Research Progress in Mechanical and Manufacturing Engineering Vol. 4 No. 1 (2023) 139-149

RPMME

Homepage:http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Design And Proportional Computational Assessment Rectangular Duct Air Conditioning System At Shopping Mall

NurAlif[1], Farid Sies[2*]

[1]Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2023.04.01.015
Received 15 July 2022; Accepted 26 January 2023; Available online 1 June 2023

Abstract: Ducting is essential in the HVAC system as it functions as a conduit to supply cooled air into the room. However, there are some problems with a long ducting system, especially in a big building like a shopping mall, where there are losses along the ductworks. So, suitable ducting needs to be designed for better performance and at the same time, effective in terms of cost. This study aims to develop a practical design for a rectangular duct system at a shopping mall using CFD analysis. In this study, both theoretical results based on manual calculation and numerical solutions using CFD analysis were discussed. Three methods of designing the ducting system are applied: equal friction, velocity reduction, and static regain. This study compares and discusses the result of these three methods. Based on the result, the static regains method produce a better ducting system in term of performance and cost because it produces the highest velocity at the last outlet. Therefore, an excellent ducting system can be designed for better HVAC system performance using the findings of this study.

Keywords: Ducting, HVAC, CFD, Shopping Mall

1. Introduction

Ducting is a method of air distribution that involves using a series of metal or plastic pipes to transport heated or cooled air from one location to another. A ducting system is often used for heating, ventilation, and air conditioning (HVAC) to distribute and extract air from the system. Most systems have only one set of ductworks, which are used to transport cool air in the summer and heated air in the winter, as well as air for general ventilation needs (Shah et al., 2018). The ideal route for the site installation to distribute the air from the supply and return system is determined by ducting layout (Pradeep et al., 2021). The duct transports heated or cooled air without leaking it into other areas.

*Corresponding author: farids@uthm.edu.my

However, a very long ducting system is required for shopping malls; at the same time, it will also increase the losses in the ducting system. The airflow flows through a long ducting with impediments, causing losses along the path and resulting in the critical point of a ducting, which is the ducting's end, not receiving enough airflow to cool the area it supplies adequately. The site needs enough airflow of cooled air to make sure certain places receive thermal comfort. So, this study determines the suitable method to use for long ducting. A proper ducting design is needed to supply adequate air to certain areas.

This research studies the performance of the fluid flow inside ducting system using theoretical calculation and simulation based on Computational Fluid Dynamic (CFD) analysis. This study aims to develop a suitable design for a rectangular duct system at a shopping mall using CFD analysis. The result between theoretical and simulation can be compared. This study also practices three methods of deigning ducting systems: equal friction, velocity reduction, and static regain. The result of these methods also can be compared. This research also can determine which technique is most suitable for designing the ducting system.

This research helps better understand the method of designing the ducting system and select a suitable technique for designing the duct. The performance of fluid flow also is learned in this study by simulation of fluid flow.

2. Materials and Methods

This section aims to help better understand the methods used in this case study by providing more precise guidelines and explanations or discussions for every technique used in this research.

2.1 Software used in this study

In this study, there are three software used. First, Fluent Software 2019 R2 was used to simulate fluid flow in the ducting system. The second software used was the Solidworks 2021 version for ducting design. The last software is McQuay DuctSizer to determine the dimension of ducting.

2.2 Methods

The methods used for designing ducting in this study are equal friction, velocity reduction, and static regain. This method determined the dimension of the ducting system. These three methods produced different dimensions of ducting. So, a comparison can be made between each ducting.

2.3 Equations

First, the continuity equation was used to study the flow rate and velocity in this system:

$$Q = Av \quad Eq. 1$$

Q = flowrate (CFM)

$A = area\ (m^2)$

v = velocity (m/s).

Static Pressure Regain (SPR),

$$SPR = SP = R[\left(\frac{v_1}{4000}\right)^2 - \left(\frac{v_2}{4000}\right)^2 Eq. 2$$

V_1 = velocity at previous duct section

V_2 = velocity at current duct section

R = recovery factor (0.7-0.9)

Equivalent duct diameter,

$$D = \sqrt{\left(\frac{4\ x\ area}{\pi}\right)}\ Eq. 3$$

3. Results and Discussion

There are two results based on this study. First, theoretical results are based on calculation. The second is the numerical analysis based on the simulation of CFD.

