), India

Abstract—**Temperature control always plays a crucial role in industrial processes. Many processes depend on a precise temperature range, and if that temperature is not maintained, the entire process is affected. This ultimately compromises the quality of the final product. Some machines generate significant heat, and therefore, it is necessary to reduce their temperature with some external arrangement because continuous heating can lead to machine failure or breakdown. Air conditioning in homes is another application of temperature control that has become very popular these days. With a lot of pollution being generated through various sources, the normal temperature range is increasing day by day. The need for cool air and cool water in homes and many other applications is ever-present. To improve the application of water-cooled air conditioning systems in the domestic sector, it is necessary to develop an estimation model for energy efficiency analysis. This paper focuses on developing a simulation model to estimate the operational efficiency and energy consumption for the application of water-cooled air conditioning systems.**

Index Terms—**Thermal efficiency, Cooling efficiency, CFD modelling.**

I. INTRODUCTION

Summer temperatures vary annually. In India, during the summer season, the average temperature is 45°C. Various appliances such as air conditioners, coolers, and fans are used to maintain comfortable conditions during the summer. The cost of such appliances is very high, making them unaffordable for many consumers. These appliances also consume a lot of electricity, resulting in high electricity bills. Also, everyone needs cool air and cold water to drink during the summer. So, to be comfortable during this season, we need cool air and chilled water. To meet these needs, we would typically need two separate appliances, one for cooling the air and another for cooling the water. Furthermore, the installation of these appliances requires considerable space. Innovation and change are inherent in engineering. That is why we have introduced the project "Design and Development of a Combined Device for Air and Water Cooling". The main objective of our project is to provide the two facilities of cool air and chilled water in a single unit and to supply this unit at an affordable price to the common man. This unit will be suitable for homes and other commercial places such as halls, corporate offices, and industries. Therefore, geometrical modifications such as installing air guides, separating hot and cold zones, and preventing air leakage improve the cooling efficiency of data centres. Due to rapid economic growth and urbanization over the past two decades, energy consumption, particularly electricity consumption, in high-rise residential buildings has increased significantly. According to previous research on household energy statistics, from 1990 to 2000, total energy consumption for air conditioning increased by 80%, while the population growth rate was only 23%. In a typical residential building in Hong Kong, air conditioning accounts for 25% of the energy consumption. These statistics clearly show that energy-saving measures that can reduce energy consumption for air conditioning are highly valuable. Hong Kong is a subtropical city with long, hot, and humid summers. Air conditioning is widely adopted in the residential sector. These air conditioning units are mostly air-cooled window or split units. This is common in Hong Kong and other parts of the world. The use of air-cooled units in Hong Kong is particularly inefficient because the energy required by the air conditioner increases at a steady rate as the amount of condenser refrigerant entering

the air conditioner increases. Another factor that further detracts from the energy efficiency of air-cooled units used in Hong Kong is the common practice of placing air conditioners in internal spaces. This design is unique to Hong Kong. The internal spaces are designed with individual apartments extending from the centre of the building outwards, creating more exterior walls and windows. These spaces can be used to meet various needs of residents in residential buildings

II. LITERATURE REVIEW

Dozer and Saleem [1] recommended a cold/hot aisle containment, return air plenum, vacating all unused server spaces, and installing gap panels between racks to reduce hot air recirculation. Neiman et al. [2] tested hot/cold aisle containment because it is more effective than controlling the cold aisle. Babde et al. [3] and Alissa et al. [4] investigated the effect of underfloor obstructions on the thermal distribution and thermal performance of a data centre. Cold aisle containment was achieved through strategically placed cooling pipes, and the entry shell concept was introduced to mitigate the bypass effect observed at the cold aisle entrance. Ham and Jeong [5] analysed the effect of cold aisle containment on the thermal management and energy consumption of cooling equipment in a data canter. The containment energy efficiency can be increased by incorporating an economizer in the retrofit design. Zheng et al. [6] studied the thermal environment and energy consumption of a data center located in Shanxi Province before and after retrofitting. The total cooling capacity of 600 kW was insufficient to meet the heat dissipation requirements of fourteen racks with high heat generation capacity, and the outlet temperature of the racks was found to be above 30 °C. Therefore, four rack cabinets were reinforced with an external water-cooling system, and the power consumption of the cooling system decreased by 18% during the summer months. Wang et al. [7] investigated design options using CFD methods to provide uniform airflow distribution in a newly constructed data center. It was determined that hot/cold aisle containment provides uniform airflow distribution with high cooling efficiency. Wang et al. [8] performed numerical simulations to eliminate bypass and recirculation in data centres in terms of performance indicators and showed that the return

temperature index (RTI) and return heat index (RHI) can be improved through data center reconfiguration. Meng et al. [9] analyzed the thermal environment and flow field of a small-scale data center located in China through CFD simulation. Some equipment was rearranged to eliminate the problems of uneven temperature distribution and chaotic airflow distribution. Meng et al. suggested that sealing gaps, opening perforated tiles under the rack, and installing an air guide at the air conditioner outlet would improve the overall performance of the data center. Implementing such design recommendations resulted in an increase in RHI from 0.918 to 0.93 and an increase in RTI from 0.222 to 0.342. Experimental and numerical studies were conducted to investigate the effects of plenum height, perforation percentage, and the location of the CRAC unit [10]. Tian et al. [11] proposed a novel mathematical model for multilevel thermal management based on entropy theory. Turkmen et al. [12] reported that geometric characteristics, the location of IT equipment, and the interaction of hot and cold air can degrade the performance of a small data centre. Consequently, the estimated Power Usage Effectiveness (PUE) value decreased by 28.2% due to the rearrangement of certain characteristics. Oro et al. [13] proposed multiple retrofits to reduce bypass and negative pressure in a data centre in Barcelona, which degrades cooling performance. In the proposed design, activating the cold aisle by switching off two CRAC units, increasing the supply air temperature, and reducing the airflow rate resulted in significant performance improvements and economic savings. During the summer, we need both cool air and cool water. Therefore, we require two separate units: an air cooler and a water chiller or refrigerator. This takes up more space and is also more expensive. There is a need to develop a single machine that can cool both water and air. This would help save space and reduce electricity consumption, making it very efficient. It would be a cost-effective unit not only for homes but also for offices.

III. WAC MODEL

The simulation models adopted to emulate WAC performance are similar to the AAC models, but the energy consumption varies with the cooling load, indoor temperature, and the entering condenser water

temperature. The electrical input for the WAC takes into account the condenser water system costs, i.e., the circulation pump and cooling tower fan costs (WF). Detailed performance models, including user behaviour during air conditioning operation and internal load intensity obtained from building surveys, provide the necessary information to create a daily cooling load profile of the house to simulate energy consumption for air conditioning.

IV. RESULTS AND DISCUSSIONS

4.1.1 Simulation of a duct based on a circular cross-sectional area

Figure 1. shows the temperature distribution within the cylindrical pipe. The boundary exhibits a nearly uniform temperature field across the entire domain, indicating very small temperature gradients. Such uniformity suggests adiabatic wall conditions or minimal heat transfer effects under the recommended operating conditions. The absence of localized hot or cold regions confirms the stable integration of the energy equation and consistent thermal boundary conditions. The uniform temperature distribution implies that thermal effects do not significantly influence the fluid properties or turbulent behaviour in the current simulation. This result allows the flow to be considered thermally stable, focusing the analysis on the hydrodynamic and turbulent characteristics.
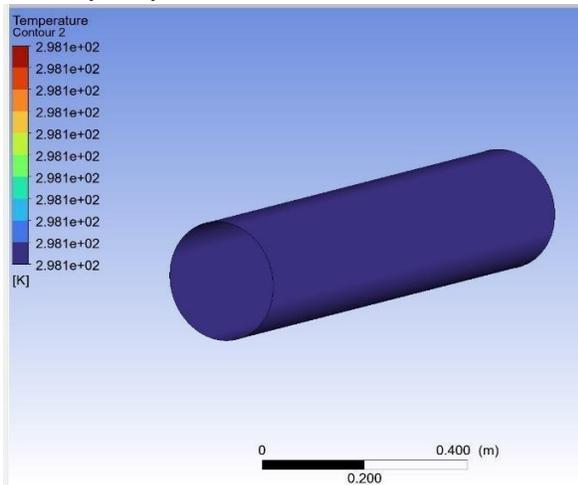