3.1 Theoretical Results

This study uses equal friction, velocity reduction, and static regain. Below is the table for the three results.

Table 3.1: Results of the equal friction method

Section	CFM (ft^3 /min)	Friction Loss in.w per 100 ft	Pa/m	Eq. D (mm)	Rectangular duct size			Duct Area, m^2	V, m/s
AB	54633.00	0.091	0.74	1643	3650	X	725	2.65	9.74
BS	48183.27	0.091	0.74						
BC	48183.27	0.091	0.74						
CR	0.00	0.091	0.74						
CD	22384.35	0.091	0.74	1168.6	2075	X	600	1.25	8.49
DE	17072.81	0.091	0.74	1054	1650	X	600	0.99	8.14
EF	15555.23	0.091	0.74	1017.3	1525	X	600	0.92	8.02
FG	13658.25	0.091	0.74	968.1	1350	X	600	0.81	7.96
GH	10623.08	0.091	0.74	879.8	1225	X	550	0.67	7.44
HI	7587.92	0.091	0.74	774.2	1025	X	500	0.51	6.99
IJ	4552.75	0.091	0.74	637.8	675	X	500	0.34	6.37
JK	3035.17	0.091	0.74	547	625	X	400	0.25	5.73
KX	1517.58	0.091	0.74	420.9	425	X	350	0.15	4.81
XL	0.00	0.091	0.74	381	350	X	350	0.12	1.20

Table 3.2: Results of the velocity reduction method

Section	CFM (ft^3 /min)	Flowrate (m^3/s)	FPM	Eq. D (mm)	Rectangular duct size			Duct Area, m^2	V, m/s
AB	54633.00	25.78	2400	1641.1	3650	X	725	2.64625	9.74
BS	48183.27	22.74	1600					0	
BC	48183.27	22.74	2400					0	
CR	0.00		1600					0	
CD	22384.35	10.56	1600	1286.30	2625	X	600	1.575	6.71
DE	17072.81	8.06	1600	1123.30	1900	X	600	1.14	7.07
EF	15555.23	7.34	1600	1072.20	1700	X	600	1.02	7.20
FG	13658.25	6.45	1600	1004.70	1475	X	600	0.885	7.28
GH	10623.08	5.01	1600	886.10	1250	X	550	0.6875	7.29
HI	7587.92	3.58	1600	748.50	975	X	500	0.4875	7.35
IJ	4552.75	2.15	1600	580.10	575	X	500	0.2875	7.47
JK	3035.17	1.43	1600	473.60	475	X	400	0.19	7.54

KX	1517.58	0.72	1600	334.90	350	X	275	0.09625	7.44
XL	0.00	0.00	1600		275	X	275	0.07563	1.86

Table 3.3: Results of the Static Regain Method

Section	CFM (ft^3 /min)	Flowrate	Velocity (m/s)	Duct Area, (in^2)	Friction loss (Inch wg/100)	Friction loss inch of wg (A)	Recovery factor	Static Pressure Regain (SPR) (inch of wg)(B)	Accept if A<B	Rectangular duct size		
AB	54633.00	25.78	17.78	2247.76	0.237	0.13	0.9	0.00	√	3650	X	650
BS	48183.27	22.74	17.78	1982.40		0.00	0.9	0.00			X	
BC	48183.27	22.74	17.78	1982.40		0.00	0.9	0.00			X	
CR	0.00	12.18	16.764	1125.77		0.00	0.9	0.08			X	
CD	22384.35	10.56	15.24	1074.45	0.270	0.07	0.9	0.11	√	1550	X	600
DE	17072.81	8.06	14.224	878.03	0.250	0.07	0.9	0.07	√	1275	X	600
EF	15555.23	7.34	13.208	861.52	0.200	0.03	0.9	0.06	√	1200	X	600
FG	13658.25	6.45	12.7	786.72	0.180	0.03	0.9	0.03	√	1150	X	600
GH	10623.08	5.01	11.684	665.10	0.180	0.05	0.9	0.05	√	1100	X	550
HI	7587.92	3.58	10.668	520.31	0.180	0.05	0.9	0.05	√	975	X	500
IJ	4552.75	2.15	9.652	345.05	0.150	0.04	0.9	0.05	√	700	X	500
JK	3035.17	1.43	9.144	242.81	0.100	0.02	0.9	0.02	√	700	X	400
KX	1517.58	0.72	8.636	128.55	0.090	0.02	0.9	0.02	√	550	X	350
XL	0	0	2.2	0	0.237	0.13	0.9	0.00	√	350	X	350

3.2 Numerical Analysis

The numerical analysis was done by using Fluent Ansys software. It indicated the result of the fluid flow in the ducting.

3.2.1 Grid Independent Test

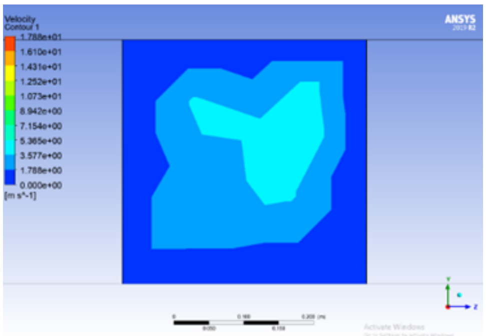
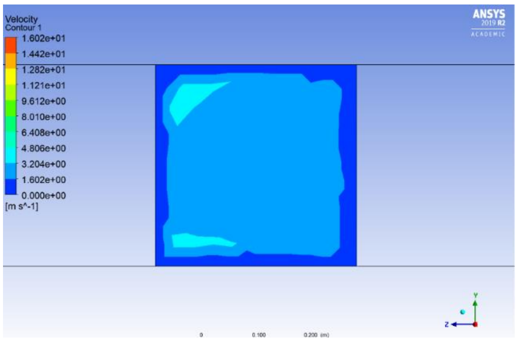
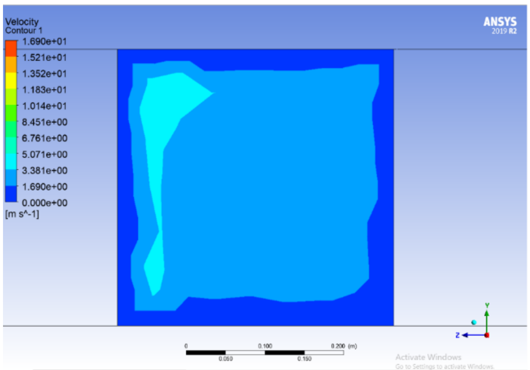
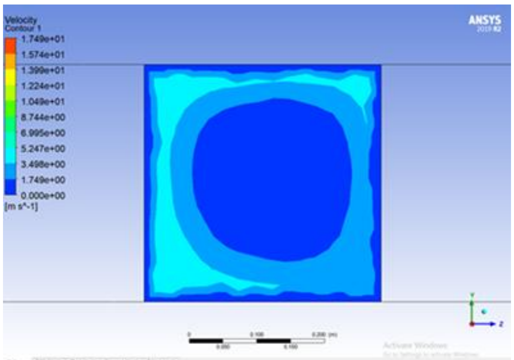
The Grid independent test was performed to evaluate the correct mesh size and number of elements that did not substantially affect the grid sensitivity of the analysis. It needs to be meshed again into a minor element size to reach the acceptable mesh size level of adaptive. The element size has been re-meshing focused on the entire ductwork as it is an essential element that needs to be simulated and the performance result analysed. Table 3.4 shows the mesh analysis for different size elements.

Table 3.4: Shows the mesh analysis for different size element

Element Size (mm)	Nodes	Element	Skewness	
			Maximum	Average
100	181312	941928	0.98987	0.26726
50	454793	2379846	0.9796	0.26832
40	664766	3469642	0.99679	0.26774
25	1773978	9410865	0.99556	0.2666

According to the grid-independent test, the 25 mm element size was chosen due to the most petite average skewness compared to the others. It was therefore considered to be the most reliable and appropriate element size.