Figure 1. Temperature Contour Distribution

Figure 2. shows a gradual decrease in pressure along the flow direction within the cylindrical pipe. The highest pressure occurs near the inlet, while the lowest pressure is observed at the outlet, reflecting frictional losses and turbulent dissipation. The smooth and continuous pressure gradient indicates steady-state flow conditions and accurate pressure-velocity coupling. The uniformity of pressure across the cross-section confirms the absence of flow separation or asymmetrical behaviour. This pressure distribution is crucial for evaluating pressure drop and flow resistance. The results demonstrate the typical physically consistent behaviour of fully developed turbulent pipe flow.
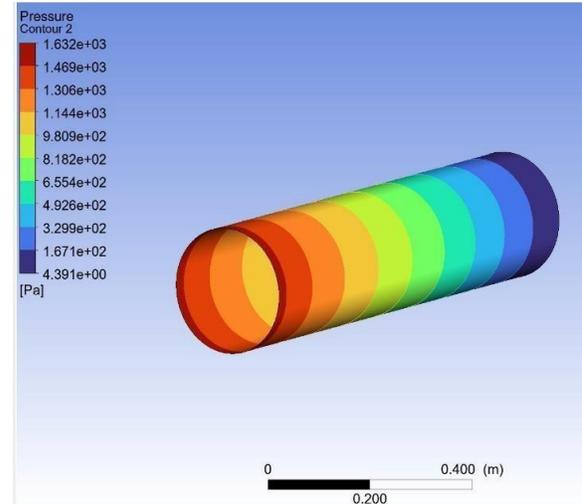

Figure 2. Static Pressure Contour Distribution

4.1.2 Simulation of a duct based on a rectangular cross-sectional area

Figure 3. plot illustrates the temperature distribution within the pipe. The domain exhibits a nearly perfectly uniform temperature field, indicating very low thermal gradients throughout the flow. Such behaviour suggests either adiabatic wall conditions or minimal heat transfer effects under the given operating conditions. The uniform colouring confirms the successful integration of the energy equation and constant thermal boundary conditions. The absence of localized hot spots or temperature stratification indicates that thermal effects do not significantly influence the fluid properties or turbulent behaviour in this simulation. This result justifies considering the flow as thermally stable and allows focusing on the hydrodynamic and turbulent characteristics.
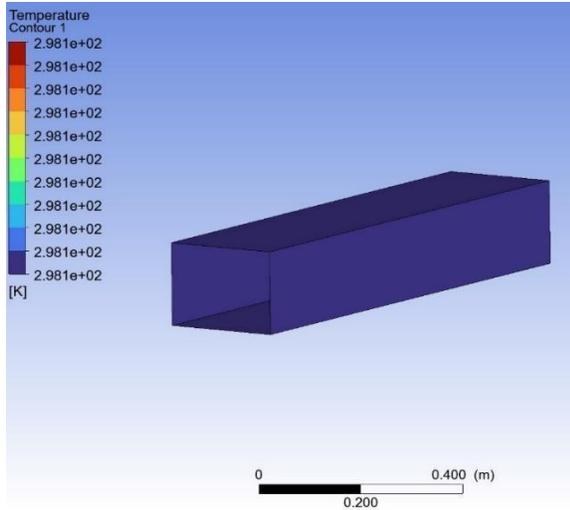
Figure 3. Temperature Contour Distribution

In Figure 4, This constant pressure contour line shows a gradual decrease in pressure along the flow direction. The highest pressure occurs at the inlet, while the lowest pressure is observed near the outlet, reflecting energy dissipation induced by frictional losses and turbulence. The smooth gradient indicates steady flow development and accurate pressure velocity coupling. The uniformity of pressure across the cross-section confirms the absence of flow separation or asymmetric behaviour. This pressure distribution is essential for estimating pressure drop, pump power requirements, and flow resistance. The results are consistent with theoretical expectations for turbulent flow in rectangular channels.
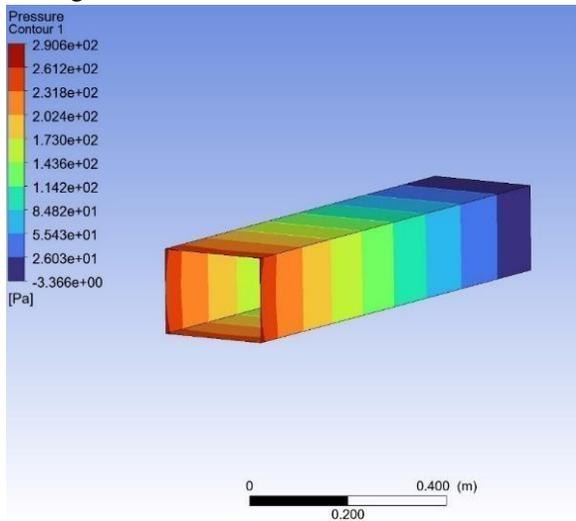