Table 3.5: Velocity contour at the last outlet

100mm element size  Average Velocity: 3.8m/s	40mm element size  Average Velocity: 2.57m/s
50mm element size  Average Velocity: 2.87m/s	25mm element size  Average velocity: 1.8m/s

Based on the velocity outlet, 25mm size of element in meshing is the most suitable velocity for the result because it is the nearest to the outcome based on calculation. The velocity based on simulation is 1.8m/s for simulation with a 25mm element size, while the result for calculation is about 1.2m/s. Based on the contour maps, it can be seen that the velocity throughout the final outlet is the least and correlatedly produce the least final velocity, which is 1.8m/s.

There is a difference in results based on different size elements of meshing. For example, Figure 3.1 shows the difference between the velocity at the last outlet for different element sizes and the result based on calculation.

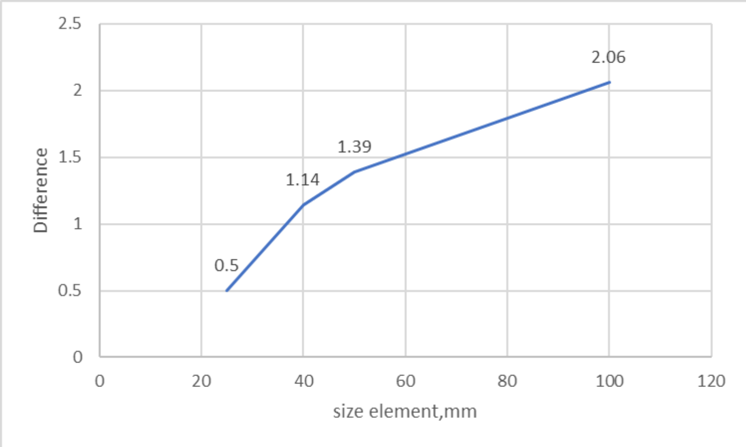


Figure 3.1: the graph for the size of the element against the difference between the result

Based on this graph, it can be concluded that the smaller size of the element for meshing, the more accurate result of the final velocity. It requires less than 25mm element size to get the result with zero difference. However, decreasing the element size in meshing will increase the number of nodes and elements. Therefore, more time was taken for the simulation to complete.

3.3 Difference Between the Three Methods

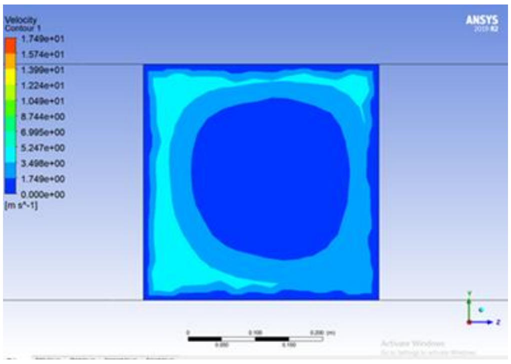
In this part, numerical solutions can compare the results of the three methods based on simulation. All the simulations produced different final velocities at the last outlet. The final velocity can be observed by analysing the velocity contour maps at the final outlet. The researcher also can detect the velocity streamline of the ductwork to investigate the duct flow in the system.

3.3.1 Velocity at the Last Outlet

`Contour maps can be produced with the final post-processing fluent software. The velocity contour is the velocity distribution at the last outlet. This result showed which method produced the highest velocity at the last outlet. Table 3.6 shows the difference in velocity at the last outlet for the three methods.

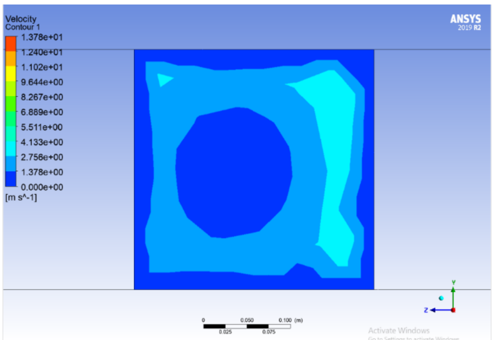
Table 3.6: Difference in velocity at the last outlet for the three method

Equal Friction Method



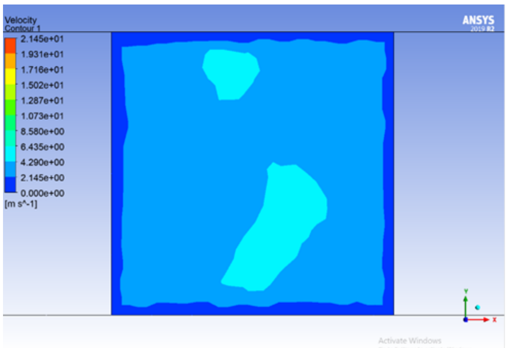
Average Velocity: 1.8m/s

Velocity Reduction Method



Average Velocity: 2.35m/s

Static Regain Method



Average Velocity: 3.4m/s

Static-regain method produces the highest velocity at the last outlet, while the equal-friction method produces the least. It is because static regain has 4.2m/s velocity at the last outlet.

3.3.2 Velocity Streamline of The Ductworks

A streamline is a direction through the fluid domain that a particle with zero mass can follow. Streamlines start on a given locator at each node, in which case the streamline begins at the inlet.

Another way to visualize the behaviour of airflow required for this study to be analysed is to streamline. Figure 3.2, figure 3.3, and figure 3.4 shows the difference in velocity streamlines for the three methods.

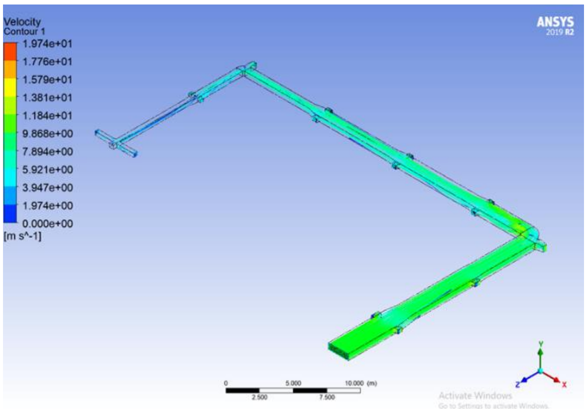


Figure 3.2: Velocity Streamline for Equal Friction Method

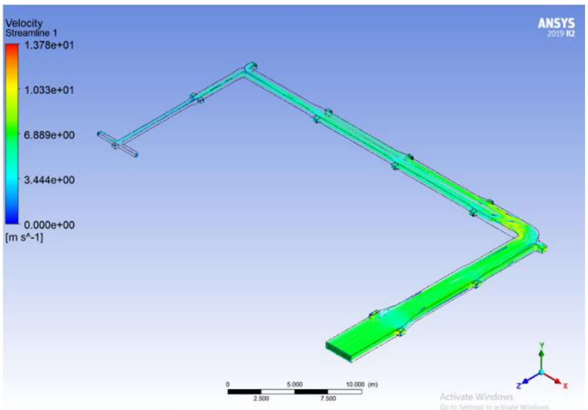


Figure 3.3: Velocity Streamline for Velocity Reduction Method

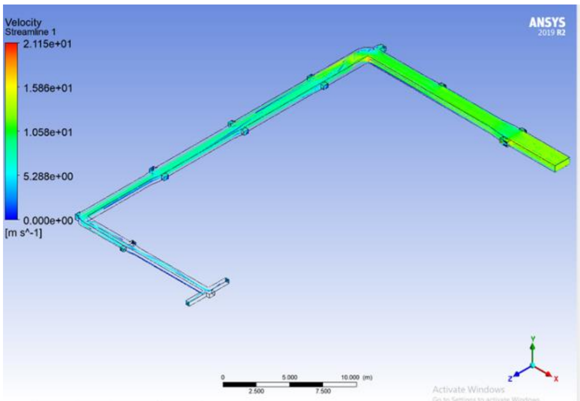


Figure 3.4: Velocity Streamline for Static Regain Method

Based on the velocity streamline of the ductworks, we can state that the static regain method has the highest velocity compared to the other two methods. On the other hand, the equal-friction method produces the least velocity in the ductworks.

3.4 Discussion

This section discusses the overall results obtained from calculating the three methods and the simulation of ducting using fluent software. Based on the simulation and calculation, it can be stated that the static regain method is the most reliable method compared to the equal friction and velocity reduction methods.