Figure 4. Static Pressure Contour Distribution

4.1.3 Simulation of a duct based on a square cross-sectional area

In Figure 5, This contour plot shows the temperature distribution across the computational domain. The contour lines reveal a nearly uniform temperature field, indicating very small temperature gradients within the pipe. Such uniformity suggests adiabatic wall conditions or minimal heat transfer effects under the given flow conditions. The absence of localized hot or cold spots confirms stable energy equation convergence and consistent boundary condition implementation. This temperature uniformity is common in incompressible or low heat-flux flow simulations where thermal effects are secondary. The smooth and uniform colour distribution across the domain indicates numerical stability and the absence of artificial thermal diffusion. These results confirm that temperature variations do not significantly affect the fluid properties or turbulence behaviour in the current analysis, allowing the focus to remain on the flow and turbulence characteristics.
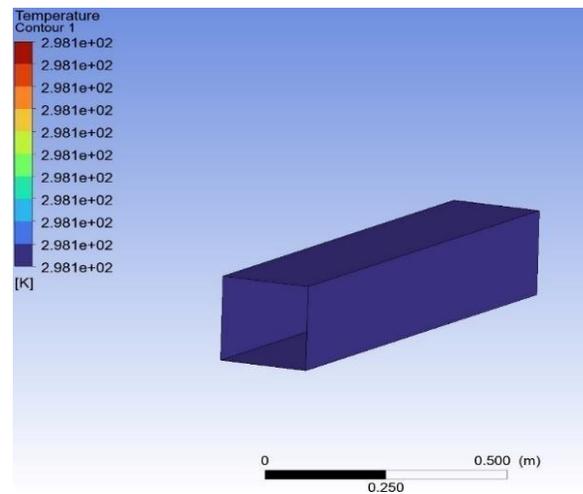

Figure 5. Temperature Contour Distribution

In Figure 6, This boundary layer pipe illustrates a steady pressure distribution along its length. A clear pressure gradient is observed from the inlet to the outlet, confirming the pressure driven flow behaviour. The higher pressure at the inlet gradually decreases downstream due to friction losses and turbulent dissipation. The nearly linear pressure drop indicates fully developed turbulent flow conditions. Uniform pressure profiles across the cross-sections suggest negligible secondary flow effects. This pressure distribution is crucial for estimating flow resistance and energy loss within the pipe. The smooth pressure

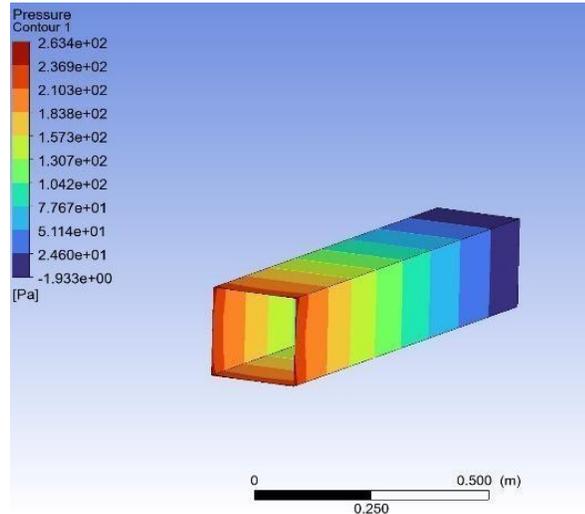variation confirms numerical stability and validates the accuracy of the momentum and turbulence model.

Figure 6. Pressure Distribution Contour

4.2.1 Convergence analysis of a duct based on a circular cross-sectional area

Figure 7 illustrates the convergence behaviour of the RMS residuals for the mass continuity and momentum equations across three velocity components. In the initial stages, residual spikes are evident due to initialization and evolving flow patterns. As the solver progresses, all residuals exhibit a systematic downward trend.
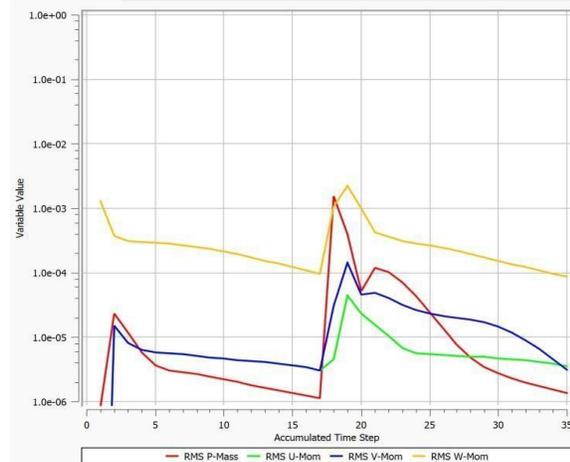

Figure 7. RMS Continuity and Momentum Residual Convergence

4.2.2 Convergence analysis of a duct based on a rectangular cross-sectional area

Figure 8 illustrates the convergence behaviour of the RMS residuals for the mass continuity and momentum

equations in three spatial directions (U-, V-, and W-momentum) plotted against the time steps collected on a logarithmic scale. The residuals initially exhibit sharp spikes, corresponding to changes in flow-field conditions during solver initialization and transient advancement.
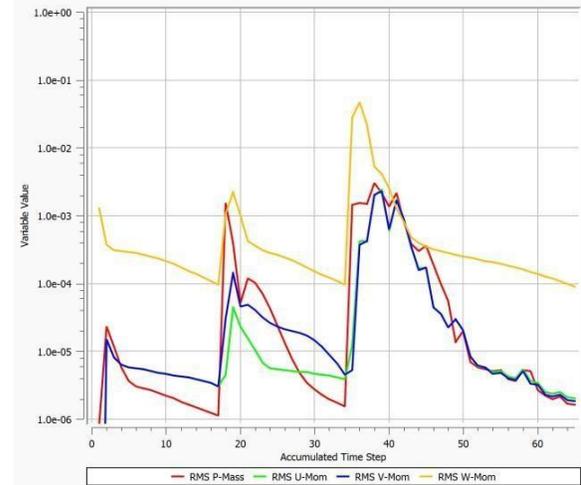

Figure 8. RMS Continuity and Momentum Residual Convergence History

4.2.3 Convergence analysis of a duct based on a square cross-sectional area

Figure 9. depicts the RMS residual convergence histories of the mass continuity and momentum equations along the x-, y-, and z- directions. Initially, residual spikes are observed due to solver initialization and flow field development. As the solution progresses, all residuals exhibit a consistent downward trend, converging to less than 10^{-5} for momentum and close to 10^{-6} for continuity
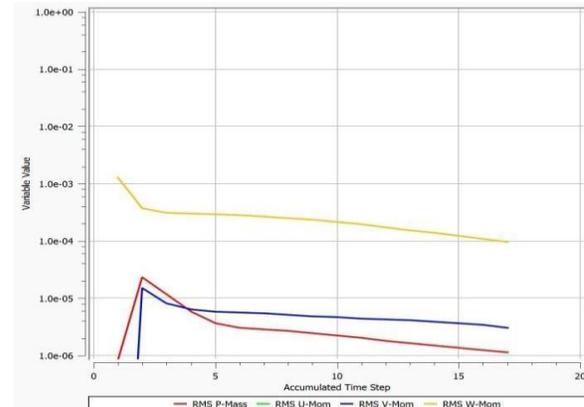

Figure 9. RMS Continuity and Momentum Residual Convergence

V. DUCTWORK SYSTEM DESIGN

The duct's cross-sectional shape can be circular or rectangular, with equivalent flow friction, meaning the cross-sectional area of the flow is the same. The equation relating the diameter D of a circular duct to the side lengths of a rectangular duct is as follows:

$$D = \frac{1.3a(b/a)^{\frac{5}{8}}}{(1+b/a)^{\frac{1}{4}}} \qquad (1)$$

Equation (1) can be expressed as

$$D = \frac{1.3a(ab)^{\frac{5}{8}}}{(1+b)^{\frac{1}{4}}} \qquad (2)$$

Then k, a perimeter ratio of the round duct to the rectangular duct, is

$$k = \frac{\pi D}{2\,(a+b)} = \frac{1.3\pi(ab)^{\frac{5}{8}}}{2\,(a+b)(a+b)^{\frac{1}{4}}} \qquad (3)$$

Equation (3) becomes after rearrangement

$$0.65\,\pi \sqrt[3]{\frac{R^5}{(1+R)^{10}}} \qquad (4)$$

where R = a/b = long-side length/short-side length

Table 1 shows the relationships between perimeter and area. The perimeter ratio k results, along with the aspect ratio R and description, are presented in Table 2. It can be observed that a smaller perimeter is required for a circular duct compared to a rectangular duct. The larger the value of R, the smaller the value of k, which means that less material is needed for a circular duct compared to a rectangular duct.