The static regain method produces the highest velocity at the last outlet of the ductwork and the most increased velocity flow in the ducting system. A good ducting system should have the highest possible velocity for fluid flow to supply the cool air to the surrounding. Therefore, the static regain method is recommended for a sizeable ducting system and high-velocity flow based on ducting notes. The static regain method has the most complicated procedure as we have to identify the static pressure regain (SPR) in the calculation. The static regains method is also economically suitable because, based on the cost estimation, the static regain method provides the ducting system with the lowest price. However, there is still some difference in terms of results for calculation and simulation. Table 3.7 shows the difference in results for theoretical and simulation.

Table 3.7: The percentage difference for each method

Method	Velocity for Theoretical Result (m/s)	Velocity the for numerical result (m/s)	Percentage difference (%)
Equal Friction	1.2	1.8	50
Velocity Reduction	1.8	2.35	30.5
Static Regain	2.2	3.4	54

Table 3.7 shows that the difference for velocity reduction is the lowest, and the difference for static regain is the highest. This result can be reduced by increasing the number of nodes and elements in meshing. The highest number of nodes and elements in meshing will improve the accuracy of the result. However, it is impossible to do the simulation with the current computer specification where it only has 16Gb RAM. This specification is not sufficient to do simulation for large geometry as it will take too much time for the whole simulation. The 25mm element size of meshing used in this study takes up to 5 days to complete the simulation. It is predicted that to get suitable meshing for the accurate result needs more than a week using the current computer specification. Higher specification of computer will reduce the time taken for the simulation. A supercomputer with 64Gb of RAM is very good for simulation with a large geometry-like ducting system.

4. Conclusion

Throughout this analysis, the approach of computational fluid dynamics (CFD) is used to evaluate the ducting system's performance analysis numerically. The steady procedure was used to calculate the quantity in the simulation research. This solution relied on K-omega turbulence, which can be expected well below the boundary wall. However, the simulation analysis was limited because it required a very high computer specification. A large model will need a small element size in meshing, increasing the number of nodes and elements. As a result of this issue, the project's time limit would take a long time to complete.

Nevertheless, the numerical simulation results thoroughly understood fluid flow in the ductwork. This study demonstrated that the Ansys Fluent CFD modeling helped process a more comprehensive numerical study of the ducting system. This report aims to use CFD analysis to create a suitable design for a rectangular duct system at a shopping mall. The simulation data show that the static regain method produces the best design in terms of performance and economy.

Based on this analysis, the ducting system designed a recommended computer specification that included an i7 processor, 32 gigabytes of RAM, a graphics card with a capacity of up to 4 gigabytes, and an internal hard disc with a total of 1 terabyte. It would shorten the simulation time required to support large-scale model geometry that requires meshing in small element sizes. Propose a better design or geometry for the ducting system where it is more appropriate and efficient. Because ducting geometry is the medium used to generate airflow, having an excellent ducting geometry will result in a good airflow result. According to previous research, axial spacing is also important in developing good airflow. The design may need a combination of rectangular and circular ducting for the whole structure. It is because rectangular ducting has better performance and reduces losses. However, we cannot manipulate the height and width of circular ducting, so it will be hard to fit in the ceiling where space is limited. So, the circular ducting can be put at the ceiling with enough space and rectangular ducting for limited space. The whole ducting system can be separated into several sections for simulation and to reduce the time taken for the simulation

Acknowledgment

This research was supported by Universiti Tun Hussein Onn Malaysia (UTHM) through TIER 1 (vot H923).

References

[1] Altalmas, T. M., Ahmad, S., Aula, A., Akmeliawati, R., & Sidek, S. N. (2013). Mechanical design and simulation of two-wheeled wheelchair using solidworks. *IOP Conference Series: Materials Science and Engineering*, *53*(1). https://doi.org/10.1088/1757-899X/53/1/012042

[2] Basavaraja. Kt. (2019). Design of an Air Conditioning System for a Commercial Building using Air Handling Unit. *International Journal of Trend in Scientific Research and Development*, *4*(1), 192–195. https://www.ijtsrd.com/papers/ijtsrd29454.pdf%0Ahttps://www.ijtsrd.com/engineering/mechanical-engineering/29454/design-of-an-air-conditioning-system-for-a-commercial-building-using-air-handling-unit/basavaraja-kt