TABLE 1. PERIMETER AND AREA RELATIONS

Sl. No.	Shape	Equal Perimeters	Equal Areas
1.	Rectangle	Has the least area	Has the greatest perimeter
2.	Square	Has a greater area than rectangle	Has a perimeter less than rectangle
3.	Round	Has the greatest area	Has the least perimeter

TABLE 2. VARIANCE OF K WITH R

Sl. No.	R Value	k Value	Perimeter of a round duct is
1	1	0.88	85% a rectangular duct
2	2	0.80	80% a rectangular duct
3	3	0.72	72% a rectangular duct
4.	4	0.65	65% a rectangular duct
5.	5	0.60	60% a rectangular duct

For a typical room, the R-value is usually kept around 1. Therefore, using round ducts can result in a 15% savings in duct material. In other words, there is a 15% cost savings in material for round ducts compared to rectangular ducts.

VI. CONCLUSIONS

In summer, temperatures rise significantly in almost all parts of India. This increase in temperature creates a need for cool air and cold water. These two things are very essential during the summer season. This is attributed to the steady-state assumption of the model, which ignores the dynamic behaviour of the air conditioning equipment.

- The conclusion is that a round cross-section shape is the best for air duct systems, compared to square and rectangular shapes. It is recommended that round ducts be used in air ductwork whenever field conditions permit.

- In the convergence analysis of a duct with a circular cross-sectional area, the temporary increase in residuals during the intermediate stages corresponds to turbulence stabilization and pressure-velocity coupling adjustment. Following this transient phase, the residuals gradually decrease to values below 10^{-5}–10^{-6}. The smooth convergence without any divergence or oscillatory behaviour confirms numerical robustness and a suitable discretization scheme. Achieving such low residual levels ensures that the governing Navier-Stokes equations are satisfactorily satisfied, instilling confidence in the accuracy of the predicted velocity, pressure, and

turbulence fields within the domain shown in Figure7.

- In the convergence analysis of a duct with a rectangular cross-sectional area, the continuity residual achieves the lowest magnitude, ensuring strict mass conservation within the computational domain. The momentum residuals exhibit consistent convergence behaviour, with minor directional differences reflecting flow anisotropy within the channel. The smooth downward trend after the final transient peak indicates numerical stability and well-resolved pressure-velocity coupling. Overall, the convergence history confirms that the governing Navier-Stokes equations have been satisfactorily solved, ensuring that the simulated velocity, pressure, and turbulence fields are reliable for post-processing and physical interpretation as shown in Figure 8.

- In the convergence analysis of a duct with a square cross-sectional area, the minor differences in convergence rates among the velocity components reflect directional flow dominance within the channel. The absence of oscillations or divergence confirms numerical stability and a suitable discretization scheme. Achieving such low residual levels ensures reliable predictions of velocity distribution, shear stress, and pressure gradient. This convergence behaviour confirms that the governing Navier-Stokes equations are fully satisfied, making the simulation results suitable for the detailed flow and turbulence analysis shown in Figure 9.

REFERENCES

[1]   S. Dozer, M. Salim, Data centre air management metrics-practical approach, in: 12th IEEE Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems, 2010, Las Vegas, NV, USA, 2010, https://doi.org/10.1109/itherm.2010.5501366. June 2-5.

[2]   J. Niemann, K. Brown, V. Avelar, Impact of hot and cold aisle containment on data center temperature and efficiency, Schneider Electric Data Center Science Center White Paper 135 (2011) 1–14.

[3]   S. Bodhe, B. Sammakia, M.K. Iyengar, R. Schmidt, Guidelines on managing under floor blockages for improved data center performance, in: IMECE2006–13711 Proceedings of IMECE 2006 ASME International Mechanical Engineering Congress and Exposition, 2006, https://doi.org/10.1115/imece2006-13711. Chicago, IL, USA, November 5–10.