[3] Gao, R., Wang, M., Guo, W., Zhang, S., Wen, S., Wu, D., Zhou, H., & Li, A. (2021). Study of the shape optimization of a variable diameter in a ventilation and air-conditioning duct based on the Boltzmann function. *Journal of Building Engineering*, *43*(March), 102833. https://doi.org/10.1016/j.jobe.2021.102833

[4] Du, X., Zhang, Y., & Lv, Z. (2020). Investigations and analysis of indoor environment quality of green and conventional shopping mall buildings based on customers' perception. *Building and Environment*, *177*(February), 106851. https://doi.org/10.1016/j.buildenv.2020.106851

[5] Inthavong, K., Wong, E., Tu, J., & Singh, N. (2021). *Clinical and Biomedical Engineering in the Human Nose*. https://doi.org/10.1007/978-981-15-6716-2_1

[6] Jing, W., Yu, J., Luo, W., Li, C., & Liu, X. Y. (2021). Energy-saving diagnosis model of central air-conditioning refrigeration system in large shopping mall. *Energy Reports*, *7*, 4035–4046. https://doi.org/10.1016/j.egyr.2021.06.083

[7] Khakre, V. V, Prof, A., Engineering, M., & Pusad, B. N. C. O. E. (2017). Design of Duct for a Three Storey Retail Shop. *International Research Journal of Engineering and Technology(IRJET)*, *4*(2), 329–332. https://irjet.net/archives/V4/i2/IRJET-V4I265.pdf

[8] Kim, S. Y., Shin, D. H., Kim, C. S., Park, G. C., & Cho, H. K. (2021). Computational fluid dynamics analysis of buoyancy-aided turbulent mixed convection inside a heated vertical rectangular duct. *Progress in Nuclear Energy*, *137*. https://doi.org/10.1016/j.pnucene.2021.103766

[9] Legg, R. (2017a). Ducted Air Systems. In *Air Conditioning System Design*. https://doi.org/10.1016/b978-0-08-101123-2.00014-5

[10] Legg, R. (2017b). Fluid Flow: General Principles. In *Air Conditioning System Design*. https://doi.org/10.1016/b978-0-08-101123-2.00013-3

[11] Lun, Y. H. V., & Tung, S. L. D. (2020). Air handling unit. *Green Energy and Technology*, *January*, 51–64. https://doi.org/10.1007/978-3-030-31387-6_4

[12] Malanichev, I., & Akhmadiev, F. (2020). Pressure loss reduction in ventilation ducts by shape optimization of the removable profiled components. *IOP Conference Series: Materials Science and Engineering*, *890*(1). https://doi.org/10.1088/1757-899X/890/1/012154

[13] Noussan, M., Carioni, G., Degiorgis, L., Jarre, M., & Tronville, P. (2017). Operational performance of an Air Handling Unit: Insights from a data analysis. *Energy Procedia*, *134*, 386–393. https://doi.org/10.1016/j.egypro.2017.09.579

[14] Pradeep, A. D., Vivekanandhan, R., Rameshkumar, T., & Sangaravadivel, P. (2021). Design of Hvac System for Beverage Industry. *IOP Conference Series: Materials Science and Engineering*, *1059*(1). https://doi.org/10.1088/1757-899X/1059/1/012063

[15] Rosly, N., Mohd, S., Zulkafli, M. F., Abdul Ghafir, M. F., Shamsuddin, S. S., & Wan Muhammad, W. N. A. (2017). Flow Simulation of Modified Duct System Wind Turbines Installed on Vehicle. *Journal of Physics: Conference Series*, *914*(1). https://doi.org/10.1088/1742-6596/914/1/012006

[16] Shah, H., Patel, P., Shah, H. A., Patel, P. T., Gaywala, K. B., Professor, A., Engineering Department, A., & Gopalbhai Patel, C. (2018). *Issue 7 www.jetir.org (ISSN-2349-5162) JETIRC006370 Journal of Emerging Technologies and Innovative Research (JETIR) www.jetir* (Vol. 5). www.jetir.org