[4]   H. Alissa, S. Alkharabsheh, S. Bhopte, B. Sammakia, Numerical investigation of underfloor obstructions in open-contained data center with fan curves, in: ITHERM, 2014, https://doi.org/10.1109/itherm.2014.6892359. Orlando, FL, USA, May 27-30.

[5]   S.W. Ham, J.W. Jeong, Impact of aisle containment on energy performance of a data center when using an integrated water-side economizer, Appl. Therm. Eng. 105 (2016) 372–384, https://doi.org/10.1016/j.applthermaleng.2015.05.069

[6]   Y. Zheng, Z. Li, X. Liu, Z. Tong, R. Tu, retrofit of air-conditioning system in data center using separate heat pipe system, in: Proceedings of the 8th International Symposium on Heating, Ventilation and Air Conditioning, 2013, https://doi.org/10.1007/978-3-642-395819_67. Xi'an, China, October 19-21.

[7]   F. Wang, Y. Huang, B.Y. Prasetyo, Energy-efficient improvement approaches through numerical simulation and field measurement for a data center, Energies 12 (14) (2019) 2757, https://doi.org/10.3390/en12142757

[8]   F.J. Wang, C.M. Lai, Y.S. Huang, J.S. Huang, Alternative layouts for air distribution improvement of a computing data center, Adv. Mater. Res. 677 (2013) 282–285. https://doi.org/10.4028/www.scientific.net/amr.677.282.

[9]   X. Meng, J. Zhou, X. Zhang, Z. Luo, H. Gong, T. Gan, Optimization of the thermal environment of a small-scale data center in China, Energy 196 (2020), 117080

[10]   H. Meng, Z. Zhang, Numerical and experimental investigations on the thermal performance of a data center, Appl. Therm. Eng. 180 (2020), 115759, https://doi.org/10.1016/j.applthermaleng.2020.115759.

[11]   H. Tian, H. Liang, Z. Li, A new mathematical model for multi-scale thermal management of data centers using entransy theory, Build. Simulat. 12 (2019) 323–336, https://doi.org/10.1007/s12273-018-0479-z.

[12]   I. Turkmen, C.A. Mercan, H.S. Erden, Experimental and computational investigations of the

thermal environment in a small operational data center for potential energy efficiency improvements, J. Electron. Packag. 142 (3) (2020), 031116, https://doi.org/10.1115/1.4047845.

[13] E. Oro, A. Garcia, J. Salom, Experimental and numerical analysis of the air management in a data centre in Spain, Energy Build. 116 (2016) 553–561, https:// doi.org/10.1016/j.enbuild.2016.01.037.

[14] W.L. Lee, Hua Chen, F.W.H. Yik, Modeling the performance characteristics of watercooled air-conditioners, Energy and Buildings 40 (2008) 1456–1465.

[15] Hua Chen, W.L. Lee, F.W.H. Yik, applying water cooled air conditioners in residential buildings in Hong Kong, Energy Conversion and Management 49 (2008) 1416–1423

[16] J. Koomey, Growth in Data Center Electricity Use 2005 to 2010, A Report by Analytical Press, Completed at the Request of the New York Times, vol. 161, 2011

[17] Green digital sector. https://digital-strategy.ec.europa.eu/en/policies/green-digital

[18] ASHRAE, Special Publication, Thermal Guidelines for Data Processing Environments, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Atlanta, GA, USA, 2004.

[19] X. Xiong, P.S. Lee, Vortex-enhanced thermal environment for air-cooled data center: an experimental and numerical study, Energy Build. 250 (2021), 111287, https://doi.org/10.1016/j.enbuild.2021.111287.

[20] D.C. Wilcox, Formulation of the k-ω turbulence model revisited, AIAA J. 46 (11) (2008) 2823–2838, https://doi.org/10.2514/6.2007-1408. DOI:10.2514/1.36541.

[21] X. Han, W. Tian, J. VanGilder, W. Zuo, C. Faulkner, An open-source fast fluid dynamics model for data centre thermal management, Energy Build. 230 (2021), 110599, https://doi.org/10.1016/j.enbuild.2020.110599.

[22] D.V. Sheth, S.K. Saha, Numerical study of thermal management of data centre using porous medium approach, J. Build. Eng. 22 (2019) 200–215, https://doi. org/10.1016/j.jobe.2018.12.012